

# ECLIPSE

Version 2014.1



# ECLIPSE

ECLIPSE reservoir simulation software

Reference Manual

**Schlumberger**

### **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

<b>1</b>	<b>New Developments .....</b>	<b>1</b>
	<b>Licensing changes .....</b>	<b>1</b>
	<b>New facilities .....</b>	<b>2</b>
	ECLIPSE 100 .....	2
	ECLIPSE 300 .....	4
	<b>Behavioral changes .....</b>	<b>7</b>
	ECLIPSE 100 .....	7
	ECLIPSE 300 .....	10
	<b>New keywords .....</b>	<b>13</b>
	ECLIPSE 100 .....	13
	ECLIPSE 300 .....	15
	<b>Altered keywords .....</b>	<b>16</b>
	ECLIPSE 100 .....	16
	ECLIPSE 300 .....	20
<b>2</b>	<b>Data File Overview .....</b>	<b>24</b>
	<b>Data file structure .....</b>	<b>26</b>
	Keyword data .....	26
	Comments .....	27
	<b>Global keywords .....</b>	<b>27</b>
	<b>ECLIPSE 300 error handling .....</b>	<b>29</b>
	<b>Example data file .....</b>	<b>30</b>
	<b>RUNSPEC section .....</b>	<b>31</b>
	Alphabetic list of RUNSPEC keywords .....	32
	<b>GRID section .....</b>	<b>38</b>
	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
<b>EDIT section .....</b>	<b>63</b>
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
<b>PROPS section .....</b>	<b>68</b>
Saturation functions .....	69
<i>Two dimensional models for saturation functions (for three phase simulations) .....</i>	<i>70</i>
<i>Stability issues .....</i>	<i>71</i>
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
<i>Alkaline option .....</i>	<i>79</i>
<i>API tracking (ECLIPSE 100) .....</i>	<i>80</i>
<i>Black Oil fluid PVT property data .....</i>	<i>80</i>
<i>Brine option .....</i>	<i>81</i>
<i>Carbon Dioxide solution in aqueous phase (ECLIPSE 300) .....</i>	<i>81</i>
<i>Carter-Tracy aquifer data .....</i>	<i>82</i>
<i>Chemical reaction keywords (ECLIPSE 300) .....</i>	<i>82</i>
<i>Coal bed methane (ECLIPSE 100) .....</i>	<i>82</i>

Coal bed methane (ECLIPSE 300) .....	83
Compositional EoS and PVT property data (ECLIPSE 300) .....	83
Dual porosity (ECLIPSE 100) .....	85
End point scaling .....	86
Environmental tracers (ECLIPSE 100) .....	88
Foam Model .....	89
GASWAT option (ECLIPSE 300) .....	89
Geomechanics (ECLIPSE 300) .....	90
GI pseudo-compositional model (ECLIPSE 100) .....	90
Gradient option (ECLIPSE 100 special option) .....	90
Initial composition (ECLIPSE 300) .....	91
K Values (ECLIPSE 300) .....	91
Low-Salinity option (ECLIPSE 100) .....	91
Miscible flood option (ECLIPSE 100) .....	91
Molecular diffusion option .....	92
NGLs (ECLIPSE 300) .....	93
Non-Darcy Flow (ECLIPSE 300) .....	93
Output .....	93
Polymer Flood Model .....	93
Relative permeability and capillary pressure data .....	94
Rivers (ECLIPSE 100) .....	97
Rock compressibility and compaction data .....	97
Solid option (ECLIPSE 300) .....	99
Solvent Model (ECLIPSE 100) .....	99
Surface tension effects (ECLIPSE 300) .....	100
Surfactant option .....	100
Temperature option .....	101
Thermal option (ECLIPSE 300) .....	101
Tracer tracking .....	103
Vertical equilibrium (ECLIPSE 100) .....	104
Water component option (ECLIPSE 300) .....	104
ODD3P three-phase relative permeability and capillary pressure hysteresis option (ECLIPSE 300) .....	105
Operational keywords .....	105
Alphabetic list of keywords .....	106
<b>REGIONS section .....</b>	<b>121</b>
REGIONS section keywords .....	121
Output .....	122
Commonly used keywords .....	122
Directional relative permeability .....	122
Dual porosity option (ECLIPSE 100 only) .....	122
Geomechanics option (ECLIPSE 300) .....	122
Hysteresis option (excluding ODD3P) .....	122
WAG hysteresis option .....	123
Other options .....	123

<i>ODD3P three-phase relative permeability and capillary pressure hysteresis option (ECLIPSE 300)</i> .....	124
<i>Operational</i> .....	124
Alphabetical list of keywords .....	125
<b>SOLUTION section</b> .....	<b>128</b>
Equilibration - the EQUIL keyword .....	129
Enumeration .....	129
Thermal enumeration with the NEI keyword .....	130
Flexible restarts .....	130
Initial state output .....	130
Analytic aquifers .....	131
Tracers, brine and API tracking .....	131
Miscellaneous options .....	132
Initial fluid in place .....	132
SOLUTION section keywords .....	132
<i>Output</i> .....	132
<i>Equilibration</i> .....	133
<i>Restarting</i> .....	134
<i>Enumeration</i> .....	134
<i>Aquifer data</i> .....	135
<i>Temperature option (ECLIPSE 100)</i> .....	136
<i>Other options</i> .....	136
<i>Operational keywords</i> .....	137
Dynamic Regions Facility .....	138
Rivers (ECLIPSE 100) .....	138
Tuning (ECLIPSE 300) .....	138
ODD3P option (ECLIPSE 300) .....	138
Alphabetic list of keywords .....	138
<b>SUMMARY section</b> .....	<b>142</b>
Keyword data format .....	142
<i>Format of SUMMARY keywords beginning with A</i> .....	143
<i>Format of SUMMARY keywords beginning with B</i> .....	143
<i>Format of SUMMARY keywords beginning with C</i> .....	144
<i>Format of SUMMARY keywords beginning with F</i> .....	145
<i>Format of SUMMARY keywords beginning with G</i> .....	145
<i>Format of SUMMARY keywords beginning with LB</i> .....	146
<i>Format of SUMMARY keywords beginning with LC</i> .....	146
<i>Format of SUMMARY keywords beginning with LW</i> .....	147
<i>Format of SUMMARY keywords beginning with R</i> .....	147
<i>Format of SUMMARY keywords beginning with S</i> .....	149
<i>Format of SUMMARY keywords beginning with W</i> .....	150
<i>Format of SUMMARY keywords that end in H</i> .....	150
<i>Format of SUMMARY keywords with a component index</i> .....	150
<i>Keywords for special options</i> .....	152
<i>Maximum number of keywords</i> .....	152

<i>Example SUMMARY section</i> .....	152
<i>Summary file names</i> .....	153
Well and group flow quantities .....	153
<i>Miscellaneous well and group quantities</i> .....	170
<i>Secondary group and well quantities (ECLIPSE 300)</i> .....	176
<i>Well list quantities</i> .....	177
Grid block and region quantities .....	178
<i>Equivalent directional keywords for relative permeabilities (ECLIPSE 100)</i> .....	189
Processed scaled end-points .....	190
Reservoir volumes .....	191
Oil recovery efficiencies .....	192
Oil recovery mechanism .....	192
Recovery plant options (ECLIPSE 300 only) .....	193
Analytic aquifer quantities .....	193
Numerical aquifer quantities .....	194
Tracer and API tracking quantities .....	194
Brine option quantities .....	195
Temperature option quantities .....	197
Environmental tracers .....	197
Coal Bed Methane Model (ECLIPSE 100) .....	199
Foam Model .....	199
Gas Field Operations Model .....	200
<i>Simulator performance keyword</i> .....	201
Gas lift optimization facility .....	201
Gas calorific value control option .....	202
Gi pseudo-compositional model .....	202
Local grid refinement and coarsening .....	203
<i>Operational keywords</i> .....	203
<i>Plotting local timestep results in GRAF</i> .....	203
Multisegment wells .....	204
Rivers .....	206
Network option .....	208
Polymer Flood Model .....	209
Pseudo Steady-State (ECLIPSE 100 only) .....	212
Reservoir coupling facility (ECLIPSE 100 only) .....	213
Solvent Model .....	213
Surfactant Model .....	214
Alkaline model .....	215
Wellbore friction model .....	215
Conductive Faults .....	216
User defined quantities .....	216
ODD3P three-phase relative permeability and capillary pressure hysteresis model ...	216
Asphaltene model .....	219
Simulator performance keywords .....	219
Other keywords .....	223

Deprecated keywords .....	224
Deleted keywords .....	225
<b>SCHEDULE section .....</b>	<b>226</b>
Define wells, groups and well connections .....	227
<i>Define well connections and set connection properties</i> .....	228
<i>Define multisegment wells</i> .....	229
<i>Define separators (ECLIPSE 300)</i> .....	231
Well controls and limits .....	231
<i>Production well control data</i> .....	232
<i>Injection well control data</i> .....	232
<i>Vertical flow performance keywords</i> .....	234
<i>Well economic limits and coning control</i> .....	235
Group controls and limits .....	235
<i>Group production control by guide rate</i> .....	235
<i>Group production control by prioritization</i> .....	236
<i>Group economic limits and coning control</i> .....	236
<i>Group injection control</i> .....	236
<i>Group efficiency factor</i> .....	237
<i>Sales gas control (ECLIPSE 100)</i> .....	237
<i>Satellite group flows</i> .....	238
<i>Sustainable capacity test</i> .....	238
Automatic drilling and workover controls .....	238
Action keyword facility .....	239
Periodic testing and cycling wells .....	239
<i>Artificial lift operations (ECLIPSE 100)</i> .....	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
<i>User defined values</i> .....	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
<b>OPTIMIZE section .....</b>	<b>247</b>
OPTIMIZE section keywords .....	248

<b>3</b>	<b>Keywords .....</b>	<b>249</b>
	<b>A .....</b>	<b>249</b>
	ACF .....	250
	<i>Examples</i> .....	250
	ACFDET .....	251
	<i>Examples</i> .....	251
	ACFS .....	252
	ACTCO2S .....	253
	ACTDIMS .....	254
	ACTION .....	255
	ACTIONG .....	258
	ACTIONR .....	263
	ACTIONS .....	266
	ACTIONW .....	270
	ACTIONX .....	275
	<i>Quantity types and definitions</i> .....	277
	<i>Notes</i> .....	281
	<i>Examples</i> .....	282
	ACTNUM .....	285
	ACTPARAM .....	287
	ADD .....	288
	<i>Legal arrays</i> .....	289
	<i>Notes</i> .....	290
	<i>Examples</i> .....	291
	ADDREG .....	292
	<i>Legal arrays</i> .....	293
	<i>Notes</i> .....	294
	<i>Additional notes</i> .....	295
	<i>Examples</i> .....	295
	ADDZCORN .....	297
	<i>Examples</i> .....	298
	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

---

ALPHAI .....	318
ALPHANUD .....	319
ALPHANUI .....	320
ALPHANUM .....	321
ALPOLADS .....	322
ALSURFAD .....	323
ALSURFST .....	324
AMALGAM .....	325
AMF .....	327
AMFVD .....	328
API .....	329
APIGROUP .....	330
APILIM .....	331
APIVD .....	333
AQANCONL .....	334
AQANNC .....	337
AQANTRC .....	338
<i>Examples</i> .....	338
AQCOEF .....	340
AQSTREAM .....	341
AQSTREAW .....	342
AQUALIST .....	343
AQUANCON .....	344
AQUCHGAS .....	347
AQUCHWAT .....	349
AQUCON .....	353
<i>Examples</i> .....	355
AQUCT .....	357
AQUCWFAC .....	360
AQUDIMS .....	361
AQUFET .....	363
AQUFETP .....	365
AQUFLUX .....	368
AQUNNC .....	370
AQUNUM .....	372
AQUTAB .....	374
ASPDEPO .....	375
ASPFLOC .....	376
ASPFLRT .....	377
ASPHALTE .....	378
<i>Examples</i> .....	378
ASPKDAM .....	380
<i>Examples</i> .....	380
ASPKROW .....	381
ASPLCRT .....	382

---

ASPP1P .....	383
<i>Examples</i> .....	383
ASPP2P .....	384
<i>Examples</i> .....	384
ASPPW2D .....	385
<i>Example</i> .....	385
ASPREWG .....	386
<i>Examples</i> .....	386
ASPVISO .....	388
<i>Examples</i> .....	388
ASPWETF .....	389
AUTOCOAR .....	390
<i>Examples</i> .....	390
AUTOREF .....	394
AUTOSAVE .....	395
<b>B</b> .....	<b>396</b>
BAKER1 .....	397
<i>Examples</i> .....	397
BAKER2 .....	399
<i>Examples</i> .....	399
BDENSITY .....	401
BGGI .....	402
BIC .....	403
<i>Examples</i> .....	403
BICAQ1 .....	404
BICAQ2 .....	405
BICAQ3 .....	406
BICS .....	407
BIGMODEL .....	408
BIOTC .....	409
BIOTCTAB .....	410
BLACKOIL .....	412
BOGI .....	413
BOUNDARY .....	414
BOX .....	416
BPARA .....	417
BPIDIMS .....	418
BRANPROP .....	419
BRINE .....	421
BTOBALFA .....	422
BTOBALFV .....	423
<b>C</b> .....	<b>424</b>
CALTRAC .....	425
CALVAL .....	426
CALVALR .....	427

CARFIN .....	428
<i>Examples</i> .....	429
CART .....	430
CATYPE .....	431
CBMOPTS .....	432
CCTYPE .....	434
CCTYPES .....	435
CECON .....	436
CECONT .....	439
<i>Examples</i> .....	441
CFLLIMIT .....	442
CGDTYPE .....	444
CGVTYPE .....	445
CHANDIMS .....	446
CIRCLE .....	448
CNAMES .....	449
CO2SOL .....	450
CO2STORE .....	451
COAL .....	452
COALADS .....	453
COALNUM .....	455
COALNUMR .....	456
<i>Examples</i> .....	456
COALPP .....	458
COARSEN .....	460
CODTYPE .....	462
COLLAPSE .....	463
<i>Examples</i> .....	463
COLUMNS .....	464
COMPAGH .....	465
COMPAGHL .....	467
COMPDAT .....	469
COMPDATL .....	474
COMPDATM .....	479
COMPFLSH .....	480
<i>Example</i> .....	481
<i>Treatment of flash transformation ratios in ECLIPSE</i> .....	481
COMPIMB .....	484
COMPINJK .....	486
COMPKRI .....	488
COMPKRIL .....	490
COMPLMPL .....	492
COMPLUMP .....	495
COMPMBIL .....	497
COMPMOBI .....	499

COMPOFF .....	500
COMPORD .....	501
COMPRIV .....	503
COMPRP .....	504
<i>Example</i> .....	505
<i>The saturation rescaling equations</i> .....	505
<i>Representing the effects of partial penetration</i> .....	508
COMPRPL .....	510
COMPS .....	512
COMPSEGL .....	513
COMPSEGS .....	515
COMPSOL .....	519
COMPVD .....	520
COMPVE .....	521
COMPVEL .....	525
COMPW .....	529
CONDFLTS .....	530
CONDFRAC .....	532
COORD .....	534
COORDSYS .....	536
COPY .....	538
<i>Legal arrays</i> .....	539
<i>Notes</i> .....	540
<i>Examples</i> .....	540
COPYBOX .....	542
<i>Examples</i> .....	543
COPYREG .....	544
<i>Legal arrays</i> .....	544
<i>Notes</i> .....	546
<i>Examples</i> .....	546
COVTYPE .....	547
CPR .....	548
CREF .....	549
CREFW .....	550
CREFWS .....	551
CRITPERM .....	552
CRNDENS .....	553
CSKIN .....	554
CVCRIT .....	556
CVTYPE .....	560
CVTYPES .....	561
CWTYPE .....	562
<i>Examples</i> .....	562
<b>D</b> .....	<b>563</b>
DATE .....	564

<i>Examples</i> .....	564
DATES .....	565
DATUM .....	566
DATUMR .....	567
DATUMRX .....	568
DBGODD3P .....	569
DCQDEFN .....	570
DEADOIL .....	571
DEBUG .....	572
DEBUG3 .....	576
DELAYACT .....	586
DENAQA .....	590
<i>Examples</i> .....	590
DENSITY .....	591
<i>Examples</i> .....	591
DEPTH .....	592
DEPTHTAB .....	593
DETAILMF .....	594
DETAILVD .....	595
DGRDT .....	597
DGRID .....	598
DIAGDISP .....	600
DIFFAGAS .....	601
DIFFAOIL .....	602
DIFFC .....	603
<i>Examples</i> .....	604
DIFFCBM .....	605
DIFFCGAS .....	606
DIFFCGO .....	607
DIFFCGW .....	608
DIFFCOAL .....	609
DIFFCOG .....	610
DIFFCOIL .....	611
DIFFCWAT .....	612
DIFFCWG .....	613
DIFFDP .....	614
DIFFMMF .....	615
DIFFMR .....	616
DIFFMR- .....	617
DIFFMTHT .....	618
DIFFMTH- .....	619
DIFFMX .....	620
DIFFMX- .....	621
DIFFMY .....	622
DIFFMY- .....	623

DIFFMZ .....	624
DIFFMZ- .....	625
DIFFR .....	626
DIFFTGAS .....	627
DIFFTHT .....	628
DIFFTOIL .....	629
DIFFUSE .....	630
DIFFX .....	631
DIFFY .....	632
DIFFZ .....	633
DIMENS .....	634
DIMPES .....	635
<i>Examples</i> .....	635
DIMPLICIT .....	636
DISGAS .....	637
DISPBC .....	638
DISPBCL .....	640
DISPDIMS .....	642
DISPERSE .....	643
DNGL .....	645
DOMAIN .....	646
DOMAINS .....	647
DPCDT .....	648
DPGRID .....	649
DPKRMOD .....	650
DPNUM .....	652
DR .....	653
DRAINAGE .....	654
DREF .....	655
DREFS .....	656
DREFT .....	657
DREFTS .....	658
DREFW .....	659
DREFWS .....	660
DRILPRI .....	661
<i>Examples</i> .....	662
DRSDT .....	664
DRV .....	665
DRVDT .....	666
DSPDEINT .....	667
DSTNUM .....	668
DTHETA .....	669
DTHETAV .....	670
DUALPERM .....	671
DUALPORO .....	672

DUMPCUPL .....	673
DUMPFLUX .....	674
DX .....	675
<i>Examples</i> .....	675
DXV .....	676
DY .....	677
<i>Examples</i> .....	677
DYNAMICR .....	678
<i>Each subsequent record</i> .....	678
<i>Notes</i> .....	680
<i>Examples</i> .....	681
DYNRDIMS .....	684
DYV .....	685
DZ .....	686
<i>Examples</i> .....	686
DZMATRIX .....	687
DZMTRX .....	688
DZMTRXV .....	689
DZNET .....	690
DZV .....	691
<b>E</b> .....	<b>692</b>
E100NOSI .....	693
ECHO .....	694
ECLMC .....	695
EDITNNC .....	696
EDITNNCR .....	698
EHYSTR .....	700
<i>Notes</i> .....	703
<i>Example</i> .....	704
EHYSTRR .....	705
END .....	706
ENDACTIO .....	707
ENDBOX .....	708
ENDDYN .....	709
ENDFIN .....	710
ENDINC .....	711
ENDNUM .....	712
<i>Examples</i> .....	712
ENDSCALE .....	713
ENDSKIP .....	715
ENKRVC .....	716
ENKRVD .....	717
<i>Examples</i> .....	718
ENKRVT .....	720
ENPCVC .....	721

ENPCVD .....	722
ENPCVT .....	723
ENPTVC .....	724
ENPTVD .....	726
ENPTVT .....	728
ENSPCVD .....	729
EOS .....	730
<i>Examples</i> .....	730
EOSNUM .....	731
EOSS .....	732
EPSCHECK .....	733
EPSCOMP .....	734
EPSDBGS .....	735
EPSDEBUG .....	740
EPSODD3P .....	744
EQLDIMS .....	750
EQLDKVCR .....	751
EQLDREAC .....	753
EQLDTAB .....	755
EQLNUM .....	756
EQLOPTS .....	757
EQLZCORN .....	758
<i>Examples</i> .....	759
EQUALREG .....	761
<i>Legal arrays</i> .....	761
<i>Notes</i> .....	763
<i>Examples</i> .....	764
EQUALS .....	765
<i>Legal arrays</i> .....	765
<i>Notes</i> .....	767
<i>Examples</i> .....	767
EQUIL .....	768
ESPNOE .....	774
ESSNOE .....	775
EXCAVATE .....	776
EXCEL .....	777
EXTFIN .....	778
EXTHOST .....	779
EXTRAPMS .....	780
EXTREPGL .....	781
<b>F</b> .....	<b>782</b>
FACTLI .....	783
<i>Examples</i> .....	783
FASTTRAC .....	784
FAULTDIM .....	785

FAULTS .....	786
FCCRIT .....	788
HERCHBL .....	789
FIELD .....	790
FIELDSEP .....	791
<i>Examples</i> .....	793
FILEUNIT .....	794
FILLEPS .....	795
FIP .....	796
FIPNUM .....	798
FIPOWG .....	799
FIPSEP .....	800
FLUXNUM .....	803
<i>Examples</i> .....	804
FLUXOPTS .....	805
FLUXREG .....	806
FLUXTYPE .....	807
FMTHMD .....	808
FMTIN .....	809
FMTOUT .....	810
FMTSAVE .....	811
FOAM .....	812
FOAMADS .....	813
FOAMDCYO .....	814
FOAMDCYW .....	815
FOAMFCN .....	816
FOAMFRM .....	818
FOAMFSC .....	820
FOAMFSO .....	822
FOAMFST .....	824
<i>Examples</i> .....	824
FOAMFSW .....	826
FOAMMOB .....	828
FOAMMOBP .....	829
FOAMMOBS .....	830
FOAMOPTS .....	831
FOAMROCK .....	832
FORMFEED .....	833
FORMOPTS .....	834
FREEZEPC .....	836
FRICTION .....	837
FULLIMP .....	838
FVST .....	839
FWELLS .....	840
<b>G</b> .....	<b>841</b>

---

GADJUST .....	842
GADVANCE .....	843
GAS .....	844
GASADCO .....	845
GASADDEC .....	846
GASBEGIN .....	847
GASCCMP .....	851
GASCONC .....	852
GASEND .....	853
GASFCOMP .....	854
<i>Examples</i> .....	855
GASFDECR .....	856
GASFDEL .....	857
GASFIELD .....	858
GASFTARG .....	859
GASMONTH .....	860
GASPERIO .....	862
<i>Examples</i> .....	865
GASSATC .....	866
GASSOL .....	867
GASVISCF .....	868
GASVISCT .....	869
GASWAT .....	870
GASYEAR .....	871
<i>Examples</i> .....	874
GCALECON .....	875
GCONCAL .....	877
GCONENG .....	878
GCONINJE .....	879
<i>Examples</i> .....	883
GCONPRI .....	885
GCONPROD .....	889
<i>Examples</i> .....	895
GCONSALE .....	897
<i>Examples</i> .....	899
GCONSUMP .....	901
GCONTOL .....	903
GCUTBACK .....	906
GCUTBACT .....	908
GCVD .....	910
GDCQ .....	911
GDCQECON .....	913
GDFILE .....	914
<i>Examples</i> .....	915
GDIMS .....	916

---

GDORIENT .....	917
GDRILPOT .....	918
GECON .....	920
GECONT .....	923
<i>Examples</i> .....	924
GEFAC .....	926
GEODIMS .....	929
GEOMECH .....	930
GEOYCAP .....	932
GEOYLDF .....	933
GETDATA .....	935
GETGLOB .....	936
GETSOL .....	937
GI .....	938
GIALL .....	939
GIMODEL .....	940
GINJGAS .....	941
GINODE .....	943
GLIFTLIM .....	944
GLIFTOPT .....	945
GLTHEX .....	946
GMDISBC .....	947
GMDISBCL .....	949
GMPSTBC .....	951
GMPSTBCL .....	954
GMTRABC .....	957
GMTRABCL .....	959
GNETDP .....	961
GNETINJE .....	963
GNETPUMP .....	964
GPMaint .....	966
GPMaint3 .....	970
<i>Examples</i> .....	973
GPTABLE .....	974
GPTABLEN .....	975
GPTABLE3 .....	976
GPTDIMS .....	977
GQUALITY .....	978
GRADGRUP .....	979
GRADRESV .....	980
GRADRFT .....	983
GRADWELL .....	984
GRAVCONS .....	985
GRAVDR .....	986
GRAVDRB .....	987

GRAVDRM .....	988
GRAVITY .....	989
GRDREACH .....	990
GREF .....	993
GREFS .....	994
GRIDFILE .....	995
<i>Examples</i> .....	996
GRIDOPTS .....	997
GRIDUNIT .....	999
GRUPFUEL .....	1000
GRUPGR .....	1002
GRUPINJE .....	1004
<i>Examples</i> .....	1007
GRUPLIM .....	1009
GRUPMAST .....	1010
GRUPNET .....	1012
GRUPPROD .....	1015
GRUPRIG .....	1017
<i>Examples</i> .....	1019
GRUPSALE .....	1020
GRUPSLAV .....	1022
GRUPTARG .....	1025
GRUPTREE .....	1027
GSATCOMP .....	1029
GSATINJE .....	1030
GSATPROD .....	1032
GSEPCOND .....	1035
<i>Examples</i> .....	1035
GSEPREPT .....	1036
GSF .....	1038
GSSCPTST .....	1039
GSWINGF .....	1042
GTADD .....	1044
GTMULT .....	1046
GUIDECAL .....	1048
GUIDERAT .....	1049
<i>Examples</i> .....	1052
GUPFREQ .....	1053
GWRTWCV .....	1054
<b>H</b> .....	<b>1055</b>
H2SSOL .....	1056
HALFTRAN .....	1057
HAxxxxxx .....	1058
HDISP .....	1059
HEATCR .....	1060

---

HEATCRT .....	1061
HEATDIMS .....	1062
HEATER .....	1063
HEATERL .....	1065
HEATTR .....	1067
HEATTTHT .....	1068
HEATTX .....	1069
HEATTY .....	1070
HEATTZ .....	1071
HEATVAP .....	1072
HEATVAPE .....	1073
HEATVAPS .....	1074
HMAQUCT .....	1075
HMAQUFET .....	1077
<i>Examples</i> .....	1078
HMAQUNUM .....	1079
HMDIMS .....	1081
<i>Examples</i> .....	1082
HMFAULTS .....	1083
HMMLAQUN .....	1084
HMMLCTAQ .....	1085
HMMLFTAQ .....	1086
HMMLTWCN .....	1087
HMMLTxxx .....	1089
HMMMREGT .....	1090
HMMROCK .....	1091
HMMROCKT .....	1092
HMMULRGT .....	1093
HMMULTxx .....	1094
<i>Examples</i> .....	1095
HMMULTFT .....	1097
HMMULTSG .....	1098
HMPROPS .....	1099
HMROCK .....	1101
HMROCKT .....	1102
HMREF .....	1104
HMWELCON .....	1105
HMWPIMLT .....	1107
HMxxxxxx .....	1108
<i>Examples</i> .....	1110
HRFIN .....	1111
HSGRO .....	1112
HSGRW .....	1113
HSORG .....	1114
HSORW .....	1115

---

HSWRG .....	1116
HSWRO .....	1117
HWELLS .....	1118
HWKRO .....	1119
HWKRORG .....	1120
HWKRORW .....	1121
HWKRW .....	1122
HWKRWR .....	1123
HWPCW .....	1124
HWSNUM .....	1125
HWSOGCR .....	1126
HWSOWCR .....	1127
HWSWCR .....	1128
HWSWL .....	1129
HWSWLPC .....	1130
HWSWU .....	1131
HXFEN .....	1132
HYDRHEAD .....	1133
HYDRO .....	1134
HYFIN .....	1135
HYKR .....	1136
HYMOBGDR .....	1137
HYPC .....	1138
HYST .....	1139
HYSTCHCK .....	1140
HYSTJ .....	1141
HYSTK .....	1142
HZFIN .....	1143
<b>I .....</b>	<b>1144</b>
IHOST .....	1145
IKRG-ORW .....	1146
IKU3P .....	1147
IMBNUM .....	1148
IMBNUMMF .....	1150
IMKRVD .....	1151
IMPCVD .....	1153
IMPES .....	1154
IMPFIE .....	1155
IMPLICIT .....	1156
IMPORT .....	1157
IMPSAT .....	1158
IMPTVD .....	1159
IMSPCVD .....	1161
INCLUDE .....	1162
<i>Examples</i> .....	1162

INIT .....	1163
INRAD .....	1164
INSPEC .....	1165
INTPC .....	1166
IONROCK .....	1167
IONXROCK .....	1168
IONXSURF .....	1169
IPCW,IPCG .....	1171
ISGAS .....	1172
ISTNUM .....	1173
ISGL-OWCR .....	1174
ISOLNUM .....	1175
<b>J .....</b>	<b>1176</b>
JALS .....	1177
JFUNC .....	1179
JFUNCR .....	1182
<b>K .....</b>	<b>1186</b>
KRGDI .....	1187
KRGDM .....	1188
KRG-IKRGR .....	1190
KRNUM .....	1193
KRNUMMF .....	1195
KRODI .....	1196
KRODM .....	1197
KRO-ORG .....	1199
KRWDI .....	1202
KRWDM .....	1203
KRW-IKRWR .....	1205
KVALUES .....	1208
KVAN .....	1209
KVCOMP .....	1210
KVCR .....	1211
KVCRS .....	1212
KVCRWAT .....	1213
KVGEN .....	1215
KVMF .....	1216
KVTABCn .....	1217
KVTABLE .....	1218
KVTABTn .....	1219
KVTEMP .....	1220
KVWI .....	1221
KWTABTn .....	1222
KWTEMP .....	1223
<b>L .....</b>	<b>1224</b>
LAB .....	1225

LAGTRAC .....	1226
LAGTRUPD .....	1227
LANGMEXT .....	1228
LANGMPL .....	1230
LANGMUIR .....	1231
<i>Examples</i> .....	1231
LANGMULC .....	1233
LANGMULT .....	1234
LANGSOLV .....	1235
LBCCOEF .....	1236
LBCCOEFR .....	1237
LCUNIT .....	1238
LGR .....	1239
LGRCOPY .....	1240
LGRFREE .....	1243
LGRLOCK .....	1244
LGROFF .....	1245
<i>Examples</i> .....	1245
LGRON .....	1246
<i>Examples</i> .....	1246
LICENSES .....	1248
LIFTOPT .....	1250
LILIM .....	1252
LINCOM .....	1253
LINKPERM .....	1254
<i>Examples</i> .....	1254
LIVEOIL .....	1255
LKRO .....	1256
LKRORG .....	1257
LKRORW .....	1258
LKRW .....	1259
LKRWR .....	1260
LOAD .....	1261
<i>Examples</i> .....	1262
LOTUS .....	1264
LOWSALT .....	1265
LPCW .....	1266
LSALTFNC .....	1267
LSCRIT .....	1269
LSCRITS .....	1270
LSLTWNUM .....	1271
LSNUM .....	1272
LSOGCR .....	1273
LSOWCR .....	1274
LSWCR .....	1275

LSWL .....	1276
LSWLPC .....	1277
LSWU .....	1278
LTOSIGMA .....	1279
LTRACE .....	1281
LUMPDIMS .....	1282
LUMPING .....	1283
LWKRO .....	1284
LWKROG .....	1285
LWKRORW .....	1286
LWKRW .....	1287
LWKRWR .....	1288
LWPCW .....	1289
LWSLTNUM .....	1290
LWSNUM .....	1291
LWSOGCR .....	1292
LWSOWCR .....	1293
LWSWCR .....	1294
LWSWL .....	1295
LWSWLPC .....	1296
LWSWU .....	1297
LX .....	1298
LY .....	1299
LZ .....	1300
<b>M .....</b>	<b>1301</b>
MAPAXES .....	1302
MAPUNITS .....	1304
MASSFLOW .....	1305
MATCORR .....	1306
<i>Examples</i> .....	1306
MAXTRANZ .....	1307
MAXVALUE .....	1308
MEMORY .....	1309
MEMSAVE .....	1310
MESSAGE .....	1311
MESSAGES .....	1312
MESSOPTS .....	1314
METRIC .....	1315
MIDS .....	1316
MINDZNET .....	1317
MINNCT .....	1318
MINPORV .....	1319
MINPORV9 .....	1320
MINPV .....	1321
MINPVV .....	1322

MINROCKV .....	1323
MINRV .....	1324
MINVALUE .....	1325
MISC .....	1326
MISCEXP .....	1327
MISCIBLE .....	1328
<i>Enable the miscible gas flood option (ECLIPSE 100)</i> .....	1328
<i>Use dependence on surface tensions in properties (ECLIPSE 300)</i> .....	1329
MISCNUM .....	1330
<i>ECLIPSE 100 miscible flood model</i> .....	1330
<i>ECLIPSE 300 miscible model - surface tension effects</i> .....	1330
MISCSTR .....	1332
MISCSTRP .....	1333
MISCSTRR .....	1334
MIXACTG .....	1336
MIXACTO .....	1338
MIXWPERM .....	1340
MLANG .....	1341
MLANGSLV .....	1342
MONITOR .....	1343
MPFA .....	1344
MPFANUM .....	1347
MPFNNC .....	1348
MSFN .....	1350
MSGFILE .....	1352
MULSGGD .....	1353
MULSGGDV .....	1354
MULTFLT .....	1355
MULTIN .....	1357
MULTIPHA .....	1358
MULTIPLY .....	1359
<i>Legal arrays</i> .....	1360
<i>Notes</i> .....	1361
<i>Examples</i> .....	1361
MULTIREG .....	1363
<i>Legal arrays</i> .....	1363
<i>Notes</i> .....	1365
<i>Additional notes</i> .....	1365
<i>Examples</i> .....	1366
MULTMF .....	1367
MULTNUM .....	1368
MULTOUT .....	1369
MULTOUTS .....	1370
MULTPV .....	1371
MULTR .....	1372

MULTR- .....	1373
MULTREAL .....	1374
MULTREGD .....	1375
MULTREGH .....	1377
MULTREGP .....	1379
MULTREGT .....	1381
MULTSAVE .....	1384
MULTSIG .....	1385
MULTSIGV .....	1386
MULTTHT .....	1387
MULTTHT- .....	1388
MULTX .....	1389
MULTX- .....	1391
MULTY .....	1393
MULTY- .....	1395
MULTZ .....	1397
MULTZ- .....	1399
MW .....	1401
MWDETAIL .....	1402
MWS .....	1403
MWW .....	1404
MWWS .....	1405
<b>N .....</b>	<b>1406</b>
NARROW .....	1407
NCOMPS .....	1408
NCONSUMP .....	1409
NEFAC .....	1410
NEI .....	1411
NETBALAN .....	1412
<i>Examples - ECLIPSE 100</i> .....	1414
NETCOMPA .....	1415
NETWORK .....	1418
NEWTON .....	1419
NEWTRAN .....	1420
NEXT .....	1421
NEXTSTEP .....	1422
NEXTSTPL .....	1423
NGASREM .....	1424
<i>Examples</i> .....	1424
NINENUM .....	1426
NINEPOIN .....	1427
NINEXZ .....	1428
NINEYZ .....	1429
NMATOPTS .....	1430
NMATRIX .....	1432

NNC .....	1433
<i>Examples</i> .....	1435
NNCGEN .....	1436
NNEWTF .....	1438
NOAIMLGR .....	1439
NOCASC .....	1440
NOCIRCLE .....	1441
NODEPROP .....	1442
NODPCDT .....	1445
NODPCO .....	1446
NODPPM .....	1447
NOECHO .....	1448
NOFREEZE .....	1449
NOGGF .....	1450
NOHMD .....	1451
NOHMO .....	1452
NOHYKR .....	1453
NOHYPC .....	1454
NOHYST .....	1455
NOINSPEC .....	1456
NOLTRACE .....	1457
NOMIX .....	1458
NOMONITO .....	1459
NONNC .....	1460
NORSSPEC .....	1461
NOSIM .....	1462
NOWARN .....	1463
NOWARNEP .....	1464
NPROCX .....	1465
NPROCY .....	1466
NRSOUT .....	1467
NSTACK .....	1468
<i>Examples</i> .....	1468
NTG .....	1469
NUMRES .....	1470
NUPCOL .....	1471
NWATREM .....	1472
<i>Examples</i> .....	1472
NXFEN .....	1474
NYFIN .....	1475
NZFIN .....	1476
<b>O</b> .....	<b>1477</b>
ODD3P .....	1478
OFM .....	1479
OIL .....	1481

OILAPI .....	1482
OILCOMPR .....	1483
OILMW .....	1484
OILSPECH .....	1485
OILVINDX .....	1486
OILVISCC .....	1488
<i>Examples</i> .....	1490
OILVISCF .....	1492
OILVISCT .....	1494
OILVTIM .....	1497
OLDEQUIL .....	1498
OLDTRAN .....	1499
OLDTRANR .....	1500
OMEGAA .....	1501
OMEGAADE .....	1502
OMEGAAS .....	1503
OMEGAASD .....	1504
OMEGAB .....	1505
OMEGABDE .....	1506
OMEGABS .....	1507
OMEGABSD .....	1508
OPERATE .....	1509
<i>Legal arrays</i> .....	1510
<i>Notes</i> .....	1511
<i>Examples</i> .....	1512
OPERATER .....	1514
<i>Legal arrays</i> .....	1515
<i>Notes</i> .....	1516
<i>Additional notes</i> .....	1517
<i>Examples</i> .....	1518
OPERNUM .....	1520
OPTCONS .....	1521
OPTDIMS .....	1523
OPTFUNC .....	1524
OPTIONS .....	1526
<i>Examples</i> .....	1548
OPTIONS3 .....	1549
<i>Examples</i> .....	1580
OPTLOAD .....	1581
OPTODD3P .....	1582
OPTOPTS .....	1584
OPTPARS .....	1585
<i>Examples</i> .....	1587
OPTTUNE .....	1588
ORTHERRO .....	1589

ORTHERRX .....	1590
ORTHERRY .....	1591
ORTHERRZ .....	1592
OUTRAD .....	1593
OUTSOL .....	1594
OVERBDI .....	1595
OVERBDIL .....	1596
OVERBURD .....	1597
OVERTR .....	1598
OVERTTHT .....	1599
OVERTX .....	1600
OVERTY .....	1601
OVERTZ .....	1602
<b>P .....</b>	<b>1603</b>
PARACHOR .....	1604
PARALLEL .....	1605
PARAOPTS .....	1606
PARTTRAC .....	1608
PATHS .....	1609
PBUB .....	1610
PBVD .....	1611
PCG, IPCG .....	1612
PCG32D .....	1614
PCODD3P .....	1615
PCODD3PG .....	1619
PCODD3PW .....	1620
PCRIT .....	1621
PCRITDET .....	1622
PCRITS .....	1623
PCRITSDE .....	1624
PCW, IPCW .....	1625
PCW32D .....	1627
PDEW .....	1628
PDIVX .....	1629
PDIVY .....	1630
PDVD .....	1631
PEBI .....	1632
PECOEFS .....	1633
PEDERSEN .....	1636
PEDIMS .....	1637
PEDTUNE .....	1638
PEDTUNER .....	1640
PEGTABx .....	1642
PEKTABx .....	1644
PENUM .....	1646

PERMAVE .....	1647
PERMMF .....	1648
PERMR .....	1649
PERMSTAB .....	1650
PERMTHT .....	1652
PERMX .....	1653
PERMXY-ZZ .....	1654
PERMY .....	1655
PERMZ .....	1656
PETGRID .....	1657
<i>Specifying PETGRID keyword and data</i> .....	1657
<i>Example</i> .....	1658
PETOPTS .....	1659
<i>PETOPTS data items</i> .....	1659
<i>Example</i> .....	1660
PICOND .....	1661
PIMTDIMS .....	1664
PIMULTAB .....	1665
PINCH .....	1666
PINCHNUM .....	1669
PINCHOUT .....	1670
PINCHREG .....	1671
PINCHXY .....	1673
PLMIXNUM .....	1674
PLMIXPAR .....	1675
PLYADS .....	1676
PLYADSS .....	1677
PLYATEMP .....	1679
PLYCAMAX .....	1680
PLYDHFLF .....	1681
PLYESAL .....	1682
PLYKRRF .....	1683
PLYMAX .....	1684
PLYOPTS .....	1685
PLYRMDEN .....	1686
PLYROCK .....	1687
PLYROCKM .....	1688
PLYSHEAR .....	1689
PLYSHLOG .....	1692
PLYTRRF .....	1695
PLYTRRFA .....	1696
PLYVISC .....	1697
PLYVISCs .....	1698
PLYVISCT .....	1700
PLYVSCST .....	1701

PMANUM .....	1703
PMAX .....	1704
PMISC .....	1706
POISSONR .....	1707
POLYMER .....	1708
PORO .....	1709
<i>Examples</i> .....	1709
PORV .....	1710
PCWMAX .....	1711
PRCORR .....	1712
PREF .....	1713
PREFS .....	1714
PREFT .....	1715
PREFTS .....	1716
PREFW .....	1717
PREFWS .....	1718
PRESSURE .....	1719
PRIORITY .....	1720
<i>Examples</i> .....	1722
PROPFLOW .....	1723
PRORDER .....	1724
PRVD .....	1728
PSEUPRES .....	1729
PSGRO .....	1730
PSGRW .....	1731
PSORG .....	1732
PSORW .....	1733
PSPLITX .....	1734
PSPLITY .....	1735
PSSTA .....	1736
PSTEADY .....	1737
PSTNUM .....	1739
PSTRBC .....	1740
PSTRBCL .....	1743
PSWRG .....	1746
PSWRO .....	1747
PVCDO .....	1748
PVCO .....	1750
PVDG .....	1753
PVDO .....	1754
PVDS .....	1755
PVTG .....	1756
PVT-M .....	1758
PVTNUM .....	1759
PVTO .....	1760

PVTW .....	1762
PVTWSALT .....	1764
PVZG .....	1767
<b>Q .....</b>	<b>1769</b>
QDRILL .....	1770
QHRATING .....	1771
QMOBIL .....	1772
<b>R .....</b>	<b>1773</b>
RADFIN4 .....	1774
RADFIN .....	1776
RADIAL .....	1779
RAINFALL .....	1780
RBEDCONT .....	1783
RCMASTS .....	1784
REACTACT .....	1785
REACCORD .....	1786
REACCRIT .....	1787
REACENTH .....	1788
REACHES .....	1789
REACLIMS .....	1792
REACOPS .....	1793
REACPHA .....	1794
REACPORD .....	1795
REACRATE .....	1796
REACSMAX .....	1797
REACSORD .....	1798
REACTION .....	1799
REACTRAC .....	1800
READDATA .....	1801
RECOVERY .....	1802
REFINE .....	1803
<i>Examples</i> .....	1806
REGDIMS .....	1808
RESIDNUM .....	1810
RESOPT .....	1811
RESORB .....	1812
RESTART .....	1813
RESVNUM .....	1814
RIVDEBUG .....	1815
RIVERSYS .....	1816
<i>Examples</i> .....	1817
RIVRDIMS .....	1818
RIVRPROP .....	1820
RIVRXSEC .....	1821
RIVSALT .....	1822

RIVTRACE .....	1823
RKPVMNMX .....	1825
RKTRMDIR .....	1826
ROCK .....	1827
ROCK2D .....	1830
ROCK2DTR .....	1832
ROCKCOMP .....	1834
ROCKCON .....	1836
ROCKCONL .....	1838
ROCKDEN .....	1840
ROCKDIMS .....	1841
ROCKFRAC .....	1842
ROCKNUM .....	1843
ROCKOPTS .....	1844
ROCKPAMA .....	1846
ROCKPAME .....	1851
ROCKPMAT .....	1854
ROCKPROP .....	1856
ROCKSPLV .....	1857
ROCKTAB .....	1858
ROCKTABH .....	1860
ROCKTABW .....	1863
ROCKTHSG .....	1864
ROCKTRMX .....	1867
ROCKTRMY .....	1869
ROCKTRMZ .....	1871
ROCKTSIG .....	1873
ROCKV .....	1875
ROCKWNOD .....	1876
ROMF .....	1878
RPRKPAME .....	1879
RPTGRID .....	1880
<i>Examples</i> .....	1884
RPTGRIDL .....	1885
RPTHMD .....	1886
RPTHMG .....	1887
<i>Examples</i> .....	1887
RPTHMW .....	1888
<i>Examples</i> .....	1888
RPTINIT .....	1890
RPTISOL .....	1893
RPTONLY .....	1894
RPTONLYO .....	1895
RPTOPT .....	1896
RPTPRINT .....	1897

RPTPROPS .....	1899
<i>Examples</i> .....	1903
RPTREGS .....	1904
<i>Examples</i> .....	1906
RPTRST .....	1908
<i>Extra output for BASIC=2</i> .....	1924
<i>Examples</i> .....	1926
RPTRUNSP .....	1927
RPTSCHED .....	1928
<i>Examples</i> .....	1945
RPTSLN .....	1946
RPTSMRY .....	1947
RPTSOL .....	1948
<i>Examples</i> .....	1962
RS .....	1963
RSCONST .....	1964
RSCONSTT .....	1965
RSGI .....	1966
RSM .....	1967
RSSPEC .....	1968
RSVD .....	1969
RSW .....	1970
<i>Examples</i> .....	1970
RSWVD .....	1971
<i>Examples</i> .....	1971
RTEMP .....	1973
RTEMPA .....	1974
RTEMPVD .....	1975
RUNSUM .....	1976
RV .....	1977
RVCONST .....	1978
RVCONSTT .....	1979
RVGI .....	1980
RVVD .....	1981
<b>S</b> .....	<b>1982</b>
SALINTR .....	1983
SALINITY .....	1984
SALT .....	1985
<i>Examples</i> .....	1985
SALTEFF .....	1986
SALTNODE .....	1988
SALTVD .....	1989
<i>Examples</i> .....	1989
SAMG .....	1990
SATNUM .....	1991

SATOPTS .....	1992
SAVE .....	1994
<i>Examples</i> .....	1994
SAVEEND .....	1996
SCALECRS .....	1997
SCALELIM .....	1999
SCDATAB .....	2000
SCDATABG .....	2001
SCDETAB .....	2002
SCDPDIMS .....	2004
SCDPTAB .....	2005
SCDPTRAC .....	2006
SCFDIMS .....	2007
SCONINJE .....	2008
SCONPROD .....	2010
SCREF .....	2013
SCVD .....	2014
SDENSITY .....	2015
SDREF .....	2016
<i>Examples</i> .....	2016
SDRGO .....	2018
SDRGW .....	2019
SDROG .....	2020
SDROW .....	2021
SDRWG .....	2022
SDRWO .....	2023
SECTBC .....	2024
SECTOR .....	2026
SEPARATE .....	2027
SEPCOND .....	2028
<i>Examples</i> .....	2030
SEPVALS .....	2031
SFOAM .....	2034
SGAS .....	2035
SGCR .....	2036
SGCWMIS .....	2038
SGF3 .....	2039
SGF32D .....	2041
SGFN .....	2043
SGL,ISGL .....	2045
SGLPC .....	2047
SGOF .....	2048
SGU,ISGU .....	2050
SGWCR .....	2052
SGWFN .....	2054

SHRATE .....	2056
SIGMA .....	2057
SIGMAGD .....	2058
<i>Examples</i> .....	2059
SIGMAGDV .....	2060
<i>Examples</i> .....	2061
SIGMATH .....	2062
SIGMAV .....	2063
SIMULATE .....	2064
SKIP .....	2065
SKIP100 .....	2066
SKIP300 .....	2067
SKIPREST .....	2068
SKIPSTAB .....	2070
SLAVES .....	2071
SLGOF .....	2073
SMF .....	2075
SMRYDIMS .....	2076
SMULTX-Z .....	2077
SOCRS .....	2078
SOF2 .....	2079
SOF3 .....	2081
SOF32D .....	2083
SOGCR .....	2085
SOIL .....	2087
SOILR .....	2088
SOLID .....	2089
SOLIDADS .....	2090
SOLIDMMC .....	2091
SOLIDMMS .....	2092
SOLUAQA .....	2093
SOLUBILI .....	2095
SOLUBILS .....	2097
SOLUBILT .....	2099
SOLUCO2H .....	2101
SOLUH2SC .....	2103
SOLVCONC .....	2105
SOLVDIMS .....	2106
SOLVDIRS .....	2107
SOLVENT .....	2108
SOLVFRAC .....	2109
SOLVNUM .....	2110
<i>Examples</i> .....	2110
SOLWNUM .....	2111
SOLWTAB .....	2112

SOLWW .....	2113
SOMGAS .....	2114
SOMWAT .....	2116
SOR .....	2118
SORBFAC .....	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
<i>Examples</i> .....	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
<i>Examples</i> .....	2171

STRESBCL .....	2172
<i>Examples</i> .....	2174
STVP .....	2175
STWG .....	2176
SUMTHIN .....	2177
SURF .....	2178
SURFACT .....	2179
SURFACTW .....	2180
SURFADDW .....	2181
SURFADS .....	2182
SURFCAPD .....	2183
SURFESAL .....	2184
SURFNUM .....	2185
SURFOPTS .....	2186
SURFROCK .....	2187
SURFST .....	2188
SURFSTES .....	2189
SURFVISC .....	2190
SURFWNUM .....	2191
SWAT .....	2192
SWATINIT .....	2193
SWCR .....	2195
SWF3 .....	2197
SWF32D .....	2199
SWFN .....	2201
SWGCR .....	2203
SWINGFAC .....	2205
SWL,ISWL .....	2206
SWLPC .....	2208
SWOF .....	2209
SWU,ISWU .....	2211
<b>T .....</b>	<b>2213</b>
TABDIMS .....	2214
TBASIS .....	2217
TBLK .....	2218
<i>Examples</i> .....	2219
TBOIL .....	2220
TBOILS .....	2221
TCBDIMS .....	2222
TCRIT .....	2223
TCRITDET .....	2224
TCRITS .....	2225
TCRITSDE .....	2226
TECBDI .....	2227
TECBDIL .....	2229

TEMP .....	2231
TEMPI .....	2232
TEMPNODE .....	2233
TEMPTVD .....	2234
TEMPVD .....	2235
THANALB .....	2236
THANALH .....	2237
THANALV .....	2238
THCGAS .....	2239
THCOIL .....	2240
THCONMF .....	2241
THCONR .....	2242
THCONSF .....	2243
THCROCK .....	2244
THCSOLID .....	2245
THCWATER .....	2246
THERMAL .....	2247
THERMEX1 .....	2248
THERMNUM .....	2249
THFLXTF .....	2250
THPRES .....	2251
THPRESFT .....	2253
THSTT97 .....	2254
THSVC .....	2255
THTABB .....	2256
THWVC .....	2257
TIGHTEN .....	2258
TIGHTENP .....	2259
TIME .....	2260
TITLE .....	2261
TLMIXPAR .....	2262
TNUM .....	2263
<i>Examples</i> .....	2263
TOLCRIT .....	2265
TOPS .....	2266
TOPT .....	2267
TPAMEPS .....	2269
TPAMEPSS .....	2271
TRACER .....	2272
<i>Examples</i> .....	2273
TRACERKM .....	2274
TRACERKP .....	2276
TRACERS .....	2277
TRACITVD .....	2279
<i>Examples</i> .....	2280

TRACK .....	2281
TRACKREG .....	2282
TRACTBC .....	2283
TRACTBCL .....	2285
TRACTVD .....	2287
TRADS .....	2288
TRANGE .....	2290
TRANGL .....	2291
TRANR .....	2292
TRANHT .....	2293
TRANX .....	2294
TRANY .....	2295
TRANZ .....	2296
TRCOEF .....	2297
TRDCY .....	2298
TRDIF .....	2299
TRDIS .....	2300
TREF .....	2302
TREFS .....	2303
TREFT .....	2304
TREFTS .....	2305
TRKPF .....	2306
TRNHD .....	2307
TRPLPORO .....	2308
TRROCK .....	2309
TSCRIT .....	2311
TSTEP .....	2314
TUNING .....	2315
<i>ECLIPSE 100</i> .....	2315
<i>Examples in ECLIPSE 100</i> .....	2317
<i>ECLIPSE 300</i> .....	2319
<i>Example in ECLIPSE 300</i> .....	2321
TUNINGDP .....	2322
TUNINGH .....	2324
TUNINGL .....	2326
TUNINGS .....	2327
TVDP .....	2328
<i>Examples</i> .....	2329
TZONE .....	2330
<b>U</b> .....	<b>2331</b>
UDADIMS .....	2332
UDQ .....	2334
<i>Syntax for specifying particular elements</i> .....	2335
<i>Functions available for quantity definitions</i> .....	2337
<i>Data available for quantity definitions</i> .....	2340

<i>Initialization of quantities</i> .....	2341
<i>Processing of quantity assignments</i> .....	2341
<i>Evaluation of quantity definitions</i> .....	2341
<i>Examples</i> .....	2343
<i>How to define a cumulative quantity</i> .....	2344
<i>How to use conditional UDQs</i> .....	2345
<i>How to use UDQs for time-derivative calculation</i> .....	2345
UDQDIMS .....	2346
UDQPARAM .....	2347
UDT .....	2348
<i>Examples</i> .....	2349
UDTDIMS .....	2352
UNCODHMD .....	2353
UNIFIN .....	2354
UNIFOUT .....	2355
UNIFOOTS .....	2356
UNIFSAVE .....	2357
USECUPL .....	2358
USEFLUX .....	2359
USENOFLO .....	2361
<b>V</b> .....	<b>2362</b>
VAPOIL .....	2363
VAPPARS .....	2364
VCOMPACT .....	2365
VCRIT .....	2366
VCRITDET .....	2367
VCRITS .....	2368
VCRITSDE .....	2369
VCRITVIS .....	2370
VDFLOW .....	2371
VDFLOWR .....	2372
VDKRG .....	2373
VDKRGC .....	2375
VDKRO .....	2376
VE .....	2378
VECTABLE .....	2379
VEDEBUG .....	2380
VEFIN .....	2382
<i>Examples</i> .....	2383
VEFRAC .....	2384
VEFRACP .....	2385
VEFRACPV .....	2386
VEFRACV .....	2387
VELDEP .....	2388
VFPCHECK .....	2389

VFPCHK .....	2390
VFPIDIMS .....	2391
VFPINJ .....	2392
<i>Examples</i> .....	2394
VFPPDIMS .....	2395
VFPPROD .....	2396
<i>Record 1:Basic data for table</i> .....	2397
<i>Record 2:FLO (flow rate) values</i> .....	2400
<i>Record 3:THP values</i> .....	2400
<i>Record 4:WFR values</i> .....	2400
<i>Record 5: GFR values</i> .....	2400
<i>Record 6:ALQ values</i> .....	2400
<i>Records 7 to NTHP*NWFR*NGFR*NALQ+6</i> .....	2401
<i>Example</i> .....	2403
VFPTABL .....	2404
VISAGE .....	2405
VISCAQA .....	2406
<i>Examples</i> .....	2406
VISCD .....	2407
VISCREF .....	2408
VISDATES .....	2409
VISOPTS .....	2410
VREFW .....	2412
<b>W</b> .....	<b>2413</b>
WADVANCE .....	2414
WAGHALT .....	2415
WAGHYSTR .....	2416
WAITBAL .....	2420
WALKALIN .....	2421
WALQCALC .....	2422
WAPI .....	2424
WARN .....	2425
WARP .....	2426
WATDENT .....	2427
WATER .....	2428
WATERTAB .....	2429
WATVISCT .....	2430
WAVAILIM .....	2432
WBHGLR .....	2433
WBOREVOL .....	2436
WCALVAL .....	2438
WCOL .....	2439
WCOMPRES .....	2440
WCONHIST .....	2441
WCONINJ .....	2446

WCONINJE .....	2447
<i>Examples</i> .....	2451
WCONINJH .....	2453
WCONINJP .....	2458
WCONPROD .....	2462
WCUTBACK .....	2467
WCUTBACT .....	2471
WCYCLE .....	2474
WDFAC .....	2476
WDFACCOR .....	2478
WDRILPRI .....	2480
WDRILRES .....	2482
WDRILTIM .....	2483
WECON .....	2486
WECONCMF .....	2491
WECONINJ .....	2492
WECONT .....	2494
<i>Examples</i> .....	2496
WEFAC .....	2497
WELCNTL .....	2499
WELDEBUG .....	2501
WELDRAW .....	2502
WELEVNT .....	2505
WELLCF .....	2506
WELLCOMP .....	2507
<i>Examples</i> .....	2509
WELLDIMS .....	2510
WELLGR .....	2512
WELLINJE .....	2514
<i>Examples</i> .....	2518
WELLKBHP .....	2520
WELLIM .....	2521
WELLOPEN .....	2522
WELLOPTS .....	2523
WELLPROD .....	2524
<i>Examples</i> .....	2526
WELLSHUT .....	2527
WELLSMA .....	2528
WELLSPEC .....	2529
WELLSTRE .....	2531
WELLSTRW .....	2532
WELLTARG .....	2533
WELLTCB .....	2534
WELLTCBC .....	2535
WELLTCBT .....	2536

WELLWAG .....	2537
<i>Examples</i> .....	2538
WELMOVEL .....	2540
WELOPEN .....	2541
WELOPENL .....	2543
WELPI .....	2546
WELPRI .....	2548
WELSEGS .....	2550
<i>Examples</i> .....	2555
WELSOMIN .....	2557
WELSPECL .....	2558
<i>Examples</i> .....	2562
WELSPECS .....	2563
<i>Examples</i> .....	2567
WELTARG .....	2568
WFOAM .....	2570
WFRICSEG .....	2571
WFRICSGL .....	2572
WFRICTN .....	2573
<i>Each subsequent record</i> .....	2574
<i>Examples</i> .....	2576
WFRICTNL .....	2577
<i>Each subsequent record</i> .....	2578
<i>Examples</i> .....	2580
WGASPROD .....	2581
WGMEM .....	2583
WGORPEN .....	2584
WGRUPCON .....	2586
WH2NUM .....	2588
WH3NUM .....	2589
WHEDREFD .....	2590
WHISTCTL .....	2591
<i>Examples</i> .....	2592
WHTEMP .....	2593
WI .....	2594
WINJEDET .....	2595
WINJGAS .....	2596
WINJMIX .....	2598
WINJMULT .....	2600
WINJOIL .....	2602
WINJORD .....	2603
WINJTEMP .....	2605
WINJW .....	2607
WINJWAT .....	2608
WLIFT .....	2609

WLIFTOPT .....	2613
WLIMTOL .....	2616
WLIST .....	2617
WLISTARG .....	2619
WLISTDYN .....	2621
<i>Examples</i> .....	2623
WLISTNAM .....	2624
WMF .....	2626
WMFVD .....	2627
WNAMEs .....	2628
WNETCTRL .....	2629
WNETDP .....	2631
WORKLIM .....	2632
WORKTHP .....	2633
WPAVE .....	2634
<i>Examples</i> .....	2635
<i>Calculation of well block average pressures</i> .....	2636
WPAVEDEP .....	2638
WPIMULT .....	2639
WPIMULTL .....	2641
WPITAB .....	2643
WPLUG .....	2644
WPOLYMER .....	2646
WPOLYRED .....	2647
WPOTCALC .....	2648
WREGROUP .....	2649
WRFT .....	2651
<i>Examples</i> .....	2651
WRFTPLT .....	2652
<i>Examples</i> .....	2655
WSALT .....	2656
<i>Examples</i> .....	2657
WSCCLEAN .....	2658
WSCCLENL .....	2660
WSCTAB .....	2662
WSEGAICD .....	2664
WSEGDFIN .....	2668
WSEGDFMD .....	2670
WSEGDFPA .....	2672
WSEGDIRS .....	2675
WSEGEXSS .....	2676
WSEGFLIM .....	2680
WSEGFMOD .....	2683
WSEGHEAT .....	2686
WSEGINIT .....	2689

WSEGITER .....	2691
WSEGLABY .....	2693
WSEGLINK .....	2697
WSEGMULT .....	2699
WSEGPROP .....	2701
WSEGPULL .....	2703
WSEGSEP .....	2706
WSEGSICD .....	2708
WSEGSOLV .....	2716
WSEGTABL .....	2718
WSEGVALV .....	2721
WSEGWELL .....	2725
WSEPCOND .....	2726
WSF .....	2727
WSOLVENT .....	2728
WSURFACT .....	2729
WTADD .....	2730
WTAKEGAS .....	2732
WTEMP .....	2734
WTEMPQ .....	2735
WTEST .....	2736
WTHPMAX .....	2739
WTMULT .....	2741
WTRACER .....	2743
WVFPDP .....	2745
WVFPEXP .....	2746
<i>Notes</i> .....	2747
<i>Example</i> .....	2748
WWPAVE .....	2749
<b>X</b> .....	<b>2752</b>
XMF .....	2753
XMFVP .....	2754
XSTC .....	2755
<b>Y</b> .....	<b>2756</b>
YLDNUM .....	2757
YMF .....	2758
YMFVP .....	2759
YOUNGMOD .....	2760
YSTC .....	2761
<b>Z</b> .....	<b>2762</b>
ZCORN .....	2763
ZCRIT .....	2764
ZCRITDET .....	2765
ZCRITS .....	2766
ZCRITSDE .....	2767

ZCRITVIS .....	2768
ZFACT1 .....	2769
ZFACT1S .....	2770
ZFACTOR .....	2771
ZFACTORS .....	2772
ZI .....	2773
ZIPP2OFF .....	2774
ZIPPY2 .....	2775
<i>Data</i> .....	2776
<i>Examples</i> .....	2778
ZMF .....	2780
ZMFVD .....	2781



# 1

## 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.

- 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

## 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.
  - CO<sub>2</sub> 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<sup>th</sup> 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<sup>th</sup> 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`, `HWKRWR`, `HWKRO`, `HWKRORW`, `HWKRORG` 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`, `LWKRWR`, `LWKRO`, `LWKRORW`, `LWKRORG` 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](#), [SURFNUM](#) and [LWLTNUM](#) 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](#), [BVELW0I](#), [BVELW0J](#) and [BVELW0K](#) may now be used in conjunction with the [PLYSHEAR](#) option. These mnemonics were previously compatible only with the [PLYSHLOG](#) option.
- 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.
- 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 [SSFRR](#) 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 [ESPNOE](#), respectively. The [ESSNODE](#) and [ESPNOE](#) tables can be output to the PRINT file with [RPTPROPS](#) arguments [ESSNODE](#) and [ESPNOE](#).
- 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 13<sup>th</sup> 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 ft<sup>3</sup>. 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](#)

- [PROPS section](#)
- [SOLUTION section](#)
- [SUMMARY section](#)
- [SCHEDULE 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](#), [KRORW](#) and [KROrg](#) (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`, `HYPC`, `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 `WRF'TPLT`) 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`, `ADDREG` 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](#), [LSWCR](#), [LSWU](#), [LSOWCR](#), [LSOGCR](#), [LKRW](#), [LKRWR](#), [LKRO](#), [LKRORW](#), [LKRORG](#) and [LPCW](#) Low salinity (oil-wet) water and oil saturation table end-points.
- [HWSWL](#), [HWSWLPC](#), [HWSWCR](#), [HWSWU](#), [HWSOWCR](#), [HWSOGCR](#), [HWKRW](#), [HWKRWR](#), [HWKRO](#), [HWKRORW](#), [HWKROORG](#) and [HWPCW](#) Adsorbed surfactant (high salinity) water-wet water and oil saturation table end-points.
- [LWSWL](#), [LWSWLPC](#), [LWSWCR](#), [LWSWU](#), [LWSOWCR](#), [LWSOGCR](#), [LWKRW](#), [LWKRWR](#), [LWKRO](#), [LWKROORG](#) and [LWPCW](#) 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 [SURFNUM](#)).
- [LWSNUM](#) Low-salinity water-wet saturation region numbers (abbreviated alias for existing keyword [LSLTNUM](#)).

## SUMMARY section

The new SUMMARY keywords are:

- [WPIO](#) Oil phase productivity index
- [WPIG](#) Gas phase productivity index
- [WPIW](#) 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.
- [PLYVISCs](#) 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.
- [PLYVISC](#) 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:

Switch	Description
191	This item has been deprecated. To control the number of messages output for dead block connections use item 13 of the <a href="#">MESSAGES</a> keyword or <a href="#">item 220</a> of the <a href="#">OPTIONS</a> keyword.
210	If set to 1, the pre-2014.1 behavior is restored for the calculation of relative permeabilities when using the <a href="#">SURFACT</a> or <a href="#">LOWSALT</a> models with <a href="#">ENDSCALE</a> .
211	If set to 1, the pre-2014.1 behavior is restored for the calculation of the capillary number when using the <a href="#">SURFACT</a> and <a href="#">DUALPORO</a> models.
212	If set to 1, connate gas will be included in the end-point scaling calculation of the saturation at which <a href="#">KRWR</a> occurs in 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 <a href="#">STONE2</a> .
214	If set to 1, the pre-2014.1 behavior is restored for the calculation of the component cell heights for the <a href="#">GRAVDR</a> model when used with either the <a href="#">SURFACT</a> or <a href="#">LOWSALT</a> 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 <a href="#">WELSPECL</a> on an LGR that is switched off is downgraded to a warning.
217	If set to 1, the pre-2014.1 behavior for <a href="#">VDFLOW</a> 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 <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> keyword. Prior to 2014.1 this was <b>only</b> applied to the phase mobility calculations for the well connections if the <a href="#">FHERCHBL</a> keyword had also been specified. For the 2014.1 version, by default, the non-Darcy flow component due to the <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> keyword is not applied to the well connections in any circumstance. If the <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> flow component is required to be applied to the well connections, this may be enabled by setting this <a href="#">OPTIONS</a> item to 2. Wells or well completions for which D-Factor calculations have been specified via the <a href="#">WDFAC</a> keyword the <a href="#">WDFACCOR</a> keyword or item 12 of the <a href="#">COMPDAT</a> keyword (or item 13 of the <a href="#">COMPDATL</a> keyword) will supersede those due to the <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> 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 <a href="#">MESSAGES</a> item 13 is ignored.

Switch	Description
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 <a href="#">MINNCT</a> 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, the pre 2014.1 behavior of allowing multiple uses of the <a href="#">GRIDOPTS</a> keyword is restored.
224	If set to 1, the pre-2014.1 behavior for <a href="#">CECON</a> 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 behavior for modeling the saturation functions for low salinity <a href="#">LOWSALT</a> , surfactant <a href="#">SURFACT</a> and surfactant wettability <a href="#">SURFACTW</a> 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.
226	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 <a href="#">item 317</a> of the <a href="#">OPTIONS3</a> 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 <a href="#">item 318</a> of the <a href="#">OPTIONS3</a> 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 <a href="#">SHRATE</a> , applies a modified conversion factor from velocity to shear rate. If <a href="#">OPTIONS</a> 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 <a href="#">PLYSHEAR</a> or <a href="#">PLYSHLOG</a> . If <a href="#">OPTIONS</a> 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 <a href="#">SHRATE</a> . If <a href="#">OPTIONS</a> 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 <a href="#">SHRATE</a> . Please see the chapter entitled " <a href="#">Polymer Flood Model</a> " in the <i>ECLIPSE Technical Description</i> for precise details of the forms of these conversions.
229	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.
230	If set to 1, the pre-2014.1 behavior of the 'TRNC' timestepping control for <a href="#">POLYMER/BRINE</a> and <a href="#">SOLVENT</a> runs will be restored.
231	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 <a href="#">SCALECRS</a> item 1 has been set to YES.
232	If set to 1, this removes the ban on EOR/tracer functionality with <a href="#">DUMPFLUX</a> and <a href="#">USEFLUX</a> keywords. You must ensure that no information on EOR/tracer functionality is needed from outside the flux boundary.

Switch	Description
233	If set to 1, the pre-2014.1 behavior for the input processing of the two-dimensional capillary pressure data specified via the <a href="#">PCG32D</a> and <a href="#">PCW32D</a> keywords will be restored. 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.
234	If set to 1, the pre-2014.1 behavior for the scaling of gas relative permeability at residual saturation of the displacing phase ( <a href="#">KRGR</a> ) 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, grid property tables in the PRT file will be printed so that the maximum scaled value always lies between 1 and 10, and no significant digits are lost from that value. The default value of 0 corresponds to earlier versions of the simulator where for some small values only one significant digit would be printed.
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, this removes the ban on brine injection with <a href="#">WSALT</a> in dual porosity models with local grid refinements. The use of this option is not recommended and 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 <a href="#">THPRES</a> keyword, will be calculated from the initial conditions considering any phase present in the upstream cell.

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.
  - **PLYVISC** outputs the polymer viscosity table specified by the **PLYVISC** keyword.
  - **SURFSTES** outputs the polymer viscosity table specified by the **SURFSTES** keyword.
  - **ESPNO** and **ESSNO** output the effective salinity nodes table specified by the **ESPNO** and **ESSNO** 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:

Switch	Description
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 <code>KRWR</code> , <code>KRGR</code> , <code>KRORW</code> and <code>KROG</code> (relative permeabilities at residual saturation of the displacing phase) occur are the same as the saturations at which <code>KRW</code> , <code>KRG</code> , <code>KRO</code> and <code>KRO</code> (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 <code>ODD3P</code> 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.
308	If set to -1, then even when the <code>DPGRID</code> 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.
309	If set to 1, this will revert to the pre-2013.2 behavior of the <code>COMPKRIL</code> 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 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.
312	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.
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 <code>SOLUTION</code> section.
314	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 <code>WAGHYSTR</code> keyword when end-point scaling has been specified and where the three-phase water relative permeability saturation table numbers have been specified via the <code>WH3NUM</code> keyword.
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.

Switch	Description
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. 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 <a href="#">item 226</a> of the <code>OPTIONS</code> keyword.
318	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 <a href="#">item 227</a> of the <code>OPTIONS</code> keyword.
319	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 <code>WDFAC</code> keyword, the <code>WDFACCOR</code> keyword or item 12 of the <code>COMPDAT</code> keyword (or item 13 of the <code>COMPDATL</code> 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.
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 <code>HYSTER</code> option of the <code>SATOPTS</code> keyword in the <code>RUNSPEC</code> section but the <code>EHYSTR</code> keyword has been omitted from the <code>PROPS</code> section. For versions prior to 2014.1, neither relative permeability or capillary pressure hysteresis were selected correctly if, in addition to omitting the <code>EHYSTR</code> keyword, none of the alternative keywords used for specifying hysteresis, for example, <code>HYKR</code> , <code>HYPG</code> , <code>HYST</code> , <code>HYSTJ</code> or <code>HYSTK</code> were specified. For the 2014.1 version, if hysteresis has been specified via the <code>HYSTER</code> option of the <code>SATOPTS</code> keyword but the <code>EHYSTR</code> keyword has been omitted, the hysteresis parameters will be assigned to the default values specified for the <code>EHYSTR</code> 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 <code>PCW</code> and <code>PCG</code> 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 <code>PCW</code> or <code>PCG</code> 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 <code>JFUNC</code> and <code>JFUNCR</code> keywords.

Table 1.2: Altered `OPTIONS3` switches

## 2

## Data File Overview

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:

Required?	Section Name	Description
Required	<a href="#">RUNSPEC</a>	Title, problem dimensions, switches, phases present and components for example.
Required	<a href="#">GRID</a>	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.
Optional	<a href="#">EDIT</a>	Modifications to calculated pore volumes, grid block center depths and transmissibilities.

Required?	Section Name	Description
Required	<a href="#">PROPS</a>	Tables of properties of reservoir rock and fluids as functions of fluid pressures, saturations and compositions (including density, viscosity, relative permeability and capillary pressure). Contains the equation of state description in compositional runs.
Optional	<a href="#">REGIONS</a>	Splits computational grid into regions for calculation of: <ul style="list-style-type: none"> <li>• PVT properties (fluid densities and viscosities)</li> <li>• Saturation properties (relative permeabilities and capillary pressures)</li> <li>• Initial conditions (equilibrium pressures and saturations)</li> <li>• Fluids in place (fluid-in-place and inter-region flows)</li> <li>• EoS regions (for compositional runs).</li> </ul> <p>If this section is omitted, all grid blocks are put in region 1.</p>
Required	<a href="#">SOLUTION</a>	Specification of initial conditions in reservoir. May be: <ul style="list-style-type: none"> <li>• Calculated using specified fluid contact depths to give potential equilibrium</li> <li>• Read from a restart file set up by an earlier run</li> <li>• Specified by the user for every grid block</li> </ul> <p>(not recommended for general use)</p>
Optional	<a href="#">SUMMARY</a>	Specification of data to be written to the Summary file after each timestep. Necessary if certain types of graphical output (for example water-cut as a function of time) are to be generated after the run has finished. If this section is omitted no Summary files are created.
Required	<a href="#">SCHEDULE</a>	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 <a href="#">SCHEDULE</a> section.
Optional (ECLIPSE 300 only)	<a href="#">OPTIMIZE</a>	Specifies a reservoir optimization problem (objective function, control parameters and constraints).

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:

```
SATNUM
1 1 1 1 1 1 1 1 1
2 2 2 2 2 2 2 2 2 /
```

```
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:

```
WECON
PROD1 2000 1* 0.8 2* CON /
/
```

## 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.

*ECLIPSE 100* 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.

*ECLIPSE 100* 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.

*ECLIPSE 100* 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 an 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

Restriction	Keyword	Description
E100 only	<a href="#">COLUMNS</a>	Reset left and right margins for reading data input file
E100 only	<a href="#">DEBUG</a>	Set debug output control switches

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
E100 only	FORMFEED	Sets the formfeed character in the Print file
E100 only	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.

Error Code	Description
1	Unable to read next line from input file
2	Length of current line is zero
3	Quote misplaced in stack or data field
4	Slash character (/) found in the wrong place
5	Data field length greater than 24
6	End of file reached

Error Code	Description
7	Invalid character found
8	Zero length stack found
9	Stack length 1 with illegal character
10	Incorrect stack pointers
11	Failed to find character in repeated data field
12	Multiplier for repeated data field has more than 24 characters
13	Error after internal read of multiplier value
14	Negative multiplier for repeated data field
15	character found as first token in stack
16	Unrecognizable number in repeated data field
17	Single field has the wrong data type
18	Empty token next on stack
19	Unknown data type
20	Data read which is not of the type expected, and which cannot easily be converted
21	Error in internal read to convert character data to number
22	Repeated data has filed length of more than 24 characters
23	Error in internal read of repeated data field multiplier
24	More than one exponential character found
29	Previous data type of repeated filed not compatible for current repeated data field
30	Error in internal read of previous repeated data field
31	Unable to convert current value due to error in previous repeated data field
32	Values for margin setting not valid
33	Zero length string found internally

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' /
.....

```

```

SOLUTION =====
RPTSOL
 20*1 /
INCLUDE
 'SOLUTION.SECT' /
SUMMARY =====
INCLUDE
 'SUMMARY.SECT' /
SCHEDULE =====
RPTSCHED
 0 1 0 1 1 0 5*2 /
INCLUDE
 'SCHEDULE.SECT' /
END =====

```

## 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:

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

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.

ECLIPSE 300

---

**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

Keyword	Description
<code>ACTDIMS</code>	Dimensions for the ACTION facility.
<code>ACTPARAM</code>	Parameters for controlling ACTION facility functions.
<code>ASPHALTE</code>	Enable the asphaltene modeling
<code>BLACKOIL</code>	Requests black oil mode.
<code>BRINE</code>	Enable brine tracking.
<code>CART</code>	Specify Cartesian geometry.
<code>COAL</code>	Enable the Coal Bed Methane option.
<code>CPR</code>	Activate the CPR linear solver.
<code>DIFFUSE</code>	Enable molecular diffusion.
<code>DIMENS</code>	Specify the grid dimensions.
<code>DUALPERM</code>	Use the Dual Permeability option.
<code>DUALPORO</code>	Use the Dual Porosity option.
<code>DYNRDIMS</code>	Sets up dimensions for the Dynamic Region facility.
<code>ENDSCALE</code>	Use saturation table end point scaling.
<code>FIELD</code>	Use Field unit convention.

Keyword	Description
FLUXOPTS	Options for Flux Boundary models.
FMTHMD	Indicates that the HMD file is to be formatted.
FMTIN	Indicates that the input files are formatted.
FMTOUT	Indicates that the output files are to be formatted.
FULLIMP	Use the Fully Implicit Solution option.
GAS	The run contains gas.
GDIMS	Specify the dimensions for the instantaneous gradient option.
GRAVDR	Use gravity drainage in dual porosity runs.
GRAVDRB	Use the Vertically Discretized Gravity Drainage Model in dual porosity runs.
GRAVDRM	Use the Alternative Gravity Drainage Model in dual porosity runs.
GRIDOPTS	Set options to process grid data.
HMDIMS	Set dimensions for the Gradient option.
IMPES	Use the IMPES Solution option.
INSPEC	Request an Initial Index file.
LAB	Use Lab unit convention.
LGRCOPY	Copy global values to local cells before operations on cells within an LGR. This will apply to all LGRs in the dataset.
MEMORY	Allocate the required memory at start of run.
METRIC	Use metric unit convention.
MISCIBLE	Enable the Miscible Gas Flood option (ECLIPSE 100) Allow properties to depend on surface tension (ECLIPSE 300).
MONITOR	Request output for run-time monitoring.
MSGFILE	Message file output options.
MULTIN	Indicates that input files are multiple.
MULTOUT	Indicates that output files are multiple.
MULTOUTS	Indicates that Summary files are multiple.
MULTREAL	Activates Multi-Realization licensing.
NETWORK	Set dimensions for Extended Network Model.
NINEPOIN	Select the Nine-point option.
NMATRIX	Use the discretized Matrix Dual-porosity Model.
NOHYST	Disable Hysteresis option.
NOINSPEC	Do not write an Initial Index file.
NOMONITO	Disable the output for run-time monitoring.
NONNC	Disallow non-neighbor connections.
NORSSPEC	Do not write a Restart Index file.
NOSIM	Turn off simulation, but continue data checking.

<b>Keyword</b>	<b>Description</b>
NSTACK	Set linear solver stack size.
NUMRES	The grid contains multiple reservoirs.
NUPCOL	Number of timestep iterations in which well targets are updated.
OIL	The run contains oil.
PARALLEL	Select a parallel run
PETOPTS	Sets options for simulator runs fired from Petrel.
RADIAL	Specify radial geometry.
REGDIMS	Set dimensions for regions data.
ROCKCOMP	Enable the rock compaction option.
RSSPEC	Request a Restart Index file.
SATOPTS	Set options for directional and hysteretic relative permeabilities.
SOLID	Indicates a solid phase in the run
START	Specify the start date.
TABDIMS	Set table dimensions.
TITLE	Specify the run title.
UDADIMS	Dimensions for the user defined arguments facility.
UDQDIMS	Dimensions for the user defined quantity facility.
UDQPARAM	Parameters for the user defined quantity facility.
UNCODHMD	Enables unencoded sensitivities from the Gradient option.
UNIFIN	Indicates that input files are unified.
UNIFOUT	Indicates that output files are unified.
UNIFOUTS	Indicates that Summary files are unified.
VFPIDIMS	Set dimensions for injection well VFP tables.
VFPPDIMS	Set dimensions for production well VFP tables.
VISAGE	Activate the VISAGE geomechanics workflow.
WARP	Enable WARP linear solver.
WATER	The run contains water.
WCOMPRES	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.
WELLDIMS	Set dimensions for well data.
WSEGDIMS	Set dimensions for multisegment wells.
WSEGSOLV	Sets parameters for the multisegment well iterative linear solver.

## ECLIPSE 100 only keywords

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

<b>ECLIPSE 100 only keyword</b>	<b>Description</b>
SMRYDIMS	Set maximum number of Summary file quantities.
SOLVENT	Enable the Four-component Solvent Model.
SURFACT	Enable the Surfactant Model.
SURFACTW	Enable modeling of changes of wettability in the Surfactant Model.
TRACERS	Set dimensions and options for tracers
TEMP	Enable the Temperature option.
VAPOIL	The run contains vaporized oil in wet gas.
VE	Enable the Vertical Equilibrium option
VISCD	Enable the Viscous Displacement option.
WPOTCALC	Control calculation of shut and stopped well potentials.

### ECLIPSE 300 only keywords

<b>ECLIPSE 300 only keyword</b>	<b>Description</b>
AIM	Use the AIM Solution option.
AJGRADNT	Use the adjoint gradient screening option.
AQUDIMS	Dimensions for aquifers.
CBMOPTS	Specify the type of adsorption model to be used with the Coal Bed Methane option.
CO2SOL	Allow CO <sub>2</sub> to dissolve in the aqueous phase.
CO2STORE	Activates the CO <sub>2</sub> storage option
COMPS	Requests compositional mode.
COMPSOL	For SOLID, BLACKOIL runs, this keyword must be used to set the number of components.
COMPW	Requests water component model.
DEADOIL	Use the Dead Oil option in thermal simulations.
DOMAIN	Set non-default domain size for phase property evaluation.
DPCDT	Set maximum rate of change of capillary pressure.
E100NOSI	Switch to NOSIM mode if an incompatible ECLIPSE 100 keyword is read.
EOS	Specify which equation of state is to be used.
EOSS	Specify the surface equation of state.
FASTTRAC	Enables shortcuts in tracer calculations.
FMTSAVE	Indicates that the Save files are formatted
FORMOPTS	Formulation options.
FREEZEPC	Freeze all capillary pressures in the run.
FWELLS	Enable the Fractured Well Completion option.

ECLIPSE 300 only keyword	Description
GASSOL	Use the GASSOL option.
GASWAT	Use the water-gas model.
GEOMECH	Selects the geomechanics option
GEODIMS	Sets the number of geomechanics regions in the simulation, sets table dimensions, in terms of the number of rows and columns, for the <a href="#">PERMSTAB</a> and <a href="#">BIOTCTAB</a> keywords.
GPTDIMS	Set gas plant table dimensions.
H2SSOL	Allow H2S to dissolve in the aqueous phase
HEATDIMS	Set heater keyword dimensions in thermal simulations.
HWELLS	Enable the Horizontal Well Completion option.
HYKR	Enable the Relative Permeability Hysteresis option.
HYPC	Enable the Capillary Pressure Hysteresis option.
HYST	Enable the Hysteresis option.
HYSTJ	Enable the Hysteresis option using the Jargon method.
HYSTK	Enable the Hysteresis option using the Killough method.
IMPSAT	Use the IMPSAT Solution option.
ISGAS	The run is gas condensate.
JALS	Activate the JALS linear solver.
KVALUES	Use K-values in the simulation
LAGTRAC	Activates the Lagrangian Tracer Solver option.
LIVEOIL	Use the live oil option in thermal simulations.
LUMPDIMS	Turns on, and dimensions, the LUMPING option.
MULTSAVE	Indicates that Save files are multiple.
MULTIPHA	Allows multi-phase flash for asphaltene prediction.
MEMSAVE	Allow use of Up Front Memory option.
NINEXZ	Select the Nine-point option in the XZ plane.
NINEYZ	Select the Nine-point option in the YZ plane.
NOAIMLGR	Suppress making all refinement cells implicit in AIM runs.
NODPCO	No dual porosity condensation
NODPCDT	Disable the <a href="#">DPCDT</a> option.
NODPPM	No dual porosity permeability multiplier.
NOFREEZE	Suppress use of the <a href="#">FREEZEPC</a> option.
NOHYKR	Do not apply hysteresis to relative permeabilities.
NOHYPC	Do not apply hysteresis to capillary pressure.
NOMIX	Do not interpolate between oil and gas relative permeabilities near the critical point.

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

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

Alternatively, data may be set to constant values using the `EQUALS` keyword, and manipulated with keywords `ADD`, `MULTIPLY`, `COPY`, `MINVALUE` and `MAXVALUE`. Data can also be edited by flux region, `MULTNUM` region or `OPERNUM` region with the keywords `ADDREG`, `EQUALREG`, `COPYREG` and `MULTIREG`. The flux regions are defined with keyword `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 `OUTRAD` keyword in ECLIPSE 100, or the `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 `TOPS` keyword.

## Essential keywords

Block centered — Cartesian	Block centered - Radial	Corner point
	INRAD	
DXV or DX	DRV or DR or OUTRAD	COORD and ZCORN
DYV or DY	DTHETAV or DTHETA	or GDFILE or
DZ	DZ	PETGRID
TOPS	TOPS	
PORO	PORO	PORO
PERMX	PERMR	PERMX/PERMR
PERMY	PERMTHT	PERMY/PERMTHT
PERMZ	PERMZ	PERMZ

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. by 50 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 /
    
```

```
DZV
10 50 /
```

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.

ECLIPSE 300

**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  $NTG = DZNET / DZ$  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 **MINPORV** (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 **PERMZ** 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.

```

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

Keyword	Description
<b>COARSEN</b>	Generates non-neighbor connections generated from using the grid coarsening option.
<b>CIRCLE</b>	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.
<b>NOCIRCLE</b>	may be used in local radial grid refinement runs to complete the circle.
<b>NNC</b>	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 <b>NNCGEN</b> keyword.
<b>SIGMA</b>	may be used to specify a coupling parameter for NNCs set up in Dual Porosity runs. The use of this is discussed in "Dual Porosity" in the <i>ECLIPSE Technical Description</i> .
<b>SIGMAGD</b>	may be used to specify a different coupling parameter for gravity drainage. By default, the <b>SIGMA</b> values are used.
<b>PINCH</b>	specifies how missing zero thickness layers are to be treated. NNCs may be generated across the absent layer.
<b>AQUCON</b>	connects a numerical aquifer to the reservoir using NNCs

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 a 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).

ECLIPSE 300

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

Cartesian keywords	Description
<a href="#">DX</a>	X-direction grid block sizes for the current input box (or use <a href="#">DXV</a> ).
<a href="#">DXV</a>	Vector of X-direction grid block sizes or use <a href="#">DX</a> . Simplified version of <a href="#">DX</a>
<a href="#">DY</a>	Y-direction grid block sizes for the current input box (or use <a href="#">DYV</a> ).
<a href="#">DYV</a>	Vector of Y-direction grid block sizes or use <a href="#">DY</a> . Simplified version of <a href="#">DY</a>

### Radial block centered

Restriction	Radial keyword	Description
Required for radial block centered geometry.	<a href="#">INRAD</a>	Sets inner radius of radial grid.
Must be set	<a href="#">DR</a>	R-direction grid block sizes for the current input box (or use <a href="#">DRV</a> or <a href="#">OUTRAD</a> ).
Optional for radial geometry	<a href="#">DRV</a>	Vector of R-direction grid block sizes (or use <a href="#">DR</a> or <a href="#">OUTRAD</a> ). Simplified version of <a href="#">DR</a>
E100 only.	<a href="#">OUTRAD</a>	Sets outer radius of radial grid (or use <a href="#">DR</a> or <a href="#">DRV</a> ).
Must be set	<a href="#">DTHETA</a>	Theta-direction grid block sizes for the current input box (or use <a href="#">DTHETAV</a> ).
	<a href="#">DTHETAV</a>	Vector of Theta-direction grid block sizes (or use <a href="#">DTHETA</a> ). Simplified version of <a href="#">DTHETA</a>

## Corner point geometry

Restriction	Corner point keyword	Description
Either COORD and ZCORN or GDFILE are required for corner point geometry.	COORD	Defines the lines that contain all grid block corner points for each (I, J) and for each reservoir in the grid.
Either COORD and ZCORN or GDFILE are required for corner point geometry.	ZCORN	Point depths of grid block corners.
Either COORD and ZCORN or GDFILE are required for corner point geometry.	GDFILE	Inputs corner point positions for each cell from a GRID file as an alternative to COORD and ZCORN.
Optional	ADDZCORN	Adds a constant value to cell corner point depths
Optional	EQLZCORN	Resets part of the corner point depth array within a specified box
E100 only Required only if > 1, or for completing the circle	COORDSYS	Information about the coordinate system for each reservoir in the grid. Specifies completion of the circle in both corner point and block-centered geometries.
Optional	RESVNUM	Initiates COORD data input for a given reservoir in the grid
Optional	MAPAXES	Inputs map origin from a grid pre-processor, for output to the GRID file
Optional	MAPUNITS	Specifies the units used for MAPAXES data for output to the GRID file
Optional	GRIDUNIT	Specifies the grid data units that can be output by grid pre-processors or by ECLIPSE simulators writing the grid geometry files

## Block properties, all geometries

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

Restriction	Keyword	Description
Optional	<a href="#">MULTPV</a>	Pore volume multipliers for the current input box
Must be if radial with $NR > 1$ .	<a href="#">PERMR</a>	R-direction permeabilities for the current input box.
Must be if radial with $NTHETA > 1$ .	<a href="#">PERMTH</a>	Theta-direction permeabilities for the current input box.
Must be set if Cartesian with $NX > 1$ .	<a href="#">PERMX</a>	X-direction permeabilities for the current input box
Must be set if Cartesian with $NY > 1$ .	<a href="#">PERMY</a>	Y-direction permeabilities for the current input box.
Must be set for any system with $NZ > 1$	<a href="#">PERMZ</a>	Z-direction permeabilities for the current input box.
May be used only if both <a href="#">ENDSCALE</a> and item <a href="#">SURFTENS</a> in <a href="#">SATOPTS</a> are entered in the <a href="#">RUNSPEC</a> section.	<a href="#">JFUNC</a>	Activates Leverett J-Function option to scale capillary pressure according to rock porosity and permeability
May be used only if both <a href="#">ENDSCALE</a> and item <a href="#">SURFTENS</a> in <a href="#">SATOPTS</a> are entered in the <a href="#">RUNSPEC</a> section.	<a href="#">JFUNCR</a>	Activates Leverett J-Function option to scale capillary pressure according to rock porosity and permeability on a per saturation region basis.  Each set of saturation function tables may be specified to use either capillary pressures values obtained from direct table look-up or via the Leverett J-Function calculation.

## 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

ECLIPSE 100 only keyword	Description
<a href="#">NEWTRAN</a>	Specifies that corner point data should be used to compute transmissibilities
<a href="#">OLDTRAN</a>	Specifies that block center data should be used to compute transmissibilities.
<a href="#">OLDTRANR</a>	Alternative calculation of transmissibilities using block characteristics
<a href="#">PINCHNUM</a>	Identifies pinchout regions for use by <a href="#">PINCHREG</a> .
<a href="#">PINCHREG</a>	Specifies pinchout data within <a href="#">PINCHNUM</a> regions.

ECLIPSE 100 only keyword	Description
<a href="#">LINKPERM</a>	Treats grid block permeabilities as applying to cell faces.
<a href="#">HALFTRAN</a>	Activates the half-block transmissibility upscaling option.
<a href="#">PERMAVE</a>	Specifies options for averaging grid block permeabilities in the calculation of transmissibilities

## Transmissibility modifiers

Restriction	Keyword	Description
Optional if radial with NR > 1	<a href="#">MULTR</a>	R-direction transmissibility multipliers for the current input box.
Optional if radial with NR > 1	<a href="#">MULTR-</a>	Negative R-direction transmissibility multipliers for the current input box.
Optional if radial with NTHETA > 1	<a href="#">MULTTHT</a>	Theta direction transmissibility multipliers for the current input box.
Optional if radial with NTHETA > 1	<a href="#">MULTTHT-</a>	Negative Theta-direction transmissibility multipliers for the current input box.
Optional if Cartesian with NX > 1	<a href="#">MULTX</a>	X-direction transmissibility multipliers for the current input box.
Optional if Cartesian with NX > 1	<a href="#">MULTX-</a>	Negative X-direction transmissibility multipliers for the current input box.
Optional if Cartesian with NY > 1	<a href="#">MULTY</a>	Y-direction transmissibility multipliers for the current input box.
Optional if Cartesian with NY > 1	<a href="#">MULTY-</a>	Negative Y-direction transmissibility multipliers for the current input box.
Optional if NZ > 1	<a href="#">MULTZ</a>	Z-direction transmissibility multipliers for the current input box.
Optional if NZ > 1	<a href="#">MULTZ-</a>	Negative Z-direction transmissibility multipliers for the current input box.
Optional	<a href="#">MULTREGT</a>	Multiplies transmissibilities between regions defined with <a href="#">FLUXNUM</a> or <a href="#">MULTNUM</a> .
Optional	<a href="#">MULTNUM</a>	Identifies regions for use with <a href="#">MULTREGT</a> , <a href="#">MULTREGD</a> (ECLIPSE 100 only) and <a href="#">MULTREGH</a> (ECLIPSE 100 only). Also defines regions, for editing data with <a href="#">ADDREG</a> , <a href="#">EQUALREG</a> , <a href="#">MULTIREG</a> and <a href="#">COPYREG</a> .

Table 2.5: Transmissibility modifiers

## Faults

Keyword	Description
<a href="#">FAULTS</a>	Identifies fault trajectories for subsequent use by the <a href="#">MULTFLT</a> keyword.

Keyword	Description
MULTFLT	Multipliers for the transmissibility and diffusivity across faults defined by the FAULTS keyword.
THPRESFT	Threshold pressures associated with faults defined by the FAULTS keyword.

## Numerical aquifers

### Common keywords

Restriction	Keyword	Description
Required for numerical aquifer definition	AQUCON	Defines a connection between a numerical aquifer and the reservoir.
Required for Carter-Tracy aquifer definition	AQUCT	Specifies property data for Carter-Tracy aquifer.
Required for numerical aquifer definition	AQUNUM	Defines a numerical aquifer

### ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
Required for analytic aquifer definition	AQUANCON	Defines connection data for analytic aquifer.

## Dual porosity

### Common keywords

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

Keyword	Description
<a href="#">SIGMAGD</a>	Matrix-to-fracture coupling for oil-gas gravity drainage, for the entire grid
<a href="#">SIGMAGDV</a>	Matrix-to-fracture coupling for oil-gas gravity drainage, for the current box.

## ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
<a href="#">LTOSIGMA</a>	Provides an easy way to calculate <a href="#">SIGMA</a> from LX, LY, LZ.
<a href="#">LX</a> , <a href="#">LY</a> , <a href="#">LZ</a>	Inputs the representative matrix block sizes for the viscous displacement option.

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">MULTMF</a>	Transmissibility multipliers for matrix-fracture coupling.
<a href="#">PERMMF</a>	Permeability for matrix-fracture coupling.

## Compressed vertical equilibrium (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
Optional, CVE runs only	<a href="#">COLLAPSE</a>	Specifies which cells should be collapsed.
Optional, CVE runs only	<a href="#">CRITPERM</a>	Sets a critical permeability to determine which cells are collapsed.
Optional, VE runs only	<a href="#">VEDEBUG</a>	Controls debug output for VE options.

## Diffusivity modifiers

### Common keywords

Restriction	Keyword	Description
Optional if radial with > 1	<a href="#">DIFFMR</a>	R-direction diffusivity multipliers for the current input box.
Optional if radial with > 1	<a href="#">DIFFMR-</a>	Negative R-direction diffusivity multipliers for the current input box.
Optional if radial with > 1	<a href="#">DIFFMTH</a>	Theta-direction diffusivity multipliers for the current input box.
Optional if radial with > 1	<a href="#">DIFFMTH-</a>	Negative Theta-direction diffusivity multipliers for the current input box.
Optional if Cartesian with > 1	<a href="#">DIFFMX</a>	X-direction diffusivity multipliers for the current input box.

Restriction	Keyword	Description
Optional if Cartesian with > 1	<a href="#">DIFFMX-</a>	Negative X-direction diffusivity multipliers for the current input box.
Optional if Cartesian with > 1	<a href="#">DIFFMY</a>	Y-direction diffusivity multipliers for the current input box.
Optional if Cartesian with > 1	<a href="#">DIFFMY-</a>	Negative Y-direction diffusivity multipliers for the current input box.
Optional if > 1	<a href="#">DIFFMZ</a>	Z-direction diffusivity multipliers for the current input box.
Optional if > 1	<a href="#">DIFFMZ-</a>	Negative Z-direction diffusivity multipliers for the current input box.
Optional, dual porosity runs only	<a href="#">DIFFMMF</a>	Matrix-fracture diffusivity multipliers for the current input box.
Optional	<a href="#">MULTFLT</a>	Multipliers for the transmissibility and diffusivity across faults defined by the keyword.

## ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
Optional.	<a href="#">MULTREGD</a>	Multipliers for the diffusivity between regions defined with <a href="#">FLUXNUM</a> or <a href="#">MULTNUM</a> .
Optional.	<a href="#">MULTNUM</a>	Identifies regions for use with <a href="#">MULTREGT</a> , <a href="#">MULTREGD</a> and <a href="#">MULTREGH</a>

## 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)

ECLIPSE 100 only keyword	Description
<a href="#">MULTREGH</a>	Multipliers for the thermal conductivity between regions defined with <a href="#">FLUXNUM</a> or <a href="#">MULTNUM</a> .
<a href="#">MULTNUM</a>	Identifies regions for use with <a href="#">MULTREGT</a> , <a href="#">MULTREGD</a> and <a href="#">MULTREGH</a> .
<a href="#">THCONR</a>	Rock thermal conductivities for the current input box.

## Nine-point scheme (ECLIPSE 300)

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

## Operational keywords

### Common keywords

Keyword	Description
ADD	Adds specified constants to specified arrays in the current input box.
ADDREG	Adds specified constants to specified arrays in a given flux, <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
BOUNDARY	Defines the section of the grid for which output maps are required.
BOX	Redefines the current input box
COPY	Copies data in the current box from one specified array to another.
COPYBOX	Copies data from one box of cells to another box of the same size.
COPYREG	Copy by <a href="#">FLUXNUM</a> region, <a href="#">MULTNUM</a> region or <a href="#">OPERNUM</a> region.
DUMPFLUX	Output a Flux file.
ENDBOX	Redefines the current input box to encompass the whole grid.
EQUALS	Sets specified arrays to specified constants within the current box.
EQUALREG	Sets specified arrays to specified constants in a given flux, <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
FLUXNUM	Defines flux regions, for editing data with <a href="#">ADDREG</a> , <a href="#">EQUALREG</a> , <a href="#">MULTIREG</a> and <a href="#">COPYREG</a> , and for inter-region multipliers <a href="#">MULTREGT</a> , <a href="#">MULTREGD</a> and <a href="#">MULTREGH</a>
FLUXREG	Select which flux regions are active in a run.
GRIDFILE	Controls the contents of the Grid Geometry file (for graphical output).
IMPORT	Imports binary data generated by a grid pre-processor.
INIT	Produces initial file for GRAF.
LGRCOPY	Copy global values to local cells before operations on cells within an LGR.
MAXVALUE	Applies a maximum value to specified arrays within the current box.
MINNNCT	Sets a cut-off transmissibility below which an NNC is deleted.
MINVALUE	Applies a minimum value to specified arrays within the current box.

Keyword	Description
MULTIPLY	Multiply specified arrays by specified constants within the current box.
MULTIREG	Multiply specified arrays by specified constants in a given flux, MULTNUM or OPERNUM region.
NOGGF	Specifies that no Grid Geometry file is to be produced (for graphical output).
OPERATE	Performs arithmetic operations on arrays.
OPERATER	Performs arithmetic operations on arrays for a specific region of the grid.
OPERNUM	Defines region on which to perform operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.
RPTGRID	Sets report levels for GRID data.
RPTINIT	Controls output to the Initialization file for GRID and EDIT data.
SOLVDIRS	Overrides default settings for principal directions in the linear solver.
SPECGRID	Specification of grid dimensions.
TOPS	Specifies the depths of the top face of each grid cell.
USEFLUX	Use a Flux file.

## ECLIPSE 300 only keywords

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

## Local grid refinement

### Common keywords

Keyword	Description
CARFIN	Defines a Cartesian local grid refinement.
COARSEN	Specifies a box of cells for grid coarsening.
ENDFIN	Terminates data for local grid refinement.
HRFIN	Local grid size ratios in the R-direction.
HXFIN	Local grid size ratios in the X-direction.
HYFIN	Local grid size ratios in the Y-direction.
HZFIN	Local grid size ratios in the Z-direction.
NXFIN	Number of local cells in each global cell of an LGR in the X-direction.

Keyword	Description
<a href="#">NYFIN</a>	Number of local cells in each global cell of an LGR in the Y-direction.
<a href="#">NZFIN</a>	Number of local cells in each global cell of an LGR in the Z-direction.
<a href="#">RADFIN</a>	Specifies a single-column radial local grid refinement.
<a href="#">REFINE</a>	Initiates data input for a named local grid.
<a href="#">RPTGRIDL</a>	Controls output of GRID section data for local grids.

## ECLIPSE 100 only keywords

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

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">NOCIRCLE</a>	Turn off completion of circle for local radial refinements.

## Tensor permeability

### Common keywords

- [MPFANUM](#) Identifies regions in which to apply the multipoint discretization or nine point scheme.
- [MPFNNC](#) Sets multi-point transmissibility coefficients.

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">GADJUST</a>	Adjusts diagonal permeability tensor.
<a href="#">MPFA</a>	Sets options for tensor permeability.
<a href="#">PERMXX</a>	X-direction tensor permeability coefficient for the current input box.
<a href="#">PERMY</a>	Y-direction tensor permeability coefficient for the current input box.
<a href="#">PERMZZ</a>	Z-direction tensor permeability coefficient for the current input box.
<a href="#">PERMXY</a>	XY-direction tensor permeability coefficient for the current input box
<a href="#">PERMYZ</a>	YZ-direction tensor permeability coefficient for the current input box.

ECLIPSE 300 only keyword	Description
PERMZ	ZX-direction tensor permeability coefficient for the current input box.
TBASIS	Specify basis for tensor permeabilities.

## 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

Keyword	Description
ACTNUM	Identifies active grid blocks.
ADD	Adds specified constants to specified arrays in the current box.
ADDREG	Adds a constant to the specified array in a given flux, <b>MULTNUM</b> or <b>OPERNUM</b> region.
ADDZCORN	Adds a constant to the cell corner point depths array ZCORN.
AQUCON	Defines a connection between a numerical aquifer and the reservoir.
AQUNUM	Defines a numerical aquifer.
BOUNDARY	Defines the section of the grid for which output maps are required.
BOX	Redefines the current box.
CARFIN	Defines a Cartesian local grid refinement.
COARSEN	Specifies a box of cells for grid coarsening.
COALNUM	Specifies coal regions for the Coal bed Methane option
COORD	Defines the lines that contain all grid block corner points for each (I,J) and for each reservoir in the grid.
COORDSYS	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.
COPY	Copies data in the current box from one specified array to another.
COPYBOX	Copies data from one box of cells to another box of the same size.
COPYREG	Copy by <b>FLUXNUM</b> region, <b>MULTNUM</b> region or <b>OPERNUM</b> region
DIFFMMF	Matrix-fracture diffusivity multipliers for current input box.

<b>Keyword</b>	<b>Description</b>
DIFFMR	R-direction diffusivity multipliers for current input box.
DIFFMR-	Negative R-direction diffusivity multipliers for current input box.
DIFFMTH	Theta-direction diffusivity multipliers for current input box.
DIFFMTH-	Negative Theta-direction diffusivity multipliers for current input box.
DIFFMX	X-direction diffusivity multipliers for current input box.
DIFFMX-	Negative X-direction diffusivity multipliers for current input box.
DIFFMY	Y-direction diffusivity multipliers for current input box.
DIFFMY-	Negative Y-direction diffusivity multipliers for current input box.
DIFFMZ	Z-direction diffusivity multipliers for current input box.
DIFFMZ-	Negative Z-direction diffusivity multipliers for current input box.
DPGRID	Enables you to enter grid data for matrix cells only.
DPNUM	Identifies single porosity regions in a dual porosity run.
DR	R-direction grid block sizes for the current box.
DRV	Vector of R-direction grid block sizes.
DTHETA	Theta-direction grid block sizes for the current box.
DTHETA V	Vector of theta-direction grid block sizes.
DUMPFLUX	Output a Flux file.
DX	X-direction grid block sizes for the current box.
DXV	Vector of X-direction grid block sizes.
DY	Y-direction grid block sizes for the current box.
DYV	Vector of Y-direction grid block sizes.
DZ	Z-direction (vertical) grid block sizes for the current box.
DZMTRX	The vertical dimension of a typical block of matrix material, for the entire grid.
DZMTRXV	The vertical dimension of a typical block of matrix material, for the current box.
DZNET	Grid block net thicknesses for the current box.
ENDBOX	Redefines the current box to encompass the whole grid.
ENDFIN	Terminates data for a local grid refinement.
EQLZCORN	Resets part of the corner point depth array.
EQUALREG	Sets a specified array to a constant value in a given flux, MULTNUM or OPERNUM region.
EQUALS	Sets specified arrays to specified constants in the current box.
EXTFIN	Specifies an unstructured local grid refinement.
EXTHOST	Specifies the global host cells of an unstructured local grid refinement.
EXTREPGL	Specifies the replaced global cells for an unstructured local grid refinement.
FAULTS	Identifies fault trajectories for subsequent use by MULTFLT keyword.
FILEUNIT	Confirms the units of the data set.
FLUXNUM	Defines flux regions, for editing GRID section data.

Keyword	Description
FLUXREG	Select which flux regions are active in a run.
GDFILE	Inputs corner point positions for each cell from a Grid file.
GDORIENT	Indicates the orientation and handedness of the grid
GRIDFILE	Controls contents of grid geometry file (for graphical output).
GRIDUNIT	Specifies the grid data units that can be output by grid pre-processors or by ECLIPSE simulators writing the grid geometry files
HMMULTPV	Specifies pore volume multipliers (cumulative).
HMMULTxx	Specify history matching multipliers on transmissibility and/or permeability.
HRFIN	Local grid size ratios in the R-direction.
HXFIN	Local grid size ratios in the X-direction.
HYFIN	Local grid size ratios in the Y-direction.
HZFIN	Local grid size ratios in the Z-direction.
IMPORT	Imports binary data generated by a grid pre-processor.
INIT	Produces Initial file for GRAF.
INRAD	Sets inner radius of radial grid.
JFUNC	Activates Leverett J-Function option for scaling capillary pressure.
JFUNCR	Activates Leverett J-Function option for scaling capillary pressure on a per saturation table basis.
LGRCOPY	Copy global values to local cells before operations on cells within an LGR.
MAPAXES	Inputs map origin from a grid pre-processor, for output to the Grid file.
MAPUNITS	Specifies the units used for MAPAXES data.
MAXVALUE	Applies a maximum value to specified arrays within the current box.
MINPV	Sets a minimum pore volume that each cell must have to be active.
MINPVV	Sets a vector of minimum pore volumes that the cells in the current box must have to be active.
MINVALUE	Applies a minimum value to specified arrays within the current box.
MPFANUM	Identifies regions in which to apply the multipoint discretization or nine point scheme.
MPFNNC	Sets multi-point transmissibility coefficients.
MULTFLT	Multipliers for the transmissibility and diffusivity across faults defined by the <a href="#">FAULTS</a> keyword.
MULTIPLY	Multiplies specified arrays by constant values in the current box.
MULTIREG	Multiplies specified arrays by constant values in a flux, MULTNUM or OPERNUM region.
MULTNUM	Identifies multiplier regions for use with MULTTREGT, MULTTREGD and MULTTREGH. Also defines regions, for editing data with ADDREG, EQUALREG, MULTIREG and COPYREG.
MULTPV	Pore volume multipliers for the current box
MULTR	R-direction transmissibility multipliers for the current box.
MULTR-	Negative R-direction transmissibility multipliers for the current box.

Keyword	Description
MULTREGP	Multiplies pore volumes between flux or multiplier regions.
MULTREGT	Multiplies transmissibilities between flux or multiplier regions.
MULTTHT	Theta-direction transmissibility multipliers for the current box.
MULTTHT-	Negative R-direction transmissibility multipliers for the current box.
MULTX	X-direction transmissibility multipliers for the current box.
MULTX-	Negative X-direction transmissibility multipliers for the current box.
MULTY	Y-direction transmissibility multipliers for the current box.
MULTY-	Negative Y-direction transmissibility multipliers for the current box.
MULTZ	Z-direction transmissibility multipliers for the current box.
MULTZ-	Negative Z-direction transmissibility multipliers for the current box.
NNC	Explicit entry of non-neighbor connections.
NOGGF	Stops production of grid geometry file (for graphical output).
NODPPM	Specifies that permeabilities in the fracture cells are not to be multiplied by the fracture porosity.
NTG	Grid block net-to-gross ratios for the current box.
NXFIN	Number of local cells in each global cell of an LGR in the X-direction.
NYFIN	Number of local cells in each global cell of an LGR in the Y-direction.
NZFIN	Number of local cells in each global cell of an LGR in the Z-direction.
OPERATE	Performs arithmetic operations on arrays.
OPERATER	Performs arithmetic operations on arrays for a specific region of the grid.
OPERNUM	Defines region on which to perform operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.
PERMR	R-direction permeabilities for the current box.
PERMTHT	Theta-direction permeabilities for the current box.
PERMX	X-direction permeabilities for the current box.
PERMY	Y-direction permeabilities for the current box.
PERMZ	Z-direction permeabilities for the current box.
PINCH	Generates connections across pinched-out layers.
PINCHOUT	Same as keyword PINCH, but threshold thickness cannot be adjusted.
PINCHXY	Generates horizontal (I, J) pinchout connections.
PORO	Grid block porosities for the current box.
RADFIN	Specifies a single-column radial local grid refinement.
REFINE	Initiates data input for a named local grid.
RESVNUM	Initiates COORD data input for a given reservoir in the grid.
ROCKFRAC	Sets the rock volume fraction for each grid block.
RPTGRID	Sets report levels for GRID data.

Keyword	Description
RPTINIT	Controls output to the Initialization file for GRID and EDIT data.
RPTGRIDL	Controls output of GRID section data for local grids.
SIGMA	Matrix-to-fracture coupling factor for the entire grid
SIGMAGD	Matrix-fracture coupling for oil-gas gravity drainage (whole grid).
SIGMAGDV	Matrix-fracture coupling for oil-gas gravity drainage (current box).
SIGMAV	Matrix-to-fracture coupling factor for the current box.
SOLVDIMS	Specifies the solver dimensions for an unstructured local grid refinement.
SOLVDIRS	Overrides default settings for principal directions in the linear solver.
SOLVNUM	Specifies the map from user numbers to solver numbers for an unstructured grid.
SPECGRID	Specification of grid dimensions and characteristics.
THCONR	Rock and fluid thermal conductivities for the current box, for the Temperature and Thermal options.
THPRESFT	Specifies a threshold pressure associated with a named fault.
TOPS	Depths of top faces of grid blocks for the current box.
TRANGL	Specifies the global-local connections and associated transmissibilities for an unstructured local grid refinement.
USEFLUX	Use a Flux file.
ZCORN	Depths of grid block corners.

## ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
	AMALGAM	Allow local grids to amalgamate.
	AQUNNC	Specifies numerical aquifer non-neighbor connection properties explicitly.
	AUTOCOAR	Specifies a box of cells for coarsening within an auto refinement
	COLLAPSE	Specifies which cells should be collapsed in Compressed VE option.
	HALF'TRAN	Activates the half-block transmissibility upscaling option
	HMAQUNUM	Activates the calculation of gradients for numerical aquifer parameters.
	HMMLAQUN	Specifies cumulative multipliers on numerical aquifer parameters.
	HMMMREGT	Specifies cumulative inter-region transmissibility multipliers
	HMMULRGT	Activates the calculation of gradients with respect to inter-region transmissibility multipliers.
	IONROCK	Rock ion exchange capacity
	ISOLNUM	Defines the extent of each independent reservoir region.
	LINKPERM	Treats grid block permeabilities as applying to cell faces.
	LTOSIGMA	Provides an easy way to calculate SIGMA from LX, LY, LZ.

Restriction	ECLIPSE 100 only keyword	Description
	LX, LY, LZ	Inputs the representative matrix block sizes for the dual porosity viscous displacement option.
	MULTREGD	Multiplies diffusivities between flux or multiplier regions.
	MULTREGH	Multiplies thermal conductivity between flux or multiplier regions
Do not use with block centered geometry.	NEWTRAN	Specifies that transmissibilities are to be computed using area of contact between blocks. Default for corner point geometry.
	NINENUM	Specifies the extent of the nine-point discretization scheme.
	NMATOPTS	Controls the discretized matrix Dual Porosity option.
	OLDTRAN	Specifies that block center data should be used to compute transmissibilities.
	OLDTRANR	Alternative calculation of transmissibilities using block center data.
	OUTRAD	Sets outer radius of radial grid.
	PEBI	Specifies that an unstructured grid is being used.
	PERMAVE	Sets options for permeability averaging in transmissibilities.
	PINCHNUM	Identifies pinchout regions for use by PINCHREG
	PINCHREG	Specifies pinchout data within PINCHNUM regions.
	QMOBIL	Controls the mobile end-point correction in an LGR.
	RADFIN4	Specifies a four-column radial local grid refinement.
	RPTISOL	Generates data for the ISOLNUM keyword, output to the Debug file.
	SMULTX, SMULTY, SMULTZ	Alternative transmissibility multipliers for auto refinement.
	VEDEBUG	Controls debug output from the Vertical Equilibrium options.
	VEFIN	Controls Vertical Equilibrium in local grids.

## ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
	AQUANCON	Defines connection data for analytic aquifer
	AQUCT	Specifies property data for Cater-Tracy aquifer
	BIOTC	Sets the Biot constant for each grid block
Radial geometry only	CIRCLE	Requests completion of circle
	COALNUMR	Redefine coal region numbers in connection with the multi-porosity option.
	CONDFLTS	Specifies geometry of conductive faults.

Restriction	ECLIPSE 300 only keyword	Description
	CONDFRAC	Specifies geometry of a conductive fracture
	CRNDENS	Specifies rock density for each coal region number
	CRITPERM	Sets a critical permeability for cell collapse in Compressed VE
	DZV	Vector of Z-direction grid block sizes
	GADJUST	Adjusts diagonal permeability tensor
	GLTHEX	Sets a coefficient of linear thermal expansion for each grid block.
Thermal	HEATCR	Rock heat capacity
Thermal	HEATCRT	Temperature dependence of rock heat capacity
	MAXTRANZ	Maximum Z-direction transmissibility.
	MIDS	Specify cell mid-point depths.
	MINDZNET	Minimum net thickness for active cell.
	MINPORV9	Set a minimum pore volume for calculation of nine-point transmissibilities.
Thermal	MINROCKV	Sets a minimum rock volume that each cell must have to be active
	MPFA	Sets options for tensor permeability.
	MULTMF	Transmissibility multipliers for matrix-fracture coupling.
	NNCGEN	Allows NNCs to be generated between any two grid cells.
	NOCIRCLE	Turn off completion of circle for local grid refinements.
	ORTHERRO	Set a minimum orthogonality error for calculation of nine-point transmissibilities.
	ORTHERRX	Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the X-direction.
	ORTHERRY	Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Y-direction.
	ORTHERRZ	Set a minimum orthogonality error for calculation of nine-point transmissibilities for interfaces in the Z-direction
	PERMMF	Permeability for matrix-fracture coupling.
	PERMXX	X-direction tensor permeability coefficient for the current input box.
	PERMYY	Y-direction tensor permeability coefficient for the current input box.
	PERMZZ	Z-direction tensor permeability coefficient for the current input box.
	PERMXY	XY-direction tensor permeability coefficient for the current input box.
	PERMYZ	YZ-direction tensor permeability coefficient for the current input box
	PERMZX	ZX-direction tensor permeability coefficient for the current input box
	POISSONR	Sets the Poisson ratio for each grid block.
	PROPFLOW	Defines parameters to be used with the proponent flowback feature

Restriction	ECLIPSE 300 only keyword	Description
Thermal	ROCKCON	Base and cap rock connections to the global grid
Thermal	ROCKCONL	Base and cap rock connections to the local grid refinements
	ROCKDEN	Sets the rock density for each grid block
Thermal	ROCKPROP	Base and cap rock properties
Thermal	SIGMATH	Matrix-fracture thermal conductivity multiplier
	TBASIS	Specify basis for tensor permeabilities
Thermal	THCGAS	Gas phase thermal conductivity
Thermal	THCOIL	Oil phase thermal conductivity
Thermal	THCONMF	Matrix-fracture conductivity
Thermal	THCONSF	Thermal conductivity saturation dependence
Thermal	THCROCK	Rock thermal conductivity
Thermal	THCSOLID	Solid phase thermal conductivity
Thermal	THCWATER	Water phase thermal conductivity
	THFLXTF	Heat transmissibility factor for the flux boundary option.
	YOUNGMOD	Sets the elastic Young's modulus for each grid block.
	XSTC	Specifies gas composition at standard conditions
	YSTC	Specifies oil composition at standard condition.

## 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
DXV
  1500 3*1000 2000 /      X dimensions of blocks only varies in X direction
DYV
  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 /
PORO    0.3 /
/
-- Set individual values for Porosity and Tops for layer 1
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

```

```

    PERMX PERMY /      Copy all 30 Permx values to Permy
    PERMX PERMZ /
    /
MULTIPLY
    PERMZ 0.07 /      Multiply all 30 Permz values by 0.07
    /
-- Grid file will be output by default
-- Init file containing block properties for input to GRAF
INIT
RPTGRID
'TRANX' 'TRANY' 'TRANZ' / requests output of transmissibilities
PROPS Next section header ends current section

```

## 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

DEPTH	PORV	TRANX	TRANR	DIFFX	DIFFR
		TRANY	TRANRHT	DIFFY	DIFFRHT
		TRANZ		DIFFZ	

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, MULTRHT

MULTZ, MULTPV

ECLIPSE 100

MULTX-, MULTR-

ECLIPSE 100

MULTY-, MULTRHT-

ECLIPSE 100

MULTZ-

and the operational keywords

BOX	EQUALS	ADD
ENDBOX	COPY	MULTIPLY
MINVALUE	MAXVALUE	

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.

---

ECLIPSE 100

---

**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.

Cartesian Keyword	Radial Keyword	Description
TRANX	TRANR	X or R direction transmissibilities
TRANY	TRANHT	Y or $\theta$ transmissibilities
TRANZ	TRANZ	Z transmissibilities

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

Restriction	Keyword	Description
	<code>ADD</code>	Adds specified constants to specified arrays in the current input box.
	<code>ADDREG</code>	Adds a constant to the specified array in a flux, <code>MULTNUM</code> or <code>OPERNUM</code> region.
	<code>BOUNDARY</code>	Defines the section of the grid for which output maps are required.
	<code>BOX</code>	Re-defines the current input box. <code>COPY</code> Copies data in the current box from one specified array to another.
	<code>COPYREG</code>	Copy by <code>FLUXNUM</code> region <code>MULTNUM</code> region or <code>OPERNUM</code> region.
	<code>DEPTH</code>	Grid block center depths for the current input box.

Restriction	Keyword	Description
Molecular Diffusion option in radial geometry only	<a href="#">DIFFR</a>	R-direction diffusivities for the current input box
Molecular Diffusion option in Cartesian geometry only	<a href="#">DIFFTHT</a>	Theta-direction diffusivities for the current input box
Molecular Diffusion option in Cartesian geometry only	<a href="#">DIFFX</a>	X-direction diffusivities for the current input box
Molecular Diffusion option in Cartesian geometry only	<a href="#">DIFY</a>	Y-direction diffusivities for the current input box
Molecular Diffusion option in radial or Cartesian geometry	<a href="#">DIFFZ</a>	Z-direction diffusivities for the current input box
	<a href="#">EDITNNC</a>	Modifies non-neighbor connections. <a href="#">EDITNNCR</a> Replaces non-neighbor connections.
	<a href="#">ENDBOX</a>	Re-defines the current input box to encompass the whole grid.
	<a href="#">ENDFIN</a>	Terminates data for a local grid refinement.
	<a href="#">EQUALREG</a>	Sets an array to a constant value in a flux, <a href="#">MULTNUM</a> , or <a href="#">OPERNUM</a> region.
	<a href="#">EQUALS</a>	Sets specified arrays to specified constants within the current box.
	<a href="#">FILEUNIT</a>	Confirms the units of the data set.
	<a href="#">HMMULTPV</a>	History matching pore volume multipliers.
	<a href="#">HMMULTxx</a>	History matching transmissibility multipliers.
	<a href="#">IMPORT</a>	Imports binary data generated by a grid pre-processor.
	<a href="#">LGRCOPY</a>	Copy global values to local cells before operations on cells within an LGR.
	<a href="#">MAXVALUE</a>	Applies a maximum value to specified arrays within the current box.
	<a href="#">MINVALUE</a>	Applies a minimum value to specified arrays within the current box.
	<a href="#">MULTFLT</a>	Applies a transmissibility multiplier to a named fault defined by the <a href="#">FAULTS</a> keyword in the GRID section.
	<a href="#">MULTIPLY</a>	Multiplies specified arrays by specified constants within the current box.
	<a href="#">MULTIREG</a>	Multiplies an array by a constant in a given flux, <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
	<a href="#">MULTPV</a>	Multiplies the calculated pore volume within the current box.

Restriction	Keyword	Description
	MULTREGT	Multiplies transmissibilities between flux or multiplier regions.
	MULTREGP	Multiplies pore volumes between flux or multiplier regions.
	OPERATE	Performs arithmetic operations on arrays.
	OPERATER	Performs arithmetic operations on arrays for a specific region of the grid (defined using the <code>OPERNUM</code> keyword in the <code>GRID</code> section).
	PORV	Grid block pore volumes for the current input box.
	REFINE	Select local grid refinement for input data.
Thermal	ROCKV	Rock volume
Radial geometry only	TRANR	R direction transmissibilities for the current input box
Radial geometry only	TRANHT	Theta direction transmissibilities for the current input box
Cartesian geometry only	TRANX	X direction transmissibilities for the current input box
Cartesian geometry only	TRANY	Y direction transmissibilities for the current input box
Radial or Cartesian geometry	TRANZ	Z direction transmissibilities for the current input box

## ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
Radial geometry only	MULTR	Sets an additional R-direction transmissibility multiplier within the current box.
Radial geometry only	MULTR-	Sets an additional negative R-direction transmissibility multiplier within the current box
	MULTREGD	Multiplies diffusivities between flux or multiplier regions.
	MULTREGH	Multiplies thermal conductivity between flux or multiplier regions
Radial geometry only	MULTTHT	Sets an additional Theta-direction transmissibility multiplier within the current box
Radial geometry only	MULTTHT-	Sets an additional negative Theta-direction transmissibility multiplier within the current box
	MULTX	Sets an additional X-direction transmissibility multiplier within the current box.
	MULTX-	Sets an additional negative X-direction transmissibility multiplier within the current box
	MULTY	Sets an additional Y-direction transmissibility multiplier within the current box.
	MULTY-	Sets an additional negative Y-direction transmissibility multiplier within the current box.

Restriction	ECLIPSE 100 only keyword	Description
	MULTZ	Sets an additional Z-direction transmissibility multiplier within the current box.
	MULTZ-	Sets an additional negative Z-direction transmissibility multiplier within the current box.

## ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
	HEATTR	Heat transmissibility values in the R-direction.
	HEATTHT	Heat transmissibility values in the Theta-direction.
	HEATTX	Heat transmissibility values in the X-direction.
	HEATTY	Heat transmissibility values in the Y-direction.
	HEATTZ	Heat transmissibility values in the Z-direction.
Thermal	ROCKV	Rock volume

## 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

```

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.

ECLIPSE 300

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:

ECLIPSE 300

- The IKU three phase method.

ECLIPSE 300

- 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:

$S_{wco}$  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.

$S_{wcr}$  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  $P_{cog} = P_g - P_o$ .

ECLIPSE 100

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:

ECLIPSE 300

The three phase IKU relative permeability is available with the [IKU3P](#) keyword.

ECLIPSE 300

The three phase ODD3P relative permeability is available with the [ODD3P](#) keyword.

ECLIPSE 300

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:

Simulator	Keyword	Description
	<a href="#">STONE1</a>	Stone model 1 (modified) for three phase oil permeability.

Simulator	Keyword	Description
	<a href="#">STONE1EX</a>	Exponent term for Stone model 1.
	<a href="#">STONE2</a>	Stone model 2 (modified) for three phase oil permeability.
E300 only	<a href="#">BAKER1</a>	Baker interpolation method 1 for three phase permeability. Applies to any specified phase
E300 only	<a href="#">BAKER2</a>	Baker interpolation method 2 for three phase permeability. Applies to any specified phase.
E300 only	<a href="#">IKU3P</a>	IKU interpolation method. Applies to all phases.
E300 only	<a href="#">ODD3P</a>	ODD3P interpolation method. Applies to all phases.
	<a href="#">SOF3</a>	For oil in three-phase models.
E300 only	<a href="#">SGF3</a>	For gas in three-phase models.
E300 only	<a href="#">SWF3</a>	For water in three-phase models.

## 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*.

*ECLIPSE 300 only*

.For the [ODD3P](#) option, the primary, secondary and tertiary hysteresis curves are specified using the [PSTNUM](#), [ISTNUM](#) and [DSTNUM](#) keywords respectively. These replace the [SATNUM](#) and [IMBNUM](#) keywords which are used for the other hysteresis models.

## 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](#), [PVTO](#), [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 [PVTNUM](#) 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.

ECLIPSE 300 only

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  $P_{effl}$ , where  $P_{effl} = P_{fluid} - \text{Overburden}$
  - or against:
  - effective stress, where Effective Stress =  $-P_{effl}$

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

ECLIPSE 300

- 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 **ROCKTSIG** 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.

ECLIPSE 300 only

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:

- [PVC0](#) 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.

ECLIPSE 100

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.

ECLIPSE 300

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 *i* 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.

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

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

- PVTO and RSCONST ( T ). The constant  $R_s$  facility is only for dead oil.
- PVTG and RVCONST ( T ). The constant  $R_v$  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.

Phase Combinations	Keywords
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

Phase Combinations	Keywords
Undersaturated Oil and Constant Dissolved Gas, Water	either: <a href="#">PVDO</a> , <a href="#">RSCONST ( T ) .PVTW</a> or: <a href="#">PVCDO</a> , <a href="#">RSCONST ( T )</a> , <a href="#">PVTW</a> either: <a href="#">SWOF</a> or: <a href="#">SOF2</a> , <a href="#">SWFN</a>
Dry Gas	<a href="#">PVDG</a> or <a href="#">PVZG</a>
Dry Gas and Constant Vaporized Oil, Water	<a href="#">RVCONST ( T )</a> , <a href="#">PVDG</a> , <a href="#">PVTW</a> either: <a href="#">SGFN</a> , <a href="#">SWFN</a> or: <a href="#">SGWFN</a>

Table 2.7: PROPS keywords to be used with the different phase combinations in a black oil run

## 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 [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 [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 [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 [COMPS](#) in the RUNSPEC section:

ECLIPSE 300 only keyword	Description
<a href="#">CNAMEs</a>	Component names
<a href="#">TCRIT</a>	Critical temperatures
<a href="#">PCRIT</a>	Critical pressures
<a href="#">VCRIT</a>	Critical volumes
<a href="#">ZCRIT</a>	Critical Z-factors
<a href="#">VCRITVIS</a>	Critical volumes (for viscosities only)
<a href="#">MW</a>	Molecular weights
<a href="#">ACF</a>	Acentric factors
<a href="#">TBOIL</a>	Boiling points (for Zudkevitch Joffe)
<a href="#">DREF</a>	Reference densities (for Zudkevitch Joffe)
<a href="#">GREF</a>	Reference gravities (for Zudkevitch Joffe)

ECLIPSE 300 only keyword	Description
TREF	Reference temperatures (for Zudkevitch Joffe)
OMEGAA	$\Omega_{a_0}$ values for reservoir
OMEGAB	$\Omega_{b_0}$ values for reservoir
PARACHOR	Parachor values for surface tension evaluation
SSHIFT	Equation of state volume shifts for reservoir

Note that **either** Z-factors or critical volumes should be entered, as these are related by the relationship  $V_c = Z_c RT / P_c$ . 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 `LBCCOEF` 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:

```
BIC
0.01
0.01 0.005
0.02 0.0 0.001
0.001 0.0 0.001 0.0005 /
```

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`, `OMEGAAS`, `OMEGABS`, `BICS` and `SSHIFTS`.

Furthermore, the EoS at the surface (say ZI) 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 `KVTABtn`. 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

ECLIPSE 300

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 `XMFPV` 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

Restriction	Keyword	Description
Required if Surfactant option is active.	<code>ALSURFST</code>	Oil/water surface tension multipliers.

#### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keywords	Description
Required if Surfactant option is active.	<code>ALSURFAD</code>	Surfactant adsorption multipliers.
Required if Polymer option is active.	<code>ALPOLADS</code>	Polymer adsorption multipliers.
	<code>ALKADS</code>	Alkaline adsorption functions (tabulated).
	<code>ADSORP</code>	Alkaline adsorption functions (analytical).
Required if <code>ALKADS</code> or <code>ADSORP</code> is specified.	<code>ALKROCK</code>	Alkaline-rock properties.

## API tracking (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
API tracking only.	APIGROUP	Allows different groups of oil PVT tables to be used in different PVTNUM regions.

## Black Oil fluid PVT property data

### Common keywords

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

### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
PVCDO or PVDO required for dead or under saturated oil.	PVCDO	FVF and viscosity of dead oil with constant compressibility
PVCDO or PVDO required for dead or under saturated oil.	PVDO	FVF and viscosity of dead oil as functions of pressure.
Dead oil only.	RSCONST	Sets a constant $R_s$ for dead oil over the whole field.
Dead oil only.	RSCONSTT	Sets constant $R_s$ for dead oil, a different value in each PVT region.
PVDG or PVZG required for dry gas systems.	PVZG	Z-factor and viscosity of dry gas as functions of pressure.
Dry gas only.	RVCONST	Sets a constant $R_v$ for dry gas over the whole field.
Dry gas only.	RVCONSTT	Sets constant $R_v$ for dry gas, a different value in each PVT region.

**ECLIPSE 300 keywords**

Restriction	ECLIPSE 300 keyword	Description
Required	<a href="#">WATERTAB</a>	Water pressure tables. Alternative to PVTW for systems with water.

**Brine option****Common keywords**

Restriction	Keyword	Description
Required	<a href="#">PVTWSALT</a>	FVF, compressibility and viscosity of water as functions of salt concentration.
	<a href="#">BDENSITY</a>	Enables the surface density of water to vary with the salt concentration.

**ECLIPSE 100 only keywords**

Restriction	ECLIPSE 100 only keyword	Description
Brine option with dispersion only.	<a href="#">DSPDEINT</a>	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.
	<a href="#">IONXROCK</a>	Rock ion exchange parameters for cation exchange.
	<a href="#">IONXSURF</a>	Surfactant micelle ion exchange parameters for cation exchange.
	<a href="#">PLYESAL</a>	Coefficients for polymer effective salinity.
	<a href="#">SURFESAL</a>	Coefficients for surfactant effective salinity.

**Carbon Dioxide solution in aqueous phase (ECLIPSE 300)**

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

Restriction	Keyword	Description
<a href="#">CO2SOL</a> and <a href="#">GASSOL</a> options.	<a href="#">RSWVD</a>	Aqueous concentration of dissolved gases versus depth.
	<a href="#">DENAQA</a>	This keyword enables the default Ezrokhi coefficients for the aqua density calculation to be overwritten.
	<a href="#">VISCAQA</a>	This keyword enables the default Ezrokhi coefficients for the aqua viscosity calculation to be overwritten.

**CO2SOL only keywords**

- [SOLUBILI](#) Specify properties of water-CO<sub>2</sub> system.
- [SOLUBILS](#) Specify properties of water-CO<sub>2</sub> system with salt dependencies.
- [SOLUBILT](#) Specify properties of water-CO<sub>2</sub> system with temperature dependencies.
- [SOLUCO2H](#) Specify properties of water-CO<sub>2</sub> system with H<sub>2</sub>S dependencies.

- [SOLUH2SC](#) Specify properties of water-H<sub>2</sub>S system with CO<sub>2</sub> dependencies.

### GASSOL only keywords

- [SOLUAQA](#) Specify properties of a water-multi component gas system.

### CO2STORE only keywords

- [ACTCO2S](#) Activity coefficient model for CO<sub>2</sub>

### Carter-Tracy aquifer data

- [AQUTAB](#) Influence functions used by Carter-Tracy aquifers.

### Chemical reaction keywords (ECLIPSE 300)

Restriction	ECLIPSE 300 only keyword	Description
	<a href="#">EQLDKVCR</a>	K-values for equilibrium deviation reactions
	<a href="#">EQLDREAC</a>	Equilibrium deviation reaction rates
	<a href="#">REACTACT</a>	Reaction activation molar energy
Thermal	<a href="#">REACENTH</a>	Reaction molar enthalpy
	<a href="#">REACCORD</a>	Reaction rate component order
	<a href="#">REACPHA</a>	Reactant phase
	<a href="#">REACPORD</a>	Reaction rate porosity order
	<a href="#">REACRATE</a>	Reaction rate constant
Solid	<a href="#">REACSMAX</a>	Reaction rate solid saturation limit.
Solid	<a href="#">REACSORD</a>	Reaction rate solid order.
	<a href="#">REACTRAC</a>	Tracer dependent reaction rate terms
	<a href="#">STOPROD</a>	Stoichiometric coefficients of products
	<a href="#">STOREAC</a>	Stoichiometric coefficients of reactants

### Coal bed methane (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
Either COALADS or COALPP should be specified.	<a href="#">COALADS</a>	Gas/solvent relative adsorption data.
Either COALADS or COALPP should be specified.	<a href="#">COALPP</a>	Gas/solvent partial pressure data.
	<a href="#">DIFFCOAL</a>	Gas diffusion data.
	<a href="#">LANGMPL</a>	Pressure scaling factor for the adsorption capacity for each grid cell.

Restriction	ECLIPSE 100 only keyword	Description
	LANGMUIR	Tables of coal surface gas concentration.
	LANGSOLV	Tables of coal solvent gas concentration.
	MLANG	Maximum surface gas concentration.
	MLANGSLV	Maximum solvent surface concentration.
	ROCKPAMA	Palmer-Mansoori rock model data.
Palmer-Mansoori rock model.	TPAMEPS	Specify strain term as a function of adsorbed gas concentration
Palmer-Mansoori rock model.	TPAMEPSS	Specify strain term as a function of adsorbed solvent concentration

### Coal bed methane (ECLIPSE 300)

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

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

Restriction	ECLIPSE 300 only keyword	Description
	ACF	Acentric factors. Required.
Defaults to ACF if not specified	ACFS	Acentric factors for surface EoS
Required	BIC	Binary interaction coefficients for EoS.
Defaults to BIC if not specified	BICS	Binary interaction coefficients for surface EoS
Required	CALVAL / CALVALR	Component calorific values and component calorific values for multiple equation of state regions.

Restriction	ECLIPSE 300 only keyword	Description
For calorific value rate control of groups and wells (and the <code>GASWAT</code> option). Required	<code>CNAMES</code>	Component names.
<code>DENSITY</code> or <code>GRAVITY</code> is required only for water, if present.	<code>DENSITY</code>	Stock tank fluid densities.
<code>DENSITY</code> or <code>GRAVITY</code> is required only for water, if present.	<code>GRAVITY</code>	Stock tank fluid gravities.
Required for Zudkevitch Joffe only.	<code>DREF</code>	Reference densities (for Zudkevitch Joffe EoS).
	<code>FACTLI</code>	Modifies the Li critical temperature correlation.
Required (but assumes default values if omitted).	<code>GREF</code>	Reference gravities (for Zudkevitch Joffe).
Used only if the default coefficients in the Lorentz-Bray-Clark viscosity correlation are changed.	<code>LBCCOEF</code>	Set non-default LBC coefficients.
Used only if the default coefficients in the Lorentz-Bray-Clark viscosity correlation are changed.	<code>LBCCOEFR</code>	Set non-default LBC coefficients for multiple equation of state regions.
	<code>LILIM</code>	Specifies the range for Li-interpolation.
	<code>MW</code>	Molecular weights.
Defaults to <code>MW</code> if not specified. Required	<code>MWS</code>	Molecular weights for surface EoS.
	<code>NCOMPS</code>	Confirm number of components. Confirms the value of <code>NCOMPS</code> set in <code>RUNSPEC</code> .
	<code>OMEGAA</code>	$\Omega_{a0}$ values for reservoir.
Defaults to <code>OMEGAA</code> if not specified. Required.	<code>OMEGAAS</code>	$\Omega_a$ values for surface EoS.
	<code>OMEGAB</code>	$\Omega_{b0}$ values for reservoir. Assumes default values if omitted.
Defaults to <code>OMEGAB</code> if not specified. Required	<code>OMEGABS</code>	$\Omega_b$ values for surface EoS.
	<code>PCRIT</code>	Critical pressures.
Defaults to <code>PCRIT</code> if not specified. Required	<code>PCRITS</code>	Critical pressures for surface EoS.
	<code>PEDERSEN</code>	Allows use of Pedersen viscosity calculation.
Required for Pedersen et al. viscosity calculation only.	<code>PEDTUNE</code>	Provides a single set of multiplicative tuning parameters to modify the mixture molecular weight calculation used in the Pedersen et al. viscosity calculation.

Restriction	ECLIPSE 300 only keyword	Description
Optional for Pedersen et al. viscosity calculation only.	PEDTUNER	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.
Optional for Pedersen et al. viscosity calculation only.	PREF	Reference pressures for the Thermal option.
	PREFS	Reference pressures for surface EoS in the Thermal option.
	PRCORR	Request modified Peng-Robinson EoS.
Optional when using PR EoS	RTEMP	Specify constant reservoir temperature.
RTEMP or TEMPVD is required	SSHIFT	Equation of state volume shifts for reservoir. Defaults to 0 if not specified.
Defaults to SSHIFT if not specified.	SSHIFTS	Equation of state volume shifts for surface EoS.
	STCOND	Specify standard conditions. Defaults if not specified.
	TBOIL	Boiling points (for Zudkevitch Joffe EoS).
Required for Zudkevitch Joffe only Defaults to TBOIL if not specified.	TBOILS	Boiling points for surface (for Zudkevitch Joffe EoS).
	TCRIT	Critical temperatures.
Defaults to TCRIT if not specified.	TCRITS	Critical temperatures for surface EoS.
RTEMP or TEMPVD is required	TEMPVD	Specify constant temperature versus depth.
Required for Zudkevitch Joffe only	TREF	Reference temperatures (for Zudkevitch Joffe).
Defaults to VCRIT if not specified.	VCRITVIS	Critical volumes (for viscosities only).
	ZCRIT	Critical Z-factors.
Defaults to ZCRIT if not specified.	ZCRITS	Critical Z-factors for surface EoS.
Defaults to ZCRIT or if not specified	ZCRITVIS	Critical Z-factors (for viscosities only).

## 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

Restriction	Keyword	Description
Saturation table end point scaling option only.	ENKRVD	Variation of end point relative permeability values with depth.
Saturation table end point scaling option only.	ENPCVD	Variation of maximum capillary pressure values with depth.
Saturation table end point scaling option only.	ENPTVD	Variation of end point saturation values with depth.
Saturation table end point scaling option only.	EPSDBGS	Writes scaled relative permeability curves to the Debug file, for multiple sets of grid blocks.
Saturation table end point scaling option only.	EPSDEBUG	Writes scaled relative permeability curves to the Debug file, for grid blocks in the specified box.
Saturation table end point scaling with hysteresis only.	IKRG, IKRGR, IKRW, IKRWR, IKRO, IKROG, IKRORW	Imbibition curve end point relative permeability values, equivalent to keywords KRG, KRGR, KRW and KRWR for example.
Saturation table end-point scaling option and WAG hysteresis option only.	IKRWH	Scales three-phase water relative permeability values at maximum water saturation, for cells in the current input box.
Saturation table end-point scaling option and WAG hysteresis option only.	IKRWRH	Scales three-phase water relative permeability values at residual oil saturation, for cells in the current input box.
Saturation table end point scaling with hysteresis only.	ISGL, ISGCR, ISGU, ISWL, ISWCR, ISWU, ISOGCR, ISOWCR	Imbibition curve end point scaling data, equivalent to keywords including SGL, SGCR and SGU.
Saturation table end point scaling option only.	KRG	Scales gas relative permeability values at maximum gas saturation, for cells in the current input box.
Saturation table end point scaling option only.	KRGR	Scales gas relative permeability values at residual oil saturation, for cells in the current input box.
Saturation table end point scaling option only.	KRO	Scales oil relative permeability values at maximum oil saturation, for cells in the current input box.
Saturation table end point scaling option only.	KROG	Scales oil relative permeability values at critical gas saturation, for cells in the current input box
Saturation table end point scaling option only.	KROW	Scales oil relative permeability values at critical water saturation, for cells in the current input box.
Saturation table end point scaling option only.	KRW	Scales water relative permeability values at maximum water saturation, for cells in the current input box.

Restriction	Keyword	Description
Saturation table end point scaling and WAG hysteresis option only.	KRWH	Scales two-phase water relative permeability values at maximum water saturation, for cells in the current input box.
Saturation table end point scaling option only.	KRWR	Scales water relative permeability values at residual oil saturation, for cells in the current input box.
Saturation table end point scaling option and WAG hysteresis option only.	KRWRH	Scales two-phase water relative permeability values at residual oil saturation, for cells in the current input box.
Saturation table end point scaling option only.	PCG	Scales the maximum gas capillary pressure, for cells in the current input box.
Saturation table end point scaling option only.	PCW	Scales the maximum water capillary pressure, for cells in the current input box.
Saturation table end point scaling option only.	PPCWMAX	Limits the calculated PCW values when the SWATINIT option is used.
Saturation table end point scaling option only.	SCALECRS	Requests alternative end point scaling method.
Saturation table end point scaling option only.	SGCR	Critical gas saturation for grid cells in the current input box.
Saturation table end point scaling option only.	SGL	Connate gas saturation for grid cells in the current input box.
Saturation table end point scaling option only.	SGU	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.
Saturation table end point scaling option only using the IKU three-phase relative permeability model with IKU3P in the PROPS section	SWGCR	Residual water-in-gas saturation for grid cells in the current input box..
Saturation table end point scaling option only.	SOWCR	Residual oil-in-water saturation for grid cells in the current input box.
Saturation table end point scaling option only.	SWATINIT	Initial water saturation, for grid cells in the current input box, to scale water $P_c$ curves.
Saturation table end point scaling option only.	SWCR	Critical water saturation for grid cells in the current input box.
Saturation table end point scaling option only.	SWL	Connate water saturation for grid cells in the current input box.
Saturation table end point scaling option only.	SWU	Maximum water saturation for grid cells in the current input box.
	TOLCRIT	Controls the method of determining the table critical saturations in pathological end point scaling cases.
Mobile fluid correction cases.	TZONE	Controls the variation of critical end-point with depth.

**ECLIPSE 100 only keywords**

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

**ECLIPSE 300 only keywords**

Restriction	ECLIPSE 300 only keyword	Description
Saturation table end point scaling option only using the IKU three-phase relative permeability model with <a href="#">IKU3P</a> in the PROPS section.	SGWCR	Residual gas-in-water saturation for grid cells in the current input box.

**Environmental tracers (ECLIPSE 100)**

Restriction	ECLIPSE 100 only keywords	Description
Optional, required for partitioned tracers	TRACERKP	Partitioning function K(P) for partitioned tracers.
Optional, required for partitioned tracers	TRACERKM	Partitioning functions K(P) for generalized, multi-partitioned tracers.
Optional, required for any adsorbing tracer	TRADS	Tracer adsorption functions (tabulated).

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

Restriction	Keyword	Description
Functional model	FOAMFCN	Gas mobility reduction dependence upon capillary number
Functional model	FOAMFRM	Gas reference mobility reduction factor
Functional model	FOAMFSC	Gas mobility reduction dependence upon surfactant concentration
Functional model	FOAMFSO	Gas mobility reduction dependence upon oil saturation
Functional model	FOAMFST	Gas-water surface tension dependence upon concentration
Functional model	FOAMFSW	Gas mobility reduction dependence upon water saturation

### ECLIPSE 100 only keywords

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

## GASWAT option (ECLIPSE 300)

The keywords are:

Restriction	ECLIPSE 300 only keyword	Description
	AQCOEF	A-coefficient terms for water component

Restriction	ECLIPSE 300 only keyword	Description
	<a href="#">BICAQ1</a> <a href="#">BICAQ2</a> <a href="#">BICAQ3</a>	Binary interaction coefficients for aqueous phase
Required for the GASWAT option	<a href="#">CALVAL</a> , <a href="#">CALVALR</a>	Component calorific values and component calorific values for multiple equation of state regions.
Required for the GASWAT option	<a href="#">GSF</a>	Gas saturation functions.
Required for the GASWAT option	<a href="#">WSF</a>	Water saturation functions.
	<a href="#">SALINITR</a> , <a href="#">SALINITY</a>	Reservoir salinity and regional reservoir salinity

## Geomechanics (ECLIPSE 300)

ECLIPSE 300 only keyword	Description
<a href="#">BIOTCTAB</a>	Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress.
<a href="#">GEOYLDF</a>	Defines parameters for the plastic yield function in each geomechanics region.
<a href="#">PERMSTAB</a>	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.
<a href="#">SCDATABG</a>	Defines a scale damage table to be used together with the proppant flowback feature, keyword <a href="#">PROPFLOW</a> .

## GI pseudo-compositional model (ECLIPSE 100)

- [BGGI](#) Variation of saturated gas FVF with pressure and  $G_i$ .
- [BOGI](#) Variation of saturated oil FVF with pressure and  $G_i$ .
- [GIALL](#) Variation of saturated properties with pressure and  $G_i$
- [GINODE](#)  $G_i$  nodal values. Required
- [RSGI](#) Variation of saturated GOR with pressure and  $G_i$ .
- [RVGI](#) Variation of saturated OGR with pressure and  $G_i$ .

## Gradient option (ECLIPSE 100 special option)

Restriction	ECLIPSE 100 only keyword	Description
	<a href="#">HMMROCK</a>	Cumulative multipliers on rock compressibility.
	<a href="#">HMMROCKT</a>	Cumulative modifiers of rock compaction parameters.
	<a href="#">HMPPROPS</a>	Section heading for end-point scaling modifiers.

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.	HMREF	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](#), [KVTABLE](#), [KVTABCn](#) is required for the [KVALUES](#) option.

ECLIPSE 300 only keyword	Description
<a href="#">KVGEN</a>	Specify that K-values are to be generated internally.
<a href="#">KVTABLE</a>	Tables of K-values.
<a href="#">KVTABCn</a> , <a href="#">KVMF</a> , <a href="#">KVCOMP</a>	Composition dependent K-values.

## Low-Salinity option (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
Required	<a href="#">LSALTFNC</a>	Tables of salt concentration versus fraction of high-salinity and low-salinity saturation functions.

## Miscible flood option (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
Miscible Flood option only	<a href="#">MSFN</a>	Allows input of relative permeability curves for miscible displacement. If not entered, a linear function is used.
Miscible Flood option only	<a href="#">PMISC</a>	Enables miscibility to be modeled as a function of pressure.

Restriction	ECLIPSE 100 only keyword	Description
Miscible Flood option only. Required	SDENSITY	Input of the solvent surface density.
	SGCWMIS	Miscible critical gas saturation tables.
Miscible Flood option only.	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	DIFFDIP	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.	DIFFGAS	Activity corrected gas diffusion coefficients.
Molecular Diffusion option only. Required.	DIFFWAT	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.

Restriction	ECLIPSE 300 only keyword	Description
Molecular Diffusion option only.	<a href="#">DIFFCOG</a>	Oil-gas cross phase diffusion coefficients.
Molecular Diffusion option only.	<a href="#">DIFFCGO</a>	Gas-oil cross phase diffusion coefficients.
	<a href="#">DIFFTGAS</a>	Gas thermal diffusion coefficients.
	<a href="#">DIFFTOIL</a>	Oil thermal diffusion coefficients.

## NGLs (ECLIPSE 300)

- [DNGL](#) Partial densities for NGLs.

## Non-Darcy Flow (ECLIPSE 300)

Restriction	ECLIPSE 300 only keyword	Description
Non-Darcy Flow option. Required	<a href="#">VDFLOW</a>	Specify velocity dependent flow coefficient.
Non-Darcy Flow option. Required	<a href="#">VDFLOWR</a>	Specify velocity dependent flow coefficient for each region.
optional if item 5 of <a href="#">VELDEP</a> is non-zero	<a href="#">VDKRG</a>	Gas velocity dependent relative permeability data. Relevant only for gas velocity dependent relative permeabilities.
	<a href="#">VDKRC</a>	Gas velocity dependent relative permeability data
Relevant only for oil velocity dependent relative permeabilities.	<a href="#">VDKRO</a>	Oil velocity dependent relative permeability data.

## 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

Restriction	ECLIPSE 100 Polymer flood keyword	Description
Required.	<a href="#">PLMIXPAR</a>	Todd-Longstaff mixing data for the Polymer Model.
	<a href="#">PLYADS</a>	Polymer adsorption functions (tabulated).
	<a href="#">PLYADSS</a>	Polymer adsorption functions varying with salinity (tabulated).

Restriction	ECLIPSE 100 Polymer flood keyword	Description
	ADSORP	Polymer adsorption functions (analytical) with optional salinity dependence.
Required	PLYMAX	Polymer/salt concentrations for mixing calculations.
	PLYROCK	Polymer-rock properties.
	PLYSHEAR	Polymer shear thinning/thickening data.
	PLYVISC	Polymer solution viscosity function.
	PLYVISCS	Polymer/salt solution viscosity function.
	SALTNODE	Salt concentration nodes for polymer solution viscosity.
	ADSALNOD	Salt concentration nodes for polymer solution adsorption
	PLYATEMP	Polymer adsorption functions varying with temperature (tabulated).

### 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

Restriction	Keyword	Description
SGOF or SLGOF required with Family (i) in systems with both gas and oil.	<a href="#">SGOF</a>	Gas relative permeability and capillary pressure and oil relative permeability in gas at connate water, as functions of $S_g$ .
SGOF or SLGOF required with Family (i) in systems with both gas and oil.	<a href="#">SLGOF</a>	Gas relative permeability and capillary pressure and oil relative permeability in gas at connate water, as functions of $S_{liq}$ .
Required with Family (i) in systems with both water and oil.	<a href="#">SWOF</a>	Water relative permeability and capillary pressure oil relative permeability in water, as functions of $S_w$ .
Required with Family (ii) in two or three-phase systems with gas, unless SGWFN is used in a gas-water run.	<a href="#">SGFN</a>	Gas relative permeability and capillary pressure as functions of $S_g$ .

Restriction	Keyword	Description
Required with Family (ii) in two or three-phase systems with water, unless SGWFN is used in a gas-water run.	SWFN	Water relative permeability and capillary pressure as functions of $S_w$ .
Required with Family (ii) for two-phase systems with oil.	SOF2	Oil relative permeability as a function of $S_o$ .
SOF3 or SOF32D required with Family (ii) for three-phase systems.	SOF3	Oil relative permeability as a function of $S_o$ .
	STONE	Use Stone's modified formula number 2 to calculate the three phase oil relative permeability.
	STONE1	Use Stone's modified formula number 1 to calculate the three phase oil relative permeability.
	STONE1EX	Apply an exponent to the saturation terms in Stone's modified formula number 1 method for calculating the three phase oil relative permeability.
	STONE2	Use Stone's modified formula number 2 to calculate the three phase oil relative permeability.
Optional, for STONE1 method only.	SOMGAS	Specifies the minimum oil saturation used in the STONE1 method as a function of the gas saturation.
Optional, for STONE1 method only.	SOMWAT	Specifies the minimum oil saturation used in the STONE1 method as a function of the water saturation.
	SGF32D	Gas relative permeability (for three-phase flows) as 2-D table.
	SWF32D	Water relative permeability (for three-phase flows) as 2-D table.
Optional, for Hysteresis option only.	WAGHYSTR	Activates the WAG Hysteresis option.

### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
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.)	SGWFN	Gas-water relative permeability and capillary pressure, as functions of $S_g$ .
SOF3 or SOF32D required with Family (ii) for three-phase systems	SOF32D	Oil relative permeability as functions of $S_w$ and $S_g$ in a 2D table.
Optional, if SURFTENS is specified in keyword SATOPTS, or if the Drift Flux Model is used in multisegment wells.	STOG	Tables of oil/gas surface tension.

Restriction	ECLIPSE 100 only keyword	Description
Optional, if SURFTENS is specified in keyword SATOPTS, or if the Drift Flux Model is used in multisegment wells.	STOW	Tables of oil/water surface tension.
Optional, if the Drift Flux Model is used in multisegment wells.	STWG	Tables of water/gas surface tension.
Required for Hysteresis option.	EHYSTR	Curvature parameters and model selection for hysteresis.
Optional, for Hysteresis option only	EHYSTRR	Hysteresis parameters by region.
Optional, for Hysteresis option only.	HYSTCHCK	Activates additional consistency checks between drainage and imbibition curve endpoints when using the Hysteresis option.

### ECLIPSE 300 only keywords

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

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

## 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

Restriction	Keyword	Description
	OVERBURD	Overburden pressure tables
	RKTRMDIR	Allows directional transmissibility multipliers to be defined in ROCKTAB tables
	ROCKTAB	Rock compaction tables Required if using standard Rock Compaction option
Required if using Hysteretic Rock Compaction option	ROCKTABH	Hysteretic rock compaction tables
	ROCKTHSG	Hysteretic rock compaction tables in dual porosity runs
	ROCKTSIG	Rock compaction tables in dual porosity runs
Optional, for Rock Compaction option only	ROCKOPTS	Controls options associated with the rock compaction facilities

Restriction	Keyword	Description
Optional, for the Coal Bed Methane option only	ROCKPAMA	Palmer-Mansoori rock model data
	TPAMEPS	Volumetric strain values to be used by ROCKPAMA.

### ECLIPSE 100 only keywords

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

### ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
	RKPVMNMX	Allows pore volume multiplier limits to be set for the BOBERG model.
Geomechanics. Required if <b>not</b> using Rock Compaction option. Optional when using the second porosity-stress relation. Rock Compaction option only.	ROCK	Rock compressibility
Optional, for the Coal Bed Methane option only.	ROCKPAME	Palmer-Mansoori rock model data
	ROCKPMAT	Thermal Palmer-Mansoori rock model data
	ROCKTRMX	Defines transmissibility variation values in the X-direction in order to define transmissibility as an exponential function of porosity.
	ROCKTRMY	Defines transmissibility variation values in the Y-direction in order to define transmissibility as an exponential function of porosity.

Restriction	ECLIPSE 300 only keyword	Description
	ROCKTRMZ	Defines transmissibility variation values in the Z-direction in order to define transmissibility as an exponential function of porosity.

### Solid option (ECLIPSE 300)

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

### Solvent Model (ECLIPSE 100)

ECLIPSE 100 only keyword	Description
MISC	Miscibility function tables.
MSFN	Miscible saturation functions.
PMISC	Pressure dependent miscibility tables.
PVDS	PVT properties of the solvent.
SDENSITY	Solvent density at surface conditions.

ECLIPSE 100 only keyword	Description
SGCWMIS	Miscible critical gas saturation tables.
SSFN	Gas/solvent saturation functions.

## Surface tension effects (ECLIPSE 300)

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

## Surfactant option

### ECLIPSE 100 only keywords

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

**ECLIPSE 300 only keywords**

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

**Temperature option**

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

**Thermal option (ECLIPSE 300)**

Restriction	ECLIPSE 300 only keyword	Description
Requires KWTABTn	<a href="#">AMFVD</a>	Aqueous mole fractions versus depth
	<a href="#">CREF</a>	Component compressibilities
	<a href="#">CCTYPE</a>	Component liquid compressibility type
	<a href="#">CCTYPES</a>	Component liquid compressibility type at surface conditions
	<a href="#">CGDTYPE</a>	Component gas (EoS) density model
	<a href="#">CGVTYPE</a>	Component gas (EoS LBC) viscosity model
	<a href="#">CODTYPE</a>	Component oil (EoS) density model

Restriction	ECLIPSE 300 only keyword	Description
	COVTYPE	Component oil (EoS LBC) viscosity model
	CVTYPE	Component volatility (dead, live, gaseous)
	CVTYPES	Component volatility (dead, live, gaseous) at surface conditions
	DREF	Fluid densities at the reference pressures and temperatures.
	DREFS	Reference densities for use in the surface calculations
	DREFT	alias for DREF
	DREFTS	alias for DREFS
	ENKRVT	Endpoint relative permeabilities versus temperature
	ENPCVT	Endpoint capillary pressures versus temperature
	ENPTVT	Endpoint saturations versus temperature
	GASVISCF	Gas viscosity function of temperature
	GASVISCT	Gas viscosity table as a function of temperature
	HEATVAP	Component heat of vaporization at standard temperature
	HEATVAPE	Component heat of vaporization
	HEATVAPS	Heat of vaporization exponent
	KVCR	K-value correlation parameters
	KVCRS	K-value correlation parameters at surface conditions
	KVGEN	K-values generated from an EoS.
	KVCRWAT	Water K-value correlation parameters
	KVTABTn	Tables of K-values
	KVTEMP	Temperatures for K-value tables
	KWTABTn	Tables of aqua Kw-values
	KWTEMP	Temperatures for Kw-value tables
	KVWI	Wilson K-value correlation parameters
	OILCOMPR	Oil compressibility, expansion coefficient
	OILMW	Oil molecular weight
	OILSPECH	Oil specific heat
	OILVINDX	Oil viscosity index
	OILVISCC	Oil viscosity-temperature correlations
	OILVISCF	Oil viscosity function
	OILVISCT	Oil viscosity table as a function of T
	OILVTIM	Interpolation method for OILVISCT data
	PREF	Component reference pressures
	PREFS	Component reference pressures for surface EoS
	PREFT	Alias for PREF

Restriction	ECLIPSE 300 only keyword	Description
	PREFTS	Alias for PREFS
	REACENTH	Reaction molar enthalpy
	SPECHA	Component oil specific heat, 1 <sup>st</sup> coefficient
	SPECHB	Component oil specific heat, 2 <sup>nd</sup> coefficient
	SPECHG	Component gas specific heat, 1 <sup>st</sup> coefficient
	SPECHH	Component gas specific heat, 2 <sup>nd</sup> coefficient
	SPECHS	Component solid specific heat, 1 <sup>st</sup> coefficient Solid option
	SPECHT	Component solid specific heat, 2 <sup>nd</sup> coefficient Solid option
Requires KWTABTn	SPECHW1	Component aqua specific heat, 1 <sup>st</sup> coefficient
Requires KWTABTn	SPECHW2	Component aqua specific heat, 2 <sup>nd</sup> coefficient
	THANALB	Analytic water and steam densities
	THANALH	Analytic water and steam enthalpy values
	THERMEX1	Analytic oil viscosities
	THANALV	Component thermal expansion coefficient
	THSVC	Steam viscosity coefficients
	THTABB	Tabular water and steam densities
	THWVC	Water viscosity coefficients
	TRANGE	Expected temperature limits
	TREF	Reference temperatures for fluid densities
	TREFS	Reference temperatures for use in the surface calculations
	TREFT	alias for TREF
	TREFTS	alias for TREFS
	VISCREF	Reference conditions for the viscosity/temperature tables
	WATDENT	Water density temperature dependence
	WATVISCT	Inputs tables of water viscosity versus temperature.
	ZFACT1	Z-factor first coefficient
	ZFACT1S	Surface Z-factor first coefficient
	ZFACTOR	Z-factors zeroth coefficient
	ZFACTORS	Surface Z-factor zeroth coefficient

## 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

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

### Vertical equilibrium (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
Vertical Equilibrium option only.	<a href="#">VEFRAC</a>	Fraction of VE relative permeabilities used in place of rock curves, for the entire grid.
Vertical Equilibrium option only	<a href="#">VEFRACV</a>	Fraction of VE relative permeabilities used in place of rock curves, for the current box.
Vertical Equilibrium option only	<a href="#">VEFRACP</a>	Fraction of VE pseudo capillary pressures used in place of rock curves, for the entire grid.
Vertical Equilibrium option only	<a href="#">VEFRACPV</a>	Fraction of VE pseudo capillary pressures used in place of rock curves, for the current box.

### Water component option (ECLIPSE 300)

ECLIPSE 300 only keyword	Description
<a href="#">CREFW</a>	Compressibility of water components at the reference pressure.
<a href="#">CREFWS</a>	Compressibility of water components for surface calculations.
<a href="#">CWTYPE</a>	Water component type
<a href="#">DREFW</a>	Density of water components at the reference pressure

ECLIPSE 300 only keyword	Description
DREFWS	Density of water components for surface calculations
MWW	Molecular weight of water components
MWWS	Molecular weight of water components for surface calculations
PREFW	Reference pressure for water component density and viscosity calculations
PREFWS	Reference pressure for water component density calculations at surface conditions.
VREFW	Viscosity and viscosibility for water components at the reference pressure
WNAMEs	Water component names

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

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

### Operational keywords

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

## Common keywords

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

## 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

Keyword	Description
ACTCO2S	Activity coefficient model for CO <sub>2</sub> . CO2STORE only
ADD	Adds specified constants to specified arrays in the current box.
ADDREG	Adds by MULTNUM, FLUXNUM or OPERNUM region.
AQUTAB	Carter-Tracy aquifer influence functions.
BOX	Redefines the current box.
COPYREG	Copy by MULTNUM, FLUXNUM or OPERNUM region.

Keyword	Description
COPY	Copies data in the current box from one specified array to another.
COPYBOX	Copies data from one box of cells to another box of the same size.
DENSITY	Stock tank fluid densities.
ENDBOX	Redefines the current box to encompass the whole grid.
ENDFIN	End data for a local grid refinement.
ENKRVD	Variation of end point relative permeability values with depth.
ENPCVD	Variation of maximum capillary pressure values with depth.
ENPTVD	Variation of saturation table end points with depth.
EPSDBGS	Outputs scaled relative permeability curves to the Debug file for multiple sets of grid block.
EPSDEBUG	Outputs scaled relative permeability curves to the Debug file.
EQLDKVCR	K-values for equilibrium deviation reactions.
EQLDREAC	Equilibrium deviation reaction rates.
EQUALREG	Set by MULTNUM, FLUXNUM or OPERNUM region.
EQUALS	Sets specified arrays to specified constants in the current box.
ESSNODE	Effective salinity nodes for salt sensitive surfactant flood studies.
FILEUNIT	Confirms the units of the data set.
FOAMFRM	Gas reference mobility reduction factor (functional model).
FOAMFSC	Gas mobility reduction dependence upon surfactant concentration (functional model).
FOAMFSO	Gas mobility reduction dependence upon oil saturation (functional model).
FOAMFST	Gas-water surface tension dependence upon concentration (functional model).
FOAMFSW	Gas mobility reduction dependence upon water saturation (functional model).
GRAVCONS	Sets a user-defined gravity constant.
GRAVITY	Stock tank fluid gravities.
IKRG, IKRGR, IKRW, IKRWR, IKRO, IKROGR, IKRORW	Imbibition curve end point relative permeability values, equivalent to KRG for example.
IMPORT	Imports binary data generated by a grid pre-processor.
KRG	Scales gas relative permeability values at maximum gas saturation, for cells in the current input box.
KRGR	Scales gas relative permeability values at residual oil saturation, for cells in the current input box.
KRO	Scales oil relative permeability values at maximum oil saturation, for cells in the current input box.
KRORG	Scales oil relative permeability values at critical gas saturation, for cells in the current input box.

Keyword	Description
KRORW	Scales oil relative permeability values at critical water saturation, for cells in the current input box.
KRW	Scales water relative permeability values at maximum water saturation, for cells in the current input box.
KRWR	Scales water relative permeability values at residual oil saturation, for cells in the current input box.
LCUNIT	Set units for linear combination of phases.
MAXVALUE	Applies a maximum value to specified arrays within the current box.
MINVALUE	Applies a minimum value to specified arrays within the current box.
MULTIPLY	Multiplies specified arrays by constant values in the current box.
MULTIREG	Multiply by <a href="#">MULTNUM</a> , <a href="#">FLUXNUM</a> or <a href="#">OPERNUM</a> region.
OILVISCT	Tables of oil viscosity versus temperature.
OILVTIM	Interpolation method for <a href="#">OILVISCT</a> data.
OPERATE	Performs arithmetic operations on arrays.
OPERATER	Performs arithmetic operations on arrays for a specific region of the grid (defined using the <a href="#">OPERNUM</a> keyword in the <a href="#">GRID</a> section).
OVERBURD	Overburden pressure tables.
PCRITS	Critical pressures for surface EoS.
PCW	Scales the maximum water capillary pressure, for cells in the current input box.
PLYVISC	Polymer solution viscosity function.
PLYVISCS	Polymer/salt solution viscosity function.
PMAX	Maximum pressure expected in the simulation.
PPCWMAX	Limits the calculated PCW values when the <a href="#">SWATINIT</a> option is used.
PVCO	FVF and viscosity of live oil as functions of bubble point pressure.
PVDO	FVF and viscosity of dead oil as functions of pressure.
PVTW	FVF, compressibility and viscosity of water.
PVZG	Z-factor and viscosity of dry gas as functions of pressure.
REFINE	Specify data for local grid refinement.
RKTRMDIR	Allows directional transmissibility multipliers to be defined in <a href="#">ROCKTAB</a> tables.
ROCK2D	Two-dimensional water induced rock compaction tables.
ROCKOPTS	Controls options associated with the rock compaction facilities.
RPTPROPS	Report levels for PROPS data.
RTEMP	Specifies a constant reservoir temperature.
SCALECRS	Requests alternative end point scaling method.
SGCR	Critical gas saturation for grid cells in the current input box.
SGF32D	Gas relative permeability (for three-phase flows) as a 2D table.
SGFN	Gas relative permeability and capillary pressure as functions of $S_g$ .

Keyword	Description
SGL	Connate gas saturation for grid cells in the current input box.
SGOF	Gas relative permeability and cap pressure, and oil relative permeability in gas, versus $S_g$ .
SGU	Maximum gas saturation for grid cells in the current input box.
SLGOF	Gas relative permeability and cap pressure, and oil relative permeability in gas, versus $S_{liq}$ .
SOF2	Oil relative permeability as a function of $S_o$ in two phase system.
SOF3	Oil relative permeability as a function of $S_o$ in three phase system.
SOF32D	Oil relative permeability as a 2D function of $S_w$ and $S_g$ in three phase systems.
SOMGAS	Minimum oil saturation for STONE1 method, as a function of the gas saturation.
SOMWAT	Minimum oil saturation for STONE1 method, as a function of the water saturation.
SOWCR	Residual oil-in-water saturation for cells in the current box.
STONE	Stone's formula for three phase oil relative permeability.
STONE1	Stone's formula 1 for three phase oil relative permeability.
STONE1EX	Exponent applied to saturation terms in Stone's formula 1 for three phase oil relative permeability.
STONE2	Stone's formula 2 for three phase oil relative permeability.
SURFST	Oil-water surface tension in the presence of surfactant.
SURFCAPD	Capillary de-saturation data.
SWATINIT	Initial water saturation, for cells in the current input box, to scale the water Pc curves.
SWCR	Critical water saturation for grid cells in the current input box.
SWF32D	Water relative permeability (for three-phase flows) as 2-D table.
SWFN	Water relative permeability and capillary pressure as functions of $S_w$ .
SWL	Connate water saturation for grid cells in the current input box.
SWOF	Water relative permeability and cap pressure, and oil relative permeability in water, versus $S_w$ .
SWU	Maximum water saturation for grid cells in the current input box.
TOLCRIT	Controls the method of determining the table critical saturations in pathological end point scaling cases.
TPAMEPS	Volumetric strain values in ROCKPAMA.
TRACER	Names and phases of passive tracers.
TZONE	Controls the variation of critical end-point with depth.
VISCREP	Reference conditions for viscosity versus temperature tables.
WATVISCT	Tables of water viscosity versus temperature.

## ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
ADSORP	Analytical adsorption functions.
ALKADS	Alkaline adsorption functions.
ALKROCK	Alkaline-rock properties.
ALPHAI	Transport coefficient tables for imbibition processes.
ALPOLADS	Polymer adsorption multipliers as a function of alkaline concentration.
ALSURFAD	Surfactant adsorption multipliers as a function of alkaline concentration.
ALSURFST	Oil/water surface tension multipliers as a function of alkaline concentration.
APIGROUP	Allows different groups of oil PVT tables to be used in different <code>PVTNUM</code> regions when using API Tracking.
BGGI	Variation of saturated gas FVF with pressure and $G_i$ .
BOGI	Variation of saturated oil FVF with pressure and $G_i$ .
DEPTHTAB	Depth data for boundary conditions.
DIFFFC	Molecular diffusion coefficients for each PVT region.
DIFFFDP	Restricts the molecular diffusion calculation to just the matrix-fracture flows in dual porosity runs.
DPKRMOD	Modifies the oil relative permeability in dual porosity runs.
EHYSTR	Curvature parameters and model selection for hysteresis.
EHYSTRR	Hysteresis parameters by region.
ENSPCVDV	Variation of the scaled connate saturation, for $P_c$ curves only, with depth.
FILLEPS	Controls the output of end point data in the <code>INIT</code> file.
FOAMADS	Foam adsorption functions.
FOAMFCN	Gas mobility reduction dependence upon capillary number (functional model).
FOAMDCYO	Foam decay as a function of oil saturation (tabular model).
FOAMDCYW	Foam decay as a function of water saturation (tabular model).
FOAMMOB	Gas mobility reduction data (tabular model).
FOAMMOBP	Pressure dependence on foam mobility reduction (tabular model).
FOAMMOBS	Shear dependence on foam mobility reduction (tabular model).
FOAMOPTS	Specifies the Foam Model options.
FOAMROCK	Specifies the foam rock properties.
GIALL	Variation of saturated properties with pressure and $G_i$ .
GINODE	$G_i$ nodal values.
HMMROCK	Cumulative multipliers on rock compressibility.
HMMROCKT	Cumulative modifiers of rock compaction parameters.
HMPROPS	Section heading for end-point modifiers.

<b>ECLIPSE 100 only keyword</b>	<b>Description</b>
HMROCK	Calculate gradients with respect to rock compressibility multipliers.
HMROCKT	Calculate gradients with respect to rock compaction parameters.
HMRREF	Reference pressures for HMROCKT.
HYDRHEAD	Reference quantities for hydraulic head calculation.
HYDRO	Defines hydrocarbon type.
HYMOBGDR	Change the way secondary drainage curves are computed in hysteresis runs with soluble gas.
HYSTCHCK	Activates additional consistency checks between drainage and imbibition curve endpoints when using the Hysteresis option.
IMKRVD	Imbibition end point relative permeabilities versus depth.
IMPCVD	Variation of maximum capillary pressure values with depth for the imbibition curve.
IMPTVD	Imbibition end point saturation values versus depth.
IMSPCVD	Variation of the scaled connate saturation, for $P_c$ curves only, with depth for the imbibition curve.
INTPC	Invokes the dual porosity integrated $P_c$ option.
IONXROCK	Rock ion exchange parameters for cation exchange.
IONXSURF	Surfactant micelle ion exchange parameters for cation exchange.
IPCG, IPCW	Imbibition curve end point capillary pressure values, equivalent to PCG, PCW. Imbibition curve end point scaling data, equivalent to SGL etc.
ISGLPC, ISWLPC	Imbibition curve end point scaling data, equivalent to SGLPC and SWLPC.
LSALTFNC	Specifies tables of salt concentration versus weighting factor for high-salinity/low-salinity saturation functions.
MASSFLOW	Massflow data for boundary conditions.
NOWARNEP	Suppresses all warning messages resulting from end point consistency checks
PECOEFS	Specifies petro-elastic fluid and rock properties, and other Petro-elastic Model options.
PEKTABx	Pressure dependent petro-elastic frame bulk modulus functions.
PEGTABx	Pressure dependent petro-elastic frame shear modulus functions.
PLMIXPAR	Todd-Longstaff mixing data for the Polymer model.
PLYESAL	Coefficients for polymer effective salinity.
REACT	Reaction activation molar energy
ROCKWNOD	Saturation nodes for 2-D water induced rock compaction tables.
RSCONST	Sets a constant $R_s$ for dead oil over the whole field.
RSCONSTT	Sets constant $R_s$ for dead oil, a different value in each PVT region.
RSGI	Variation of saturated GOR with pressure and $G_i$ .
RVGI	Variation of saturated OGR with pressure and $G_i$ .

<b>ECLIPSE 100 only keyword</b>	<b>Description</b>
RVCONST	Sets a constant $R_v$ for dry gas over the whole field.
RVCONSTT	Sets constant $R_v$ for dry gas, a different value in each PVT region.
SALTNODE	Salt concentration nodes for polymer solution viscosity.
SCALELIM	Sets saturation scaling limits for each end point scaling versus depth region.
SDENSITY	Input of the solvent surface density.
SGCWMIS	Miscible critical gas saturation tables.
SGLPC	Connate gas saturation, for Pc curves only, for cells in the current input box.
SGWFN	Gas-water relative permeability and capillary pressure, as functions of $S_g$ .
SOCRS	Scaled critical oil-in-water saturations at miscible conditions.
SORWMIS	Input of the residual oil saturation for miscible displacement.
SPECHEAT	Tables of fluid specific heat data.
SPECROCK	Tables of rock specific heat data.
STOG	Tables of oil/gas surface tension.
STOW	Tables of oil/gas surface tension.
SURFADDW	Tables of water/gas surface tension.
SURFESAL	Tables of adsorbed surfactant mass vs. fraction of oil-wet and water-wet curves, for calculating immiscible relative permeabilities.
STWG	Coefficients for surfactant effective salinity.
SWLPC	Connate water saturation, for Pc curves only, for cells in the current input box.
TEMPTVD	Requests flux limited transport for the Temperature option.
TLMIXPAR	Input of the Todd-Longstaff mixing parameter.
TPAMEPSS	Volumetric strain values for solvent in ROCKPAMA.
TRACERKM	Partitioning functions $K(P)$ for generalized, multi-partitioned tracers.
TRACERKP	Partitioning function $K(P)$ for partitioned tracers.
TRACTVD	Requests explicit flux limiting scheme to reduce tracer numerical dispersion.
TRACITVD	Requests implicit flux limiting scheme, to reduce numerical dispersion.
VEFRAC	Fraction of VE relative permeabilities used in place of rock curves, for the entire grid.
VEFRACP	Fraction of VE pseudo capillary pressures used in place of rock curves, for the entire grid.
VEFRACPV	Fraction of VE pseudo capillary pressures used in place of rock curves, for the current box.
VEFRACV	Fraction of VE relative permeabilities used in place of rock curves, for the current box.
WAGHYSTR	Activates the WAG Hysteresis option.

## ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
	ACF	Acentric factors.
	ACFDET	Acentric factor for detailed components
	ACFS	Acentric factors for surface production system
	ALPHA	Transport coefficient tables.
	ALPHAD	Transport coefficient tables for drainage processes.
Thermal only	AMFVD	Aqueous mole fractions versus depth.
	BIC	Binary interaction coefficients.
	BICAQ1BICAQ2BICAQ3	Binary interaction coefficients for aqueous phase.
	BICS	Binary interaction coefficients for surface EoS.
Geomechanics only	BIOTCTAB	Defines a multiplier for the Biot constant, this being a two-dimensional tabular function of maximum and minimum principal stress.
	BOUNDARY	Set area of grid to be printed.
	CALVAL	Component calorific values.
	CALVALR	Component calorific values for multiple equation of state regions.
	CATYPE	Component type for asphaltene modeling.
Thermal only	CCTYPE	Component liquid compressibility type.
Thermal only	CCTYPES	Component liquid compressibility type at surface conditions.
	CNAMES	Component names.
Thermal only	CGDTYPE	Component gas (EoS) density model.
Thermal only	CGVTYPE	Component gas (EoS LBC) viscosity model.
Thermal only	CODTYPE	Component oil (EoS) density model.
Thermal only	COVTYPE	Component oil (EoS LBC) viscosity model.
	COMPVD	Total composition with respect to depth
Thermal only	CREF	Component compressibilities.
	CREFW	Compressibility of water components at the reference pressure.
	CREFWS	Compressibility of water components for surface calculations
	CVTYPE	Component volatility (dead, live, gaseous).
	CVTYPES	Component volatility (dead, live, gaseous) at surface conditions.
	CWTYPE	Water component type.
	DETAILMF	Initial make-up of a lumped component in terms of its detailed components versus depth.

Restriction	ECLIPSE 300 only keyword	Description
	DETAILVD	Initial make-up of a lumped component in terms of its detailed components versus depth.
	DIFFAGAS	Activity corrected gas diffusion coefficients.
	DIFFAOIL	Activity corrected oil diffusion coefficients.
	DIFFCGAS	Gas diffusion coefficients.
	DIFFCGO	Gas-Oil cross phase diffusion coefficients.
CO2STORE or GASWAT	DIFFCGW	Gas-Water cross phase diffusion coefficients
	DIFFCOIL	Oil diffusion coefficients.
	DIFFCOG	Oil-Gas cross phase diffusion coefficients.
GASWAT, CO2STORE or GASSOL	DIFFCWAT	Water diffusion coefficients.
GASWAT, CO2STORE or GASSOL	DIFFCWG	Water-Gas cross phase diffusion coefficients
	DIFFTGAS	Gas thermal diffusion coefficients.
	DIFFTOIL	Oil thermal diffusion coefficients.
	DNGL	Partial densities for NGLs.
	DRAINAGE	Forces the Killough relative permeability scanning curve to be below the drainage curve.
	DREF	Reference densities.
	DREFS	Reference densities for surface EoS.
Thermal only	DREFT	Alias for DREF.
Thermal only	DREFTS	Alias for DREFS.
	DREFW	Density of water components at the reference pressure.
	DREFWS	Density of water components for surface calculations.
Thermal only	ENKRVT	Endpoint relative permeabilities versus temperature.
Thermal only	ENPCVT	Endpoint capillary pressures versus temperature.
Thermal only	ENPTVT	Endpoint saturations versus temperature.
	EOS	Specify which equation of state is to be used.
	EOSS	Specify surface equation of state.
ODD3P option.	EPSODD3P	End-point scaling, reference surface tensions and threshold interfacial tensions.
Thermal only	EQLDKVCR	K-values for equilibrium deviation reactions
Thermal only	EQLDREAC	Equilibrium deviation reaction rates

Restriction	ECLIPSE 300 only keyword	Description
	ESPNODE	Effective salinity nodes for salt sensitive polymer flood studies.
	ESSNODE	Effective salinity nodes for salt sensitive polymer flood studies
	FACTLI	Modify Li critical temperature correlation.
	FVST	Specifies miscibility variation with surface tension.
Thermal only	GASVISCF	Gas viscosity function of temperature.
Thermal only	GASVISCT	Gas viscosity table as a function of temperature.
Geomechanics only	GEOYLDF	Defines parameters for the plastic yield function in each geomechanics region.
	GRES	Reference gravities for EoS.
	GRES	Reference gravities for surface EoS.
	GSF	Gas saturation functions.
Thermal only	HEATVAP	Component heat of vaporization.
Thermal only	HEATVAPE	Heat of vaporization exponent.
Thermal only	HEATVAPS	Component heat of vaporization at standard temperature.
ODD3P option.	HSGRO	Hysteresis end-point (residual) grid-block gas saturations for a gas-oil system
ODD3P option.	HSGRW	Hysteresis end-point (residual) grid-block gas saturations for a gas-water system
ODD3P option.	HSORG	Hysteresis end-point (residual) grid-block oil saturations for an oil-gas system
ODD3P option.	HSORW	Hysteresis end-point (residual) grid-block oil saturations for an oil-water system
ODD3P option.	HSWRG	Hysteresis end-point (residual) grid-block water saturations for a water-gas system
ODD3P option.	HSWRO	Hysteresis end-point (residual) grid-block water saturations for a water-oil system
	HYDRO	Defines hydrocarbon type.
	IKU3P	Requests IKU three-phase relative permeabilities.
	KRGDI	Gas relative permeability damage initiation.
	KRGDM	Gas relative permeability damage multipliers.
	KRODI	Oil relative permeability damage initiation.
	KRODM	Oil relative permeability damage multipliers.
	KRWDI	Water relative permeability damage initiation.
	KRWDM	Water relative permeability damage multipliers.
	KVCOMP	Interpolation component for composition dependent K-values.

Restriction	ECLIPSE 300 only keyword	Description
	KVCR	K-value correlation parameters.
Thermal only	KVCRS	K-value correlation parameters at surface conditions.
Thermal only	KVCRWAT	Water K-value correlation parameters.
Thermal only	KVGEN	Specify that K-values are to be generated internally.
	KVMF	Interpolation mole fractions for composition dependent K-values.
	KVTAB <sub>Cn</sub>	Tables of K-values.
	KVTABLE	Tables of K-values.
	KVTAB <sub>Tn</sub>	Tables of K-values.
Thermal only	KVTEMP	Temperatures for K-value tables.
Thermal only	KVWI	Wilson K-value correlation parameters.
Thermal only	KWTAB <sub>Tn</sub>	Tables of aqua Kw-values.
Thermal only	KWTEMP	Temperatures for Kw-value tables.
Thermal only	LBCCOEF	Set non-default LBC coefficients.
	LBCCOEFR	Set non-default LBC coefficients for multiple equation of state regions.
	LGRCOPY	Copy global values to local cells before operations on cells within an LGR.
	LILIM	Specifies the range for Li-interpolation.
	LUMPING	Set up names of tracers describing detailed composition of a lumped hydrocarbon component.
	MISCEXP	Miscibility exponent.
	MISCSTR	Miscibility surface tension reference.
	MISCSTRP	Miscibility surface tension reference pressure.
	MISCSTRR	Allows the reference miscible surface tension to be set by saturation region number.
	MSFN	Relative permeability curves for miscible displacement.
	MW	Molecular weights.
	MWDETAIL	Molecular weights for detailed components.
	MWS	Molecular weights for surface EoS.
	MWW	Molecular weight of water components.
	MWWS	Molecular weight of water components for surface calculations.
	NCOMPS	Confirm number of components.
	ODD3P	ODD3P three-phase relative permeability and capillary pressure hysteresis option.

Restriction	ECLIPSE 300 only keyword	Description
ODD3P option	OPTODD3P	Provides backwards compatibility and saturation switching options
	OILCOMPR	Oil compressibility, expansion coefficient.
Thermal only	OILMW	Oil molecular weight.
Thermal only	OILSPECH	Oil specific heat.
Thermal only	OILVINDX	Oil viscosity index.
Thermal only	OILVISCC	Oil viscosity-temperature correlation
Thermal only	OILVISCF	Oil viscosity function.
Thermal only	OMEGAA	Overrides default $\Omega_a$ values.
	OMEGAADE	Overrides default $\Omega_a$ values for detailed composition.
	OMEGAAS	Overrides default surface $\Omega_a$ values.
	OMEGAASD	Overrides default surface $\Omega_a$ values for detailed composition.
	OMEGAB	Overrides default $\Omega_b$ values.
	OMEGABDE	Overrides default $\Omega_b$ values for detailed composition.
	OMEGABS	Overrides default surface $\Omega_b$ values.
	OMEGABSD	Overrides default surface $\Omega_b$ values for detailed composition.
	PARACHOR	Component parachors.
	PCG	Scales the maximum gas capillary pressure, for cells in the current input box.
	PCRIT	Critical pressures.
	PCRITDET	Critical pressures for detailed components.
	PCRITSDE	Critical pressures for surface EoS for detailed components.
ODD3P option	PCODD3P	Capillary pressure calculation control.
ODD3P option.	PCODD3PG	Capillary pressure gas saturation function
ODD3P option.	PCODD3PW	Capillary pressure water saturation function
	PERMSTAB	Defines a permeability multiplier, this being a two-dimensional tabular function of maximum and minimum principal stress.
	PLYOPTS	Options for the polymer model.
	PREFS	Component reference pressures for surface EoS.
Thermal only	PREFW	Reference pressure for water component density and viscosity calculations.
	PREFWS	Reference pressure for water component density calculations at surface conditions.

Restriction	ECLIPSE 300 only keyword	Description
ODD3P option.	PSGRO	Primary end-point (residual) grid-block gas saturations for the gas-oil system
ODD3P option.	PSGRW	Primary end-point (residual) grid-block gas saturations for the gas-water system
ODD3P option	PSORG	Primary end-point (residual) grid-block oil saturations for the oil-gas system
ODD3P option	PSORW	Primary end-point (residual) grid-block oil saturations for the oil-water system for the option.
ODD3P option	PSWRG	Primary end-point (residual) grid-block water saturations for the water-gas system
ODD3P option.	PSWRO	Primary end-point (residual) grid-block water saturations for the water-oil system
	REACLIMS	Pressure and temperature limits for reaction rate calculations.
	REACPORD	Reaction rate porosity order.
	REACSMAX	Maximum solid saturation in chemical reaction rates.
	REACSORD	Order of solid term in chemical reaction rates.
	RKPVMNMX	Allows pore volume multiplier limits to be set for the BOBERG model.
	ROCKTRMX	Define transmissibility variation values in the X-direction in order to define transmissibility as an exponential function of porosity.
	ROCKTRMY	Define transmissibility variation values in the Y-direction in order to define transmissibility as an exponential function of porosity.
	ROCKTRMZ	Define transmissibility variation values in the Z-direction in order to define transmissibility as an exponential function of porosity.
	RSWVD	Aqueous concentration of CO <sub>2</sub> versus depth.
	SALTEFF	Coefficients to calculate effective salinity
Polymer and Surfactant models.	SALINITY	Reservoir salinity.
	SALINITR	Regional reservoir salinity.
Geomechanics only	SCDATABG	Defines a scale damage table to be used together with the proppant flowback feature.
Thermal only	SCREF	Component solid phase compressibility.
Thermal	SDREF	Component solid phase reference density.
Thermal	SGF3	Three-phase gas saturation functions.
	SOGCR	Residual oil-in-gas saturation for cells in the current box.

Restriction	ECLIPSE 300 only keyword	Description
	SOLIDADS	Solid adsorption table.
	SOLIDMMC	Solid mobility multiplier (as a function of adsorbed concentration).
	SOLIDMMS	Solid mobility multiplier (as a function of adsorbed saturation).
	SOLUBILI	Specify properties of Water-CO <sub>2</sub> systems.
	SOLUBILS	Specify properties of Water-CO <sub>2</sub> systems with salt dependencies.
	SOLUCO2H	Specify properties of Water-CO <sub>2</sub> systems with H <sub>2</sub> S dependencies.
	SOLUH2SC	Specify properties of Water-H <sub>2</sub> S systems with CO <sub>2</sub> dependencies.
	SOLWTAB	Defines an interpolation parameter as a function of adsorbed solid concentration.
	SOLWW	Defines a component weighting factor for calculating the wetting-solid concentration.
	SOR	Residual oil saturation.
	SOROPTS	Residual oil model options.
	SPECHA	Component oil specific heat, 1 <sup>st</sup> coefficient.
	SPECHB	Component oil specific heat, 2 <sup>nd</sup> coefficient.
	SPECHG	Component gas specific heat, 1 <sup>st</sup> coefficient.
	SPECHH	Component gas specific heat, 2 <sup>nd</sup> coefficient.
	SPECHS	Component solid specific heat, 1 <sup>st</sup> coefficient.
	SPECHT	Component solid specific heat, 2 <sup>nd</sup> coefficient.
	SPECHW1	Component aqua specific heat, 1 <sup>st</sup> coefficient.
	SPECHW2	Component aqua specific heat, 2 <sup>nd</sup> coefficient.
	SPREF	Component solid phase reference pressure.
	SSHIFT	Equation of state shift parameters.
	SSHIFTS	Equation of state shift parameters for surface EoS.
	STCOND	Specify standard conditions.
	STONEPAR	Modify Stone's formula 1 three phase oil relative permeability
	STOPROD	Stoichiometric coefficients of products.
	STOREAC	Stoichiometric coefficients of reactants.
	STREF	Component solid phase reference temperature.
	STHERMX1	Component solid phase thermal expansion coefficient.
	STVP	Surface tension with respect to pressure.

Restriction	ECLIPSE 300 only keyword	Description
	SURFOPTS	Surfactant options
	SWF3	Three phase water saturation functions.
	TBOIL	Component boiling points.
	TBOILS	Component boiling points for surface EoS.
	TCRIT	Critical temperatures.
	TCRITDET	Critical temperatures for detailed components.
	TCRITS	Critical temperatures for surface EoS.
	TCRITSDE	Critical temperatures for surface EoS for detailed components
	TEMPVD	Temperature vs. depth data.
	THANALB	Analytic water and steam densities.
	THANALH	Analytic water and steam enthalpy values.
	THANALV	Analytic oil viscosities.
	THERMEX1	Component thermal expansion coefficient.
	THSVC	Steam viscosity coefficients.
	THTABB	Tabular water and steam densities.
	THWVC	Water viscosity coefficients.
	TRANGE	Expected temperature limits.
	TREF	Reference temperatures.
	TREFS	Reference temperatures for surface EoS.
Thermal	TREFT	Alias for TREF
Thermal	TREFTS	Alias for TREFS.
	VCOMPACT	Compact volumes.
	VCRIT	Critical volumes.
	VCRITDET	Critical volumes for detailed components.
	VCRITS	Critical volumes for surface EoS.
	VCRITSDE	Critical volumes for surface EoS for detailed components.
	VCRITVIS	Critical volumes for viscosity calculations.
	VDFLOW	Specify velocity dependent flow coefficient.
	VDKRO	Oil velocity dependent relative permeability data.
	VDKRG	Gas velocity dependent relative permeability data.
	VDKRGC	Gas velocity dependent relative permeability data.
	VREFW	Viscosity and viscosibility for water components at the reference pressure.
Thermal	WATDENT	Water density temperature dependence.
	WATERTAB	Water pressure tables.

Restriction	ECLIPSE 300 only keyword	Description
	WNAMEs	Water component names.
GASWAT case	WSF	Water saturation functions
	XMFVP	Liquid composition with respect to pressure tables.
	YMFVP	Vapor composition with respect to pressure tables.
	ZCRIT	Critical Z-factors.
	ZCRITDET	Critical Z-factors for detailed components.
	ZCRITS	Critical Z-factors for surface EoS
	ZCRITSDE	Critical Z-factors for surface EoS for detailed components.
	ZCRITVIS	Critical Z-factors for viscosity calculations.
	ZI	Enter overall composition
	ZMFVD	Total composition with respect to depth tables.
Thermal	ZFACTOR	Z-factors.

## 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.

ECLIPSE 300

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.
- [PVTNUM](#) 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

Restriction	Keyword	Description
Keywords should not be used unless DIRECT is selected in keyword <a href="#">SATOPTS</a> .	<a href="#">KRUNUMX</a> , <a href="#">KRUNUMY</a> , <a href="#">KRUNUMZ</a>	Saturation table numbers for the +X, +Y, +Z faces of a grid block.
Keywords should not be used unless IRREVERS is also selected in keyword <a href="#">SATOPTS</a> .	<a href="#">KRUNUMX-</a> , <a href="#">KRUNUMY-</a> , <a href="#">KRUNUMZ-</a>	Saturation table numbers for the -X, -Y, -Z faces of a grid block.

## Dual porosity option (ECLIPSE 100 only)

- [KRUNUMMF](#) 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)

Restriction	Keyword	Description
	<a href="#">IMBNUM</a>	Imbibition saturation table numbers for all faces of every grid block in current box.

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

## WAG hysteresis option

Restriction	Keyword	Description
This keyword is optional if WAGHYSTR has been specified but should not be specified otherwise.	WH2NUM	Two-phase water relative permeability saturation region numbers.
This keyword is optional if WAGHYSTR has been specified but should not be specified otherwise.	WH3NUM	Three-phase water relative permeability saturation region numbers.

## Other options

### Common keywords

Keyword	Description
ENDNUM	End point scaling regions.
TNUM	Tracer tracking regions.
MISCNUM	Miscible Flood option regions.
OPERNUM	Defines region on which to do operations using the OPERATER keyword. The data can also be edited using ADDREG, EQUALREG, MULTIREG and COPYREG.
ROCKNUM	Rock compaction regions.
SURFNUM	Surfactant miscible regions.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
TRKPF	Partitioned tracer K(P) partitioning function regions.
LSLTWNUM	Low-salinity (water-wet) saturation function regions.
LWSLTNUM	Low-salinity (oil-wet) saturation function regions.
PENUM	Petro-elastic Model regions.
RESIDNUM	Vertical Equilibrium residual flow option regions.
SURFWNUM	Surfactant adsorption function regions.

**ECLIPSE 300 only keywords**

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

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

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

**Operational**

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

**Common keywords**

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

Keyword	Description
EQUALREG	Set by FLUXNUM, MULTNUM or OPERNUM region.
EQUALS	Sets specified arrays to specified constants within the current box.
IMPORT	Imports binary data generated by a grid pre-processor.
MULTIPLY	Multiplies specified arrays by specified constants within the current box.
MULTIREG	Multiply by FLUXNUM, MULTNUM or OPERNUM region.
OPERATE	Performs arithmetic operations on arrays.
OPERATER	Performs arithmetic operations on arrays for a specific region of the grid.
REFINE	Initiates data for a named local grid.

### ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
BOUNDARY	Set area of grid to be printed
LGRCOPY	Copy global values to local cells before operations on cells within an LGR.

## Alphabetical list of keywords

### Common keywords

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

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

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

## ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
FIP	Additional sets of fluid-in-place region numbers, for every grid block in the current input box.
FIPOWG	Creates a family of FIP regions to represent the gas, oil and water zones at equilibration
IMBNUMMF	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).
KRNUMMF	$K_r$ region for matrix-fracture flows, for every grid block in the current input box (if using the Dual Porosity option).
LSLTWNUM	Low-salinity (oil-wet) saturation function regions.
LWSLTNUM	Low-salinity (water-wet) saturation function regions.
PENUM	Petro-elastic Model regions.
RESIDNUM	Relative permeability table numbers for Vertical Equilibrium residual flow option.
SURFWNUM	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.
TRKPF	Region numbers for selecting which K(P) table to use for each partitioned tracer.

## ECLIPSE 300 only keywords

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

Restriction	ECLIPSE 300 only keyword	Description
	PMANUM	Defines pressure maintenance regions (See the <a href="#">GPMaint</a> keyword).
ODD3P option	PSTNUM	Grid-block primary saturation table numbers
ODD3P option	SDRGO	Gas phase initial grid-block saturation direction indicators for the gas-oil system
ODD3P option	SDRGW	Gas phase initial grid-block saturation direction indicators for the gas-water system
ODD3P option	SDROG	Oil phase initial grid-block saturation direction indicators for the gas-oil system
ODD3P option	SDROW	Oil phase initial grid-block saturation direction indicators for the oil-water system
ODD3P option	SDRWG	Water phase initial grid-block saturation direction indicators for the gas-water system
ODD3P option	SDRWO	Water phase initial grid-block saturation direction indicators for the oil-water system
SOLID option	SOLWNUM	Solid wettability region numbers.
	TRACKREG	Tracer <a href="#">TRACK</a> ing regions.
	THERMNUM	Thermal regions.

## 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 EQLNUM 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

Keyword	Description
PRESSURE	Cell pressures
SWAT	Water saturations
SGAS	Gas saturations
RS	Liquid gas-oil ratios

Keyword	Description
RV	Vapor oil-gas ratios
PBUB	Initial bubble point pressures
PDEW	Initial dew point pressures

## 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

ECLIPSE 300

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:

```
RPTSOL
PRESSURE SOIL SWAT SGAS /
```

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 [AQUDIMS](#)):

Aquifer Type	Description
Carter-Tracy aquifers	<a href="#">AQUCT</a> and <a href="#">AQUANCON</a> / <a href="#">AQANCONL</a> ( <a href="#">AQUTAB</a> in PROPS section)
Fetkovich aquifers	<a href="#">AQUFETP</a> and <a href="#">AQUANCON</a> / <a href="#">AQANCONL</a> , or <a href="#">AQUFET</a> (ECLIPSE 100 only)
Constant flux aquifers	<a href="#">AQUFLUX</a> and <a href="#">AQUANCON</a> / <a href="#">AQANCONL</a>
Constant head gas aquifers	<a href="#">AQUCHGAS</a> and <a href="#">AQUANCON</a> / <a href="#">AQANCONL</a>
Constant head water aquifers	<a href="#">AQUCHWAT</a> and <a href="#">AQUANCON</a> / <a href="#">AQANCONL</a>

Table 2.8: Analytical aquifer keywords

ECLIPSE 300 only

Gas composition for [AQUCHGAS](#) and, for the [GASWAT](#) and [CO2STORE](#) options, water composition for [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) and [AQUFLUX](#): [AQSTREAM](#)

For the multi-component water option ([COMPW](#)), water composition for [AQUCHWAT](#), [AQUCT](#), [AQUFETP](#) and [AQUFLUX](#), [AQSTREAW](#):

Restriction	Keyword	Description
E100 only	<a href="#">AQUALIST</a>	Aquifer lists

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:

	API	Brine	Tracers
Equilibration	<a href="#">APIVD</a>	<a href="#">SALTVD</a>	<a href="#">TVDP</a>
	one table per equilibration region		
Enumeration	<a href="#">OILAPI</a>	<a href="#">SALT</a>	<a href="#">TBLK</a>
	for each grid block		

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

Restriction	Keyword	Description
Optional	<b>RPTSOL</b>	Report arguments for <b>SOLUTION</b> data.
Optional	<b>BOUNDARY</b>	Defines section of grid for which output maps are required.

Restriction	Keyword	Description
Optional	DATUM	Defines datum depth for output of depth corrected pressures.
Optional	DATUMR	Defines datum depth for output of depth corrected pressures for each FIP region.
Optional	DATUMRX	Defines datum depth for output of depth corrected pressures for each FIP region in each FIP set

## Equilibration

### Common keywords

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

### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
Required for equilibration when using API tracking.	APIVD	Variation of oil API with depth

### ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
Optional.	OLDEQUIL	Perform Equilibration using pre-2000A method
Optional, for the CO2SOL and GASSOL options.	.RSWVD	Aqueous concentration of dissolved gases versus depth
Thermal only	AMFVD	Aqueous mole fractions versus depth.

## 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

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

### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
Required for enumeration when using API tracking	<a href="#">OILAPI</a>	Oil API gravity values in every grid block
Either PRESSURE or PRVD is required for enumeration.	<a href="#">PRVD</a>	Pressure versus depth.
	<a href="#">SFOAM</a>	Initial Foam concentrations.
	<a href="#">SPOLY</a>	Initial Polymer concentrations.
	<a href="#">SURF</a>	Initial Surfactant concentrations

Restriction	ECLIPSE 100 only keyword	Description
Optional for systems with both live oil and wet gas.	VAPPARS	Parameters controlling vaporization of oil in under saturated gas

### ECLIPSE 300 only keywords

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

## Aquifer data

### Common keywords

Restriction	Keyword	Description
	AQANTRC	Initialize concentration of water phase tracers in each analytic aquifer.
	AQUANCON	Connects an analytic aquifer, which was defined by AQUCT, AQUFETP, AQUFLUX, AQUCHWAT and AQUCHGAS, to one or more reservoir cells.
	AQUANCONL	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

Restriction	Keyword	Description
Required in compositional run with AQUCHGAS aquifer.  Optional for the GASWAT or CO2STORE options.	AQSTREAM	Sets composition of gas stream for AUCHGAS aquifer.  For the GASWAT and CO2STORE options, sets water composition for AUCHWAT, AUCT, AUFETP and AUFUX.
	AUCHGAS	Constant head gas aquifer data specification.
	AUCHWAT	Constant head aquifer data specification.
	AUCT	Carter-Tracy aquifer property data specification.
For use with AQUANCON / AQUANCONL	AUFETP	Fetkovich aquifer property data specification
	AUFUX	Constant flux aquifer data specification.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
AQANNC	Specifies analytic aquifer connections explicitly.
AUFET	Complete Fetkovich aquifer specifications.
AUFUX	Constant flux aquifer data specification.
AQUALIST	Aquifer lists.
RAINFALL	Specifies monthly flux rates for constant flux aquifers.

### ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
AQSTREAMW	For the multi-component water option (COMPW), sets water composition for AUCHWAT, AUCT, AUFETP and AUFUX.

### 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

Option	Keyword	Description
Coal Bed Methane	GASCONC	Initial coal gas concentration.

Option	Keyword	Description
Coal Bed Methane	<a href="#">GASSATC</a>	Initial saturated coal gas concentration.

### ECLIPSE 100 only keywords

Option	Keyword	Description
Coal Bed Methane	<a href="#">GCVD</a>	Initial coal gas concentration with depth.
Gi (enumeration only)	<a href="#">GI</a>	Initial Gi values.
Coal Bed Methane	<a href="#">SCVD</a>	Initial coal solvent concentration with depth.
Solvent	<a href="#">SSOL</a>	Initial solvent saturation.

### ECLIPSE 300 only keywords

Option	Keyword	Description
Coal Bed Methane	<a href="#">GASADCO</a>	Initial coal gas concentration for each component.
Coal Bed Methane	<a href="#">GASADEC</a>	Initial saturated/equilibrium coal gas concentration for each component.
Coal Bed Methane	<a href="#">GASCCMP</a>	Specify the component number for GASCONC and GASSATC.
<a href="#">THERMAL</a>	<a href="#">TEMPI</a>	Initial temperatures.
Coal Bed Methane	<a href="#">SORBFRAC</a>	Scale initial adsorbed gas in the coal.
Coal Bed Methane	<a href="#">SORBPRES</a>	Specify the initial sorption pressure for the adsorbed gas in the coal.

## Operational keywords

### Common keywords

Restriction	Keyword	Description
Optional.	<a href="#">ADD</a>	Add value to an array.
Optional.	<a href="#">ADDREG</a>	Add by <a href="#">FLUXNUM</a> , <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
Optional.	<a href="#">BOX</a>	Redefines the current input box.
Optional.	<a href="#">COPY</a>	Copies data in the current box from one specified region to another.
Optional.	<a href="#">COPYREG</a>	Copy by <a href="#">FLUXNUM</a> , <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
Optional.	<a href="#">ENDBOX</a>	Redefines the current input box to encompass the whole grid.
Optional.	<a href="#">ENDFIN</a>	End input for local grid refinement.
Optional.	<a href="#">EQUALREG</a>	Set by <a href="#">FLUXNUM</a> , <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
Optional.	<a href="#">EQUALS</a>	Set array to constant.
Optional.	<a href="#">IMPORT</a>	Imports binary data generated by a grid pre-processor.
Optional.	<a href="#">MULTIPLY</a>	Multiply array to constant.
Optional.	<a href="#">MULTIREG</a>	Multiply array by <a href="#">FLUXNUM</a> , <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
Optional.	<a href="#">OPERATE</a>	Performs arithmetic operations on arrays.

Restriction	Keyword	Description
Optional.	<a href="#">OPERATER</a>	Performs arithmetic operations on arrays for a specific region of the grid (defined using the <a href="#">OPERNUM</a> keyword in the <a href="#">GRID</a> or <a href="#">REGIONS</a> section).
Optional.	<a href="#">REFINE</a>	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

Restriction	Keyword	Description
	<a href="#">ADD</a>	Add value to an array.
	<a href="#">ADDREG</a>	Add by <a href="#">FLUXNUM</a> , <a href="#">MULTNUM</a> or <a href="#">OPERNUM</a> region.
	<a href="#">AQANCONL</a>	Connects an analytic aquifer which was defined by <a href="#">AQUCT</a> , <a href="#">AQUFETP</a> , <a href="#">AQUFLUX</a> , <a href="#">AQUCHWAT</a> and <a href="#">AQUCHGAS</a> , to one or more reservoir cells located within local grid refinements.
	<a href="#">AQANTRC</a>	Initializes concentration of water phase tracers in each analytic aquifer.
	<a href="#">AQUANCON</a>	Connects an analytic aquifer which was defined by <a href="#">AQUCT</a> , <a href="#">AQUFETP</a> , <a href="#">AQUFLUX</a> , <a href="#">AQUCHWAT</a> and <a href="#">AQUCHGAS</a> , to one or more reservoir cells.

Restriction	Keyword	Description
	AQUCHGAS	Constant head gas aquifer data specification.
	AQUCHWAT	Constant head water aquifer data specification.
	AQUCT	Carter-Tracy aquifer property data specification.
for use with AQUANCON/ AQANCONL	AQUFETP	Fetkovich aquifer property data specification
	AQUFLUX	Constant flux aquifer data specification.
	BOUNDARY	Defines section of grid for which output maps are required.
	BOX	Redefines the current box.
	COPY	Copies data in the current box from one specified region to another.
	COPYREG	Copy by FLUXNUM, MULTNUM or OPERNUM region.
	DATUM	Defines datum depth for output of depth corrected pressures.
	DATUMR	Defines datum depth for each FIP region.
	DATUMRX	Defines datum depth for each FIP region in each FIP set.
	DYNAMICR	Defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
	ENDBOX	Redefines the current box to encompass the whole grid.
	ENDDYN	Indicates the end of a DYNAMICR keyword sequence.
	ENDFIN	End input for local grid refinement.
	EQUALREG	Set by FLUXNUM, MULTNUM or OPERNUM region.
	EQUALS	Set array to constant.
	EQUIL	Fluid contact depths and other equilibration parameters.
	FILEUNIT	Confirms the units of the data set.
	GASCONC	Initial coal gas concentration.
	GASSATC	Initial coal gas saturated concentration.
	IMPORT	Imports binary data generated by a grid pre-processor.
	MULTIPLY	Multiply array by constant.
	MULTIREG	Multiply array by FLUXNUM, MULTNUM or OPERNUM region.
	OPERATE	Performs arithmetic operations on arrays.
	OPERATER	Performs arithmetic operations on arrays for a specific region of the grid (defined using the OPERNUM keyword in the GRID or REGIONS section).
	PBUB	Bubble point pressures in every grid block.
	PBVD	Variation of bubble point pressure with depth.
	PDEW	Dew point pressures in every grid block.
	PDVD	Variation of dew point pressure with depth.
	PRESSURE	Pressures in every grid-block.
	REFINE	Initiates data for a named local grid.

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

Restriction	ECLIPSE 300 only keyword	Description
	AMF	Specifies cell initial aqueous composition explicitly.
	AQSTREAM	Sets gas composition for use with AQUCHGAS aquifer and, for the GASWAT and CO2STORE options, water composition for use with AQUCHWAT, AQUCT, AQUFETP and AQUFLUX.
	AQSTREAMW	For the multi-component water option (COMPW), sets water composition for use with AQUCHWAT, AQUCT, AQUFETP and AQUFLUX.
	DBGODD3P	Debug output for ODD3P option.
	FCCRIT	Flash convergence criteria.
	FIELDSEP	Introduces a field separator
	FIPSEP	Introduces a separator to be used in a fluid-in-place region.
	GASADCO	Initial coal gas concentration.
	GASADCO	Initial coal gas equilibrium/saturated concentration.
	GETSOL	Obtain initial solution arrays from a Restart file.
	GPTABLE	Tables of gas plant recovery factors.
	GPTABLEN	Gas plant table with NGL recoveries.
	GPTABLE3	Gas plant table with oil, NGL and gas recoveries.
	NEI	Specify compositions for non-equilibrium initialization
	OLDEQUIL	Perform Equilibration using pre-2000A method.
	OUTSOL	Controls solution output for GRAF and flexible restarts
	RECOVERY	Recovery plant table.
SOR option.	ROMF	Cell residual oil composition.
	RSW	Aqueous concentration of CO <sub>2</sub> in every grid block.
	RSWVD	Aqueous concentration of CO <sub>2</sub> versus depth.
	SMF	Cell initial solid composition.
	SOIL	Oil saturation in every grid-block.
SOR option.	SOILR	Cell residual oil saturation.
	SSOLID	Solid saturation in every grid-block.
	TEMPI	Initial temperatures.
water component model	WI	Initial water composition
water component model	WMF	Cell initial water composition

Restriction	ECLIPSE 300 only keyword	Description
water component model	WMFVD	Initial water composition as a function of depth
	XMF	Cell initial oil composition.
	YMF	Cell initial gas composition
	ZMF	Cell initial total composition.

## 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:

```
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"](#).

## 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 /
```

```

9 1 14 /
12 3 6 /
/

```

Keywords that begin with a B and ends 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 /
/

```

ECLIPSE 300

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 /

```

```
'PROD3' /
/
```

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.

```
COFR
-- Well Name Cell
-- '*' /
/
```

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.

ECLIPSE 300

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
```

ECLIPSE 300

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

## Format of SUMMARY keywords beginning with F

ECLIPSE 300

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.

## Format of SUMMARY keywords beginning with G

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'
/
```

specifies that the water cut for the three named groups of wells are to be stored on the summary files.

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

ECLIPSE 300

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.

```
LBOSAT
-- LGR Name      Local Cell
  'LGR1'         1  1  1 /
  'LGR2'         1  2  3 /
/
```

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.

```
LBPR_X
-- LGR Name Local I J K Cell X Y Z Coordinate Ref-Height GroupName
  'LGR1'     1  1  1      122.1 -63.4 -1.4 100.0      'ALP' /
  'LGR2'     1  2  3      122.1 -68.4 -3.4 1*          'ALP' /
/
```

## 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 (/).

```
LCOPT
-- LGR Name Well Name Local Cell
  'LGR1'     'WELL1'   1  1  1 /
  'LGR2'     'PROD'    2  2  2 /
  'LGR3'     'PROD2'   2  2  2 /
/
```

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).

```
LCOFR
-- LGR Name Well Name Local Cell
  'LGR1'     'WELL1'   1  1  1 /
  'LGR2'     'PROD'    2  2  2 /
```

```
'LGR3'      'PROD2'      /
/
```

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.

```
LCOFR
-- LGR Name  Well Name  Local Cell
' LGR1 '    '*'      /
' LGR2 '    '*'      2 2 2 /
/
```

**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 '    'WELL1' /
' LGR2 '    'PROD'  /
' LGR3 '    /
/
```

ECLIPSE 100

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 300

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 /
 /
```

ECLIPSE 300

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.

ECLIPSE 300

Component flow keywords that start with RKFR and RKFT are available for the standard FIPNUM set only and cannot be compounded with FIP 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:

```

ROIPLFST   for the oil-in-place in the liquid phase for region 2
 2 /       of the set of regions named FST
ROIP_SND   for the total oil-in-place in all the regions in the
 /         set named SND
RPR__THR   for the pressure in all the regions in the set named
 /         THR

```

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.

ECLIPSE 300

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:

```

RCOM       Molar amount of component 2 dissolved in oil, within
1 2 /     region 1 of standard FIP set.
 /
RCGMIFST   Molar amount of all components trapped in gas, within
2 /       region 2 of the set of region named FST
 /

```

## 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:

```

SOFR
'WELL1' 2 /
'WELL1' 3 /
'WELL2' /
 /

```

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.

ECLIPSE 100

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.

ECLIPSE 300

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

ECLIPSE 300

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 STEPTYPE, [SEPARATE](#) and [SUMTHIN](#) 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, WGRH, 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

ECLIPSE 300

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 7<sup>th</sup>, 8<sup>th</sup> and 9<sup>th</sup> 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
```

```

FREAT
/  -- Reaction totals summed over all cells for all reactions

```

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, W<sub>XMF\_7</sub>). 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 WCGMR2\_3, the molar rate of component 15 has to be written as WCGMR215).

#### Water components

For the multi-component water option, the water component index starts at  $N_c+1$  and ends at  $N_c+N_w$  where  $N_c$  is the number of components given by [COMPS](#) and  $N_w$  is the number of water components given by the [COMPW](#) keyword. The water component indices can be used for the summary mnemonics described in ["SUMMARY section keywords"](#) in the *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 *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 NSUMMX set in RUNSPEC keyword [SMRYDIMS](#).

---

**Note:** For example, WOPR applied to four individual wells counts as four separate quantities towards this total.

---

It is quite simple to construct a context-independent SUMMARY section that can be [INCLUDE](#)d in any run, and which provides sufficient data for many requirements.

## Example SUMMARY section

Here is an example of a context independent SUMMARY section:

```

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

```

```

/
-- Field gas inj rate. Cumulative gas inj for field and for every
-- well.
FGIR
FGIT
WGIT
/
-- Instantaneous GOR for field and for every well.
FGOR
WGOR
/
-- Instantaneous water cuts for field and for every well.
FWCT
WWCT
/
-- Oil in place for field and for every FIP region.
FOIP
ROIIP
/
-- Water in place for field and for every FIP region.
FWIP
RWIP
/
-- Gas in place for field and for every FIP region.
FGIP
RGIP
/
-- Average pressure for field and for every FIP region.
FPR
RPR
/
-- Aquifer influx rate for field and every liquid analytic aquifer.
FAQR
AAQR
/

```

## 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 file-name 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:

- See ["Format of SUMMARY keywords with a component index"](#).

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x				COFR	COFRL	Oil Flow Rate (+ve=prod -ve=inj)
x	x					WOFRL	Oil Flow Rate (as above)
x	x					COPRL	Oil Flow Rate (as above)

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

E100	E300	Field	Group	Well	Connection	Completion	Information
x		FOIRH	GOIRH	WOIRH			Oil Injection Rate History (WCONINJH)
x	x	FOIRT	GOIRT	WOIRT			Oil Injection Rate Target/Limit
x	x		GOIRL				Oil Injection Rate Target/Limit (as above)
x	x	FOIT	GOIT	WOIT			Oil Injection Total
x					COIT	COITL	Oil Injection Total (as above)
x						WOITL	Oil Injection Total (as above)
x		FOITH	GOITH	WOITH			Oil Injection Total History (WCONINJH)
x	x	FOPP	GOPP	WOPP			Oil Potential Production rate
x					COPP		Oil Potential Production rate (as above)
x	x	FOPP2	GOPP2	WOPP2			Oil Potential Production rate (including values calculated on shut/stopped wells)
x		FOPI	GOPI	WOPI	COPI		Oil Potential Injection rate
x		FOPI2	GOPI2	WOPI2			Oil Potential Injection rate (including values calculated on shut/stopped wells)
x	x		GOPGR	WOPGR			Oil Production Guide Rate
x			GOIGR	WOIGR			Oil Injection Guide Rate
	x	FOVPR	GOVPR	WOVPR			Oil Voidage Production Rate
	x	FOVPT	GOVPT	WOVPT			Oil Voidage Production Total
	x	FOVIR	GOVIR	WOVIR			Oil Voidage Injection Rate
	x	FOVIT	GOVIT	WOVIT			Oil Voidage Injection Total
	x	FOnPR	GOnPR	WOnPR			nth 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
	x	FOnPT	GOnPT	WOnPT			nth separator stage oil total (n = 1 to 99)
	x	FEOR	GEOR				Export Oil Rate
	x	FEOT	GEOT				Export Oil Total
	x	FEOMF	GEOMF				Export Oil Mole Fraction. See Note 1.

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:

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

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x				CWFR	CWFRL	Water Flow Rate (+ve=prod - ve=inj)
x	x					WWFRL	Water Flow Rate (as above)
x	x					CWPRL	Water Flow Rate (as above)
x	x					WWPRL	Water Flow Rate (as above)
x					CWFRU		Sum of connection water flow rates upstream of, and including, this connection
x	x	FWPR	GWPR	WWPR	CWPR		Water Production Rate
	x	FWMR	GWMR	WWMR			Water Mass Rate
	x	FWMT	GWMT	WWMT			Water Mass Total
x	x	FWPRH	GWPRH	WWPRH			Water Production Rate History ( <a href="#">WCONHIST</a> )
x	x	FWPRT	GWprt	WWPRT			Water Production Rate Target/Limit
x	x		GWpRL				Water Production Rate Target/Limit (as above)
x	x	FWPT	GWPT	WWPT	CWPT	CWPTL	Water Production Total
x	x					WWPTL	Water Production Total (as above)
x	x	FWPTH	GWPTH	WWPTH			Water Production Total History ( <a href="#">WCONHIST</a> )
x	x	FWIR	GWIR	WWIR	CWIR	CWIRL	Water Injection Rate
x	x					WWIRL	Water Injection Rate (as above)
x	x	FWIRH	GWIRH	WWIRH			Water Injection Rate History ( <a href="#">WCONINJH</a> )
x	x	FWIRT	GWIRT	WWIRT			Water Injection Rate Target/Limit
x	x		GWIRL				Water Injection Rate Target/Limit (as above)
x	x	FWIT	GWIT	WWIT	CWIT	CWITL	Water Injection Total
x	x					WWITL	Water Injection Total (as above)
x	x	FWITH	GWITH	WWITH			Water Injection Total History ( <a href="#">WCONINJH</a> )
x	x	FWPP	GWPP	WWPP			Water Potential Production rate
x					CWPP		Water Potential Production rate (as above)
x	x	FWPP2	GWPP2	WWPP2			Water Potential Production rate (including values calculated on shut/stopped wells)

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x	FWPI	GWPI	WWPI			Water Potential Injection rate
x					CWPI		Water Potential Injection rate (as above)
x	x			WWIP			Water Potential Injection rate (as above)
x	x	FWPI2	GWPI2	WWPI2			Water Potential Injection rate (including values calculated on shut/stopped wells)
x	x			WWIP2			Water Potential Injection rate (including values calculated on shut/stopped wells) (as above)
x	x		GWPGR	WWPGR			Water Production Guide Rate
x	x		GWIGR	WWIGR			Water Injection Guide Rate
	x	FWVPR	GWVPR	WWVPR			Water Voidage Production Rate
	x	FWVPT	GWVPT	WWVPT			Water Voidage Production Total
	x	FWVIR	GWVIR	WWVIR			Water Voidage Injection Rate
	x	FWVIT	GWVIT	WWVIT			Water Voidage Injection Total
x		FWPIR	GWPIR	WWPIR			Ratio of produced water to injected water expressed as a percentage
	x	FWMPR	GWMPR	WWMPR			Water component molar production rate. See Note 1.
	x	FWMPT	GWMPPT	WWMPPT			Water component molar production total. See Note 1.
	x	FWMIR	GW MIR	WW MIR			Water component molar injection rate. See Note 1.
	x	FWMIT	GW MIT	WW MIT			Water component molar injection total. See Note 1.

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.

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x				CGFR	CGFRL	Gas Flow Rate (+ve=prod - ve=inj)
x	x					WGFRL	Gas Flow Rate (as above)
x	x					CGPRL	Gas Flow Rate (as above)
x	x					WGPRL	Gas Flow Rate (as above)
x					CGFRF		Free Gas Flow Rate

E100	E300	Field	Group	Well	Connection	Completion	Information
x					CGFRS		Solution Gas Flow Rate
x					CGFRU		Sum of connection gas flow rates upstream of, and including, this connection
x	x	FGPR	GGPR	WGPR	CGPR		Gas Production Rate Note: In the presence of lift gas (see the <a href="#">VFPPROD</a> keyword), the produced gas reported does not include produced lift gas, only produced formation gas.
	x	FGPRA	GGPRA	WGPRA			Gas Production Rate above GOC
	x	FGPRB	GGPRB	WGPRB			Gas Production Rate below GOC
	x	FGPTA	GGPTA	WGPTA			Gas Production Total above GOC
	x	FGPTB	GGPTB	WGPTB			Gas Production Total below GOC
	x	FGPR1	GGPR1	WGPR1			Gas Production Rate above GOC (molar method)
	x	FGPR2	GGPR2	WGPR2			Gas Production Rate below GOC (molar method)
	x	FGPT1	GGPT1	WGPT1			Gas Production Total above GOC (molar method)
	x	FGPT2	GGPT2	WGPT2			Gas Production Total below GOC (molar method)
	x	FGMR	GGMR	WGMR			Gas Mass Rate
	x	FGMT	GGMT	WGMT			Gas Mass Total
	x	FGDN	GGDN	WGDN			Gas Density at Surface Conditions
x	x	FGPRH	GGPRH	WGPRH			Gas Production Rate History ( <a href="#">WCONHIST</a> )
x	x	FGPRT	GGPRT	WGPRT			Gas Production Rate Target/Limit
x	x		GGPRL				Gas Production Rate Target/Limit (as above)
x		FGPRF	GGPRF	WGPRF			Free Gas Production Rate
x		FGPRS	GGPRS	WGPRS			Solution Gas Production Rate
x	x	FGPT	GGPT	WGPT	CGPT	CGPTL	Gas Production Total
x	x					WGPTL	Gas Production Total (as above)

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x	FGPTH	GGPTH	WGPTH			Gas Production Total History (WCONHIST)
x		FGPTF	GGPTF	WGPTF	CGPTF		Free Gas Production Total
x		FGPTS	GGPTS	WGPTS	CGPTS		Solution Gas Production Total
x	x	FGIR	GGIR	WGIR	CGIR	CGIRL	Gas Injection Rate
x	x					WGIRL	Gas Injection Rate (as above)
x	x	FGIRH	GGIRH	WGIRH			Gas Injection Rate History (WCONINJH)
x	x	FGIRT	GGIRT	WGIRT			Gas Injection Rate Target/Limit
x	x		GGIRL				Gas Injection Rate Target/Limit (as above)
x	x	FGIT	GGIT	WGIT	CGIT	CGITL	Gas Injection Total
x	x					WGITL	Gas Injection Total (as above)
x	x	FGITH	GGITH	WGITH			Gas Injection Total History (WCONINJH)
x	x	FGPP	GGPP	WGPP			Gas Potential Production rate
x	x				CGPP		Gas Potential Production rate (as above)
x	x	FGPP2	GGPP2	WGPP2			Gas Potential Production rate (including values calculated on shut/stopped wells)
x		FGPPS	GGPPS	WGPPS			Solution (dissolved) Gas Potential Production rate
x		FGPPS2	GGPPS2	WGPPS2			Solution (dissolved) Gas Potential Production rate (including values calculated on shut/stopped wells)
x		FGPPF	GGPPF	WGPPF			Free Gas Potential Production rate
x		FGPPF2	GGPPF2	WGPPF2			Free Gas Potential Production rate (including values calculated on shut/stopped wells)
x	x	FGPI	GGPI	WGPI			Gas Potential Injection rate
x	x				CGPI		Gas Potential Injection rate (as above)
x	x			WGIP			Gas Potential Injection rate (as above)
x	x	FGPI2	GGPI2	WGPI2			Gas Potential Injection rate (including values calculated on shut/stopped wells)

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x			WGIP2			Gas Potential Injection rate (including values calculated on shut/stopped wells) (as above)
x	x		GGPGR	WGPGR			Gas Production Guide Rate
x	x		GGIGR	WGIGR			Gas Injection Guide Rate
x	x	FSGR	GSGR				Sales Gas Rate
x	x	FGSR	GGSR				Sales Gas Rate (as above)
x	x	FSGT	GSGT				Sales Gas Total
x	x	FGST	GGST				Sales Gas Total (as above)
	x	FSMF	GSMF				Sales Gas Mole Fraction. See Note 1.
	x	FFGR	GFGR				Fuel Gas Rate, at and below this group
	x	FFGT	GFGT				Fuel Gas cumulative Total, at and below this group
	x	FFMF	GFMF				Fuel Gas Mole Fraction. See Note 1.
x		FGCR	GGCR				Gas Consumption Rate, at and below this group
x		FGCT	GGCT				Gas Consumption Total, at and below this group
x		FGIMR	GGIMR				Gas Import Rate, at and below this group
x		FGIMT	GGIMT				Gas Import Total, at and below this group
x	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	FWGPR	GWGPR	WWGPR			Wet Gas Production Rate
	x	FWGPT	GWGPT	WWGPT			Wet Gas Production Total
	x	FWGPRH	GWGPRH	WWGPRH			Wet Gas Production Rate History ( <a href="#">WCONHIST</a> )
	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.

E100	E300	Field	Group	Well	Connection	Completion	Information
	x	FEMF	GEMF				Export Gas Mole Fraction. See Note 1.
	x	FEXGR	GEXGR				Excess Gas Rate. See Note 2.
	x	FEXGT	GEXGT				Excess Gas Total. See Note 2.
	x	FRGR	GRGR				Re-injection Gas Rate. See Note 2.
	x	FRGT	GRGT				Re-injection Gas Total. See Note 2.
	x	FGnPR	GGnPR	WGnPR			nth separator stage gas rate (n = 1 to 99).  Note that the gas volume rate is calculated at standard conditions and not at the temperature and pressure of the separator stage
	x	FGnPT	GGnPT	WGnPT			nth separator stage gas total (n = 1 to 99)
	x	FGVPR	GGVPR	WGVPR			Gas Voidage Production Rate
	x	FGVPT	GGVPT	WGVPT			Gas Voidage Production Total
	x	FGVIR	GGVIR	WGVIR			Gas Voidage Injection Rate
	x	FGVIT	GGVIT	WGVIT			Gas Voidage Injection Total
	x	FGQ	GGQ	WGQ	CGQ		Gas Quality

Table 2.13: Well and group gas flows

The well and group liquid flow options are:

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x				CLFR	CLFRL	Liquid Flow Rate (+ve=prod - ve=inj)
x	x					WLFRL	Liquid Flow Rate (as above)
x	x	FLPR	GLPR	WLPR			Liquid Production Rate
x	x	FLPRH	GLPRH	WLPRH			Liquid Production Rate History ( <a href="#">WCONHIST</a> )
x	x	FLPRT	GLPRT	WLPRT			Liquid Production Rate Target/Limit
x	x		GLPRL				Liquid Production Rate Target/Limit (as above)
x	x	FLPT	GLPT	WLPT	CLPT	CLPTL	Liquid Production Total
x	x					WLPTL	Liquid Production Total (as above)

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x	FLPTH	GLPTH	WLPTH			Liquid Production Total History (WCONHIST)

Table 2.14: Well and group liquid flows

The well and group linearly combined flow options are:

E100	E300	Field	Group	Well	Information
x	x	FJPR	GJPR	WJPR	Linearly Combined Production Rate
x	x	FJPRH	GJPRH	WJPRH	Linearly Combined Production Rate History (WCONHIST)
x	x	FJPRT	GJPRT	WJPRT	Linearly Combined Production Rate Target/Limit
x	x		GJPRL		Linearly Combined Production Rate Target/Limit (as above)
x	x	FJPT	GJPT	WJPT	Linearly Combined Production Total
x	x	FJPTH	GJPTH	WJPTH	Liquid Production Total History (WCONHIST)

Table 2.15: Well and group linearly combined flows

The well and group reservoir volume flow options are:

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x				CVFR	CVFRL	Reservoir (Res) Volume Flow Rate (+ve=prod -ve=inj)
x	x					WVFRL	Res Volume Flow Rate (as above)
	x					CVFRL	Res Volume Production Flow Rate
	x					WVFRL	Res Volume Production Flow Rate (as above)
x	x					CVIRL	Res Volume Injection Flow Rate
x	x					WVIRL	Res Volume Injection Flow Rate (as above)
x	x	FVPR	GVPR	WVPR			Res Volume Production Rate
	x				CVPR		Res Volume Production Rate
x	x	FVPRT	GVPRT	WVPRT			Res Volume Production Rate Target/Limit
x	x		GVFRL				Res Volume Production Rate Target/Limit (as above)
x	x	FVPT	GVPT	WVPT	CVPT	CVPTL	Res Volume Production Total
x	x					WVPTL	Res Volume Production Total (as above)
x	x		GVFGR	WVFGR			Res Volume Production Guide Rate
x	x	FVIR	GVIR	WVIR	CVIR		Res Volume Injection Rate
x	x	FVIRT	GVIRT	WVIRT			Res Volume Injection Rate Target/Limit

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x		GVIRL				Res Volume Injection Rate Target/ Limit (as above)
x	x	FVIT	GVIT	WVIT	CVIT	CVITL	Res Volume Injection Total
x	x					WVITL	Res Volume Injection Total (as above)

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](#)”

E300	Field	Group	Information
x	FEGR	GEGR	Export Gas Rate. See Note 1.
x	FEGT	GEGT	Export Gas Total. See Note 1.
x	FEOR	GEOR	Export Oil Rate
x	FEOT	GEOT	Export Oil Total
x	FENR	GENR	Export NGL Rate
x	FENT	GENT	Export NGL Total
x	FEMF	GEMF	Export Gas Mole Fraction. See Note 2.
x	FEOMF	GEOMF	Export Oil Mole Fraction. See Note 2.
x	FENMF	GENMF	Export NGL Mole Fraction. See Note 2.

Table 2.17: Group export flows (ECLIPSE 300)

The well and group calorific value flow options are:

E300	Field	Group	Well	Connection	Information
x	FCVPR	GCVPR	WCVPR	CCVPR	Calorific Value Production Rate
x	FCVPT	GCVPT	WCVPT		Calorific Value Production Total
x	FCVPP	GCVPP	WCVPP		Calorific Value Production Potential
x	FCVPP2	GCVPP2	WCVPP2		Calorific Value Production Potential (including values calculated on shut/stopped wells)

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

The well and group production ratio options are:

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x	FWCT	GWCT	WWCT	CWCT	CWCTL	Water Cut
x	x					WWCTL	Water Cut (as above)
x	x	FWCTH	GWCTH	WWCTH			Water Cut History ( <a href="#">WCONHIST</a> )
x	x	FGOR	GGOR	WGOR	CGOR	CGORL	Gas-Oil Ratio

E100	E300	Field	Group	Well	Connection	Completion	Information
x	x					WGORL	Gas-Oil Ratio (as above)
x	x	FGORH	GGORH	WGORH			Gas-Oil Ratio History (WCONHIST)
x	x	FOGR	GGR	WGR	COGR	COGRL	Oil-Gas Ratio
x	x					WOGRL	Oil-Gas Ratio (as above)
x	x	FOGRH	GGRH	WGRH			Oil-Gas Ratio History (WCONHIST)
x	x	FWGR	GWGR	WWGR	CWGR	CWGRL	Water-Gas Ratio
x	x					WWGRL	Water-Gas Ratio (as above)
x	x	FWGRH	GWGRH	WWGRH			Water-Gas Ratio History (WCONHIST)
x	x	FGLR	GGLR	WGLR	CGLR		Gas-Liquid Ratio
x						CGLRL	Gas-Liquid Ratio
x						WGLRL	Gas-Liquid Ratio (as above)
x		FGLRH	GGLRH	WGLRH			Gas-Liquid Ratio History (WCONHIST)
x				WBGLR			Bottom hole Gas-Liquid Ratio

Table 2.19: Well and group production ratios

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

- No units are given in FIELD, as noted in "The productivity index" in the *ECLIPSE Technical Description*

E100	E300	Well	Connection	Completion	Information
x	x		CPR		Connection Pressure
x	x			CPRL	Average Connection Pressure in lumped completion
x	x	WBHP			Bottom Hole Pressure
x	x	WBHPH			Bottom Hole Pressure History (WCONHIST, WCONINJH)
x	x	WTHP			Tubing Head Pressure
x	x	WTHPH			Tubing Head Pressure History (WCONHIST, WCONINJH)
x	x	WPI	CPI		Productivity Index of well's preferred phase. See Note 1.
x	x	WPIO			Oil phase PI
x	x	WPIG			Gas phase PI
x	x	WPIW			Water phase PI
x	x	WPIL			Liquid phase PI
x	x		CTFAC		Connection Transmissibility Factor

E100	E300	Well	Connection	Completion	Information
x	x		CDBF		Blocking factor for generalized pseudo-pressure method
x	x	WBP			One-point Pressure Average (see keywords <a href="#">WPAVE</a> and <a href="#">WWPAVE</a> )
x	x	WBP4			Four-point Pressure Average (see keywords <a href="#">WPAVE</a> and <a href="#">WWPAVE</a> )
x	x	WBP5			Five-point Pressure Average (see keywords <a href="#">WPAVE</a> and <a href="#">WWPAVE</a> )
x	x	WBP9			Nine-point Pressure Average (see keywords <a href="#">WPAVE</a> and <a href="#">WWPAVE</a> )
x	x	WPI1			Productivity Index based on the value of WBP. See Note 1.
x	x	WPI4			Productivity Index based on the value of WBP4. See Note 1.
x	x	WPI5			Productivity Index based on the value of WBP5. See Note 1.
x	x	WPI9			Productivity Index based on the value of WBP9. See Note 1.
x		WHD			Hydraulic head in well based on the reference depth given in <a href="#">HYDRHEAD</a> and the well's reference depth. Requires Water Services license.
x		WHDF			Hydraulic head in well based on the reference depth given in <a href="#">HYDRHEAD</a> and the well's reference depth calculated at freshwater conditions. Requires Water Services license.

Table 2.20: Well pressures and PIs

The well and group control options are:

E100	E300	Field	Group	Well	Information
x	x			WSTAT	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.

E100	E300	Field	Group	Well	Information
x	x			WMCTL	<p>Mode of Control:</p> <p>0.0 = SHUT/STOP,</p> <p>1.0 = ORAT,</p> <p>2.0 = WRAT,</p> <p>3.0 = GRAT,</p> <p>4.0 = LRAT,</p> <p>5.0 = RESV,</p> <p>6.0 = THP,</p> <p>7.0 = BHP,</p> <p>9.0 = CRAT,</p> <p>11.0 = GOR Penalty,</p> <p>12.0 = Drawdown,</p> <p>13.0 = NGL,</p> <p>30.0 = Availability,</p> <p>31.0 = REIN,</p> <p>32.0 = TMRA,</p> <p>33.0 = WGRA,</p> <p>34.0 = CVAL,</p> <p>-1.0 = well under group control (ECLIPSE 300),</p> <p>-ve if rate target was set by group (ECLIPSE 100),</p> <p>41.0 = STRA (Thermal: Steam Rate),</p> <p>42.0 = SATP (Thermal: Saturated Pressure),</p> <p>43.0 = SATT (Thermal: Saturated Temperature).</p> <p>The output is rendered as mnemonics in the .RSM file.</p>

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

E100	E300	Field	Group	Well	Information
x	x	FMCTG	GMCTG		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:

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

E100	E300	Field	Group	Well	Information
x	x	FMWWO	GMWWO		Number of workover events this timestep
x	x	FMWWT	GMWWT		Number of workover events in total
	x		GMWL		Number of wells grouped together in a well list (either static or dynamic)

Table 2.22: Group well counts

The well and group energy rate options are:

E100	E300	Field	Group	Well	Information
x	x	FEPR	GEPR	WEPR	Energy Production Rate
x	x	FEPT	GEPT	WEPT	Energy Production Total
	x	FEIR	GEIR	WEIR	Energy Injection Rate
	x	FEIT	GEIT	WEIT	Energy Injection Total
	x	FHTR		WHTR	Heater energy rate (WHTR requires heater names)
	x	FHTT		WHTT	Heater energy total (WHTT requires heater names)
	x			WENE	Molar Energy (specified)
	x			WENO	Molar Energy of oil (specified)
	x			WENG	Molar Energy of gas (specified)
	x			WENW	Molar Energy of water (specified)
	x			WAEN	Molar Energy (actual)
	x			WAEO	Molar Energy of oil (actual)
	x			WAEG	Molar Energy of gas (actual)
	x			WAEW	Molar Energy of water (actual)
	x			WENR	Energy Rate
	x			WEOR	Energy rate for oil
	x			WEGR	Energy rate for gas
	x			WEWR	Energy rate for water
	x			WENT	Cumulative energy
	x			WERO	Ratio of oil energy to total energy
	x			WERG	Ratio of gas energy to total energy
	x			WERW	Ratio of water energy to total energy
	x			WPOFF	Difference between the bottom hole pressure and the highest water saturation pressure in the grid blocks connected to the well
	x			WTOFF	Difference between the water saturation temperature at the bottom hole pressure and the hottest grid block connected to the well

E100	E300	Field	Group	Well	Information
	x			WTSUB	Difference between the water saturation temperature at the bottom hole pressure and the well bore temperature at the bottom hole reference point

Table 2.23: Well and group energy rates

The solid production totals mnemonics are:

E300	Field	Group	Well	Information
x	F <sub>SMR</sub>	G <sub>SMR</sub>	W <sub>SMR</sub>	Solid mass production rate
x	F <sub>SMT</sub>	G <sub>SMT</sub>	W <sub>SMT</sub>	Solid mass production total
x	F <sub>SVR</sub>	G <sub>SVR</sub>	W <sub>SVR</sub>	Solid volume production rate
x	F <sub>SVT</sub>	G <sub>SVT</sub>	W <sub>SVT</sub>	Solid volume production total

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:

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

E100	E300	Field	Group	Well	Connection	Information
x	x				CDSM	Current mass of scale deposited
x	x				CDSML	Current mass of scale deposited per unit perforation length
x	x				CDSF	PI multiplicative factor due to scale damage
x	x		GEFF	WEFF		Efficiency Factor
x	x			WEFFG		Product of efficiency factors of the well and all its superior groups
x				WALQ		Well Artificial Lift Quantity
x	x			WMVFP		VFP table number used by the well
	x	FNLPR	GNLPR	WNLPR		NGL Production Rate
	x	FNLPT	GNLPT	WNLPT		NGL Production Total
	x	FNLPRH	GNLPRH	WNLPRH		NGL Production Rate History ( <a href="#">WCONHIST</a> )
	x	FNLPTH	GNLPTH	WNLPTH		NGL Production Total History ( <a href="#">WCONHIST</a> )
	x			WNLprt		NGL Production Rate Target

E100	E300	Field	Group	Well	Connection	Information
	x	FAMF	GAMF	WAMF	CAMF	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.
	x	FXMF	GXMF	WXMF		Liquid Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.
	x	FYMF	GYMF	WYMF		Vapor Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.
	x	FXMF <sup>n</sup>	GXMF <sup>n</sup>	WXMF <sup>n</sup>		Liquid Mole Fraction for nth separator stage (n = 1 to 99). Group and Field level mnemonics only include production wells. See Note 1.
	x	FYMF <sup>n</sup>	GYMF <sup>n</sup>	WYMF <sup>n</sup>		Vapor Mole Fraction for nth separator stage (n = 1 to 99). Group and Field level mnemonics only include production wells. See Note 1.
	x	FZMF	GZMF	WZMF	CZMF	Total Mole Fraction. Group and Field level mnemonics only include production wells. See Note 1.
	x	FCMPR	GCMPR	WCMPR		Hydrocarbon Component Molar Production Rates. See Note 1.
	x	FCMPT	GCMPT	WCMPT		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.
	x	FCMIR	GCMIR	WCMIR		Hydrocarbon Component Molar Injection Rates. See Note 1.
	x	FCMIT	GCMIT	WCMIT		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.
	x			WCGIR		Hydrocarbon Component Gas Injection Rate. See Note 1.
	x			WCGPR		Hydrocarbon Component Gas Production Rate. See Note 1.

E100	E300	Field	Group	Well	Connection	Information
	x			WCOPR		Hydrocarbon Component Oil Production Rate. See Note 1.
	x	FHMIR	GHMIR	WHMIR		Hydrocarbon Molar Injection Rate
	x	FHMIT	GHMIT	WHMIT		Hydrocarbon Molar Injection Total
	x	FHMPR	GHMPR	WHMPR		Hydrocarbon Molar Production Rate
	x	FHMPT	GHMPT	WHMPT		Hydrocarbon Molar Production Total
	x	FCHMR	GCHMR	WCHMR	CKFR	Hydrocarbon Component Molar Flow Rate. See Note 1.
	x	FCHMT	GCHMT	WCHMT	CKFT	Hydrocarbon Component Molar Flow Total. See Note 1.
	x	FCWGPR	GCWGPR	WCWGPR		Hydrocarbon Component Wet Gas Production Rate. See Note 1.
	x	FCWGPT	GCWGPT	WCWGPT		Hydrocarbon Component Wet Gas Production Total. See Note 1.
	x	FCWGIR	GCWGIR	WCWGIR		Hydrocarbon Component Wet Gas Injection Rate. See Note 1.
	x	FCWGIT	GCWGIT	WCWGIT		Hydrocarbon Component Wet Gas Injection Total. See Note 1.
	x	FCGMR	GCGMR	WCGMR		Hydrocarbon component molar rates in the gas phase. See Note 1.
	x	FCGMT	GCGMT	WCGMT		Hydrocarbon component molar totals in the gas phase. See Note 1.
	x	FCOMR	GCOMR	WCOMR		Hydrocarbon component molar rates in the oil phase. See Note 1.
	x	FCOMT	GCOMT	WCOMT		Hydrocarbon component molar totals in the oil phase. See Note 1.
	x	FCNMR	GCNMR	WCNMR		Hydrocarbon component molar rates in the NGL phase. See Note 1.
	x	FCNWR	GCNWR	WCNWR		Hydrocarbon component mass rates in the NGL phase
	x	FCGMRn	GCGMRn	WCGMRn		Hydrocarbon component molar rates in the gas phase for nth separator stage (n = 1 to 9). See Note 1.
	x	FCGRn	GGRn	WGRn		Hydrocarbon component molar rates in the gas phase for nth separator stage (n = 1 to 99). See Note 1.
	x	FCOMRn	GCOMRn	WCOMRn		Hydrocarbon component molar rates in the oil phase for nth separator stage (n = 1 to 9). See Note 1.
	x	FCORn	GORn	WORn		Hydrocarbon component molar rates in the oil phase for nth separator stage (n = 1 to 99). See Note 1.

E100	E300	Field	Group	Well	Connection	Information
	x	FMUF	GMUF	WMUF		Make-up fraction (calculated at reinjecting wells)
	x	FAMR	GAMR			Make-up gas rate (calculated at source of reinjection gas)
	x	FAMT	GAMT			Make-up gas total (calculated at source of reinjection gas)
x	x				CDFAC	D-factor for flow dependent skin factor
x	x			WTHT		Tubing Head Temperature
	x			WMMW		Mean molecular weight of wellstream
x	x	FGSPR	GGSPR			Target sustainable rate for most recent sustainable capacity test for gas.
x	x	FGSRL	GGSRL			Maximum tested rate sustained for the test period during the most recent sustainable capacity test for gas.
x	x	FGSRU	GGSRU			Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for gas.
x	x	FGSSP	GGSSP			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.
x	x	FGSTP	GGSTP			Test period for the most recent sustainable capacity test for gas.
x	x	FOSPR	GOSPR			Target sustainable rate for most recent sustainable capacity test for oil.
x	x	FOSRL	GOSRL			Maximum tested rate sustained for the test period during the most recent sustainable capacity test for oil.
x	x	FOSRU	GOSRU			Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for oil.
x	x	FOSSP	GOSSP			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.
x	x	FOSTP	GOSTP			Test period for the most recent sustainable capacity test for oil.
x	x	FWSPR	GWSPR			Target sustainable rate for most recent sustainable capacity test for water.

E100	E300	Field	Group	Well	Connection	Information
x	x	FWSRL	GWSRL			Maximum tested rate sustained for the test period during the most recent sustainable capacity test for water.
x	x	FWSRU	GWSRU			Minimum tested rate not sustained for the test period during the most recent sustainable capacity test for water.
x	x	FWSSP	GWSSP			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.
x	x	FWSTP	GWSTP			Test period for the most recent sustainable capacity test for water.
	x	FGPRG	GGPRG			Gas production rate (production summed then separated)
	x	FOPRG	GOPRG			Oil production rate (production summed then separated)
	x	FNLPRG	GNLPRG			NGL production rate (production summed then separated)
	x	FXMFG	GXMFG			Liquid mole fraction (production summed then separated). See Note 1.
	x	FYMFG	GYMFG			Vapor mole fraction (production summed then separated). See Note 1.
	x	FCOMRG	GCOMRG			Hydrocarbon component molar rates in the oil phase (production summed then separated). See Note 1.
	x	FCGMRG	GCGMRG			Hydrocarbon component molar rates in the gas phase (production summed then separated). See Note 1.
	x	FCNMRG	GCNMRG			Hydrocarbon component molar rates in the NGL phase (production summed then separated). See Note 1.
x	x			WPWE0		Well drilled indicator. Set to one if the well was drilled this timestep and zero at other times.
x	x			WPWE1		Connections opened indicator. Set to the number of connections opened in the well this timestep if the well is not shut or stopped.
x	x			WPWE2		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.

E100	E300	Field	Group	Well	Connection	Information
x	x			WPWE3		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.
x	x			WPWE4		Well stopped indicator. Set to one if the well is stopped this timestep and zero otherwise.
x	x			WPWE5		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.
x	x			WPWE6		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.
x	x			WPWE7		Well shut indicator. Set to one if the well is shut this timestep and zero otherwise.
x	x			WPWEM		<a href="#">WELEVNT</a> output mnemonic.
x	x			WDRPR		Well drilling priority.
x	x			WBHWCn		Derivative of well BHP with respect to parameter n (n=1 to 99). See <a href="#">GWRTWCV</a> .
x	x			WGFWCn		Derivative of well gas flow rate with respect to parameter n, n=1 to 99. See <a href="#">GWRTWCV</a> .
x	x			WOFWCn		Derivative of well oil flow rate with respect to parameter n, n=1 to 99. See <a href="#">GWRTWCV</a> .
x	x			WWFWCn		Derivative of water flow rate with respect to parameter n, n=1 to 99. See <a href="#">GWRTWCV</a> .

Table 2.25: Miscellaneous well and group quantities

ECLIPSE 300 Thermal

The miscellaneous well quantity options are:

E300	Field	Group	Well	Information
x			WSQU	Injected steam quality
x			WTEMP	Temperature
x	FSTPR	GSTPR	WSTPR	Steam production rate
x	FSTPT	GSTPT	WSTPT	Steam production total
x	FOSRC	GOSRC		Cumulative oil steam ratio
x	FSORC	GSORC		Cumulative steam oil ratio
x	FHSRC	GHSRC		Cumulative hydrocarbon steam ratio
x	FSHRC	GSHRC		Cumulative steam hydrocarbon ratio

Table 2.26: Miscellaneous well quantities (ECLIPSE 300 Thermal)

## Secondary group and well quantities (ECLIPSE 300)

The secondary group and well quantities are:

E300	Field	Group	Well	Information
x			WDOPR	Secondary well oil production rate.
x			WDWPR	Secondary well water production rate.
x			WDGPR	Secondary well gas production rate.
x			WDLPR	Secondary well liquid production rate.
x			WDVPR	Secondary well reservoir volume production rate.
x			WDOPT	Secondary well oil production total.
x			WDWPT	Secondary well water production total.
x			WDGPT	Secondary well gas production total.
x			WDLPT	Secondary well liquid production total.
x			WDVPT	Secondary well reservoir volume production total.
x			WDWIR	Secondary well water injection rate.
x			WDGIR	Secondary well gas injection rate.
x			WDWIT	Secondary well water injection total.
x			WDGIT	Secondary well gas injection total.
x			WDOGR	Secondary well oil-gas ratio.
x			WDGOR	Secondary well gas-oil ratio.
x			WDGLR	Secondary well gas-liquid ratio.
x			WDWGR	Secondary well water-gas ratio.
x			WDWCT	Secondary well water cut.
x			WDTEM	Secondary well temperature.
x			WDPR	Secondary well pressure.
x	FDOPR	GDOPR		Oil production rate including secondary well contributions.
x	FDWPR	GDWPR		Water production rate including secondary well contributions.
x	FDGPR	GDGPR		Gas production rate including secondary well contributions.
x	FDLPR	GDLPR		Liquid production rate including secondary well contributions.
x	FDVPR	GDVPR		Reservoir volume production rate including secondary well contributions.
x	FDOPT	GDOPT		Oil production total including secondary well contributions.
x	FDWPT	GDWPT		Water production total including secondary well contributions.
x	FDGPT	GDGPT		Gas production total including secondary well contributions.
x	FDLPT	GDLPT		Liquid production total including secondary well contributions.
x	FDWIR	GDWIR		Water injection rate including secondary well contributions
x	FDGIR	GDGIR		Gas injection rate including secondary well contributions.
x	FDWIT	GDWIT		Water injection total including secondary well contributions.

E300	Field	Group	Well	Information
x	FDGIT	GDGIT		Gas injection total including secondary well contributions.
x	FDOGR	GDOGR		Oil-gas ratio including secondary well contributions.
x	FDGOR	GDGOR		Gas-oil ratio including secondary well contributions.
x	FDGLR	GDGLR		Gas-liquid ratio including secondary well contributions.
x	FDWGR	GDWGR		Water-gas ratio including secondary well contributions.
x	FDWCT	GDWCT		Water cut including secondary well contributions.

Table 2.27: Secondary group and well quantities (ECLIPSE 300)

## Well list quantities

The well list quantity options are:

E300	Well list	Information
x	GOPR	Oil Production Rate
x	GOPT	Oil Production Total
x	GWPR	Water Production Rate
x	GWPT	Water Production Total
x	GWIR	Water Injection Rate
x	GWIT	Water Injection Total
x	GGPR	Gas Production Rate
x	GGPT	Gas Production Total
x	GGIR	Gas Injection Rate
x	GGIT	Gas Injection Total
x	GVPR	Voidage Production Rate
x	GVPT	Voidage Production Total
x	GVIR	Voidage Injection Rate
x	GVIT	Voidage Injection Total
x	GLPR	Liquid Production Rate
x	GLPT	Liquid Production Total
x	GGOR	Gas to Oil Ratio
x	GAGR	Oil to Gas Ratio
x	GWCT	Water Cut
x	GWGR	Water to Gas Ratio
x	GGLR	Gas to Liquid Ratio
x	GOPRH	Oil Production Rate History ( <a href="#">WCONHIST</a> )
x	GGPRH	Gas Production Rate History ( <a href="#">WCONHIST</a> )
x	GWPRH	Water Production Rate History ( <a href="#">WCONHIST</a> )
x	GLPRH	Liquid Production Rate History ( <a href="#">WCONHIST</a> )

E300	Well list	Information
x	GOPH	Oil Production Total History (WCONHIST)
x	GGPH	Gas Production Total History (WCONHIST)
x	GWPH	Water Production Total History (WCONHIST)
x	GLPH	Liquid Production Total History (WCONHIST)
x	GGIRH	Gas Injection Rate (WCONINJE)
x	GWIRH	Water Injection Rate (WCONINJE)
x	GGITH	Gas Injection Total (WCONINJE)
x	GWITH	Water Injection Total (WCONINJE)

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".

E100	E300	Field	Region	Block	Information
x	x			BOSAT	Oil SATuration
x	x			BSOIL	Oil Saturation (same as BOSAT)
	x			BSOILM	Mobile Oil Saturation (SOR option)
	x			BSOILR	Residual Oil Saturation (SOR option)
	x			BCOILR	Residual Oil compressibility (SOR option)
x	x	FOSAT	ROSAT		Oil SATuration average value
x	x	FOIP	ROIP	BOIP	Oil In Place (in liquid and wet gas phases). See Note 1.
x	x	FOIPL	ROIPL	BOIPL	Oil In Place (Liquid phase). See Note 1.
x	x	FOIPG	ROI PG	BOIPG	Oil In Place (Gas phase). See Note 1.
	x	FOIPR			Oil Reservoir Volume in Place
	x	FCOM	RCOM		Molar amount of specified component dissolved in oil. See Note 2.
	x	FOIPA	ROI PA		Oil in Place above TRACK depth. See Note 1.
	x	FOIPB	ROI PB		Oil in Place below TRACK depth. See Note 1.
x	x	FPPO	RPPO	BPPO	Oil Potential (or "depth corrected pressure")
x	x			BVOIL	Oil Viscosity
x	x			BOVIS	Oil VIScosity (same as BVOIL)
x		FOVIS	ROI VIS		Oil VIScosity average value
x	x			BDENO	Oil reservoir Density
x	x			BODEN	Oil reservoir DENsity (same as BDENO)

E100	E300	Field	Region	Block	Information
x		FODEN	RODEN		Oil reservoir DENsity average value
	x			BBOIL	Oil Reservoir Molar Densities
	x			BFUGO	Hydrocarbon component FUGacities in Oil ( <a href="#">DIFFUSE</a> option only, see Note 2).
x			ROP		Net oil production from a region
x	x		ROPR		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.
x	x		ROPT		Oil Production Total
x	x		ROIR		Oil Injection Rate
x	x		ROIT		Oil Injection Total
x	x			BFLOOI	Inter-block oil flow rate in the positive I direction. This value is, by default, calculated based on the first FIP set.
x	x			BFLOOJ	Inter-block oil flow rate in the positive J direction. This value is, by default, calculated based on the first FIP set.
x	x			BFLOOK	Inter-block oil flow rate in the positive K direction. This value is, by default, calculated based on the first FIP set.
x				BVELOI	Oil velocity for positive I direction.
x				BVELOJ	Oil velocity for positive J direction.
x				BVELOK	Oil velocity for positive K direction.

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”](#).

E100	E300	Field	Region	Block	Information
x	x			BWSAT	Water SATuration
x	x			BSWAT	Water Saturation (same as BWSAT)
x	x	FWSAT	RWSAT		Water SATuration average value
x	x	FWIP	RWIP	BWIP	Water In Place
	x		RNLIP		NGL In Place
	x	FWIPR			Water Reservoir Volume in Place
	x	FCWM	RCWM		Molar amount of specified component dissolved in water. See Note 1.
	x	FWCD	RWCD		CO <sub>2</sub> (“carbon dioxide”) dissolved in water phase ( <a href="#">CO2STORE</a> option only)
x	x	FPPW	RPPW	BPPW	Water Potential (or “depth corrected pressure”)
x	x			BVWAT	Water Viscosity
x	x			BWVIS	Water VIScosity (same as BVWAT)

E100	E300	Field	Region	Block	Information
x		FWVIS	RWVIS		Water VIScosity average value
x	x			BDENW	Water reservoir Density
x	x			BWDEN	Water reservoir DENsity (same as BDENW)
x		FWDEN	RWDEN		Water reservoir DENsity average value
	x			BBWAT	Water Reservoir Molar Densities
	x			BXWAT	Fractional Saturation of Water (Gravity Drainage Option only)
x			RWP		Net water production from a region (minus injection from wells and analytic aquifers)
x	x		RWPR		Water Production Rate
x	x		RWPT		Water Production Total
x	x		RWIR		Water Injection Rate
x	x		RWIT		Water Injection Total
x	x			BFLOWI	Inter-block water flow rate in the positive I direction.
x	x			BFLOWJ	Inter-block water flow rate in the positive J direction.
x	x			BFLOWK	Inter-block water flow rate in the positive K direction.
x				BVELWI	Water velocity in the positive I direction.
x				BVELWJ	Water velocity in the positive J direction.
x				BVELWK	Water velocity in the positive K direction.

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"](#).

E100	E300	Field	Region	Block	Information
x	x			BGSAT	Gas SATuration
x	x			BSGAS	Gas Saturation (same as BGSAT)
x	x	FGSAT	RGSAT		Gas SATuration average value
x	x	FGIP	RGIP	BGIP	Gas In Place (in liquid and gas phases). See Note 1.
x	x	FGIPL	RGIPL	BGIPL	Gas In Place (Liquid phase). See Note 1.
x	x	FGIPG	RGIPG	BGIPG	Gas In Place (Gas phase). See Note 1.
	x	FAGIP			Gas In Place (Aqueous phase) (compatible with <a href="#">CO2SOL</a> , <a href="#">H2SSOL</a> and <a href="#">GASSOL</a> )
	x	FGIPA	RGIPA		Gas in Place above <a href="#">TRACK</a> depth. See Note 1.
	x	FGIPB	RGIPB		Gas in Place below <a href="#">TRACK</a> depth. See Note 1.
	x	FGIPR			Gas Reservoir Volume in Place

E100	E300	Field	Region	Block	Information
	x	FCGMI	RCGMI		Molar amount of specified component trapped in gas. See Note 2.
	x	FCGMM	RCGMM		Molar amount of specified component mobile in gas. See Note 2.
	x	FGCDI	RGCDI		CO <sub>2</sub> (“carbon dioxide”) trapped (immobile) in gas phase (CO2STORE option only)
	x	FGCDM	RGCDM		CO <sub>2</sub> (“carbon dioxide”) mobile in gas phase (CO2STORE option only)
x	x	FPPG	RPPG	BPPG	Gas Potential (or “depth corrected pressure”)
x	x			BVGAS	Gas Viscosity
x	x			BGVIS	Gas VISCosity (same as BVGAS)
x		FGVIS	RGVIS		Gas VISCosity average value
x	x			BDENG	Gas reservoir Density
x	x			BGDEN	Gas reservoir DENsity (same as BDENG)
x		FGDEN	RGDEN		Gas reservoir DENsity average value
	x			BBGAS	Gas Reservoir Molar Densities
	x			BFUGG	Hydrocarbon component FUGacities in Gas (DIFFUSE option only, see Note 2)
	x			BXGAS	Fractional Saturation of Gas (Gravity Drainage Option only)
x			RGP		Net gas production from a region (minus injection)
x	x		RGPR		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.
x			RGPRF		Free Gas Production Rate
x			RGPRS		Solution Gas Production Rate
x	x		RGPT		Gas Production Total
x			RGPTF		Free Gas Production Total
x			RGPTS		Solution Gas Production Total
x	x		RGIR		Gas Injection Rate
x	x		RGIT		Gas Injection Total
	x		RNLPR		NGL Production Rate
	x		RNLPT		NGL Production Total
x	x			BFLOGI	Inter-block gas flow rate in the positive I direction. This value is, by default, calculated based on the first FIP set.
x	x			BFLOGJ	Inter-block gas flow rate in the positive J direction. This value is, by default, calculated based on the first FIP set.
x	x			BFLOGK	Inter-block gas flow rate in the positive K direction. This value is, by default, calculated based on the first FIP set.
x				BVELGI	Gas velocity in the positive I direction.

E100	E300	Field	Region	Block	Information
x				BVELGJ	Gas velocity in the positive J direction.
x				BVELGK	Gas velocity in the positive K direction.

Table 2.31: Grid block and region gas quantities

ECLIPSE 300

The grid block solid quantity options are:

E300	Block	Information
x	BBSOL	Solid reservoir molar density
x	BDENS	Solid Density
x	BHSOL	Solid molar enthalpy (Thermal option)
x	BSDEN	Solid DENSity (same as BDENS)
x	BSOLADS	Adsorbed solid concentration
x	BSOLADW	Adsorbed wetting-solid concentration
x	BSOLWET	Solid wettability interpolation parameter
x	BSSAT	Solid SATuration
x	BSSOLID	Solid Saturation (same as BSSAT)
x	BSSFRAC	Suspended Solid FRACTION
x	BSMMULT	Solid Mobility MULTiplier

Table 2.32: Grid block solid quantities (ECLIPSE 300)

The block mobility quantity options are: given in the table below. Notes for this table:

- See [“Format of SUMMARY keywords with a component index”](#).

E300	Block	Information
x	BGMCO	Generalized mobilities of hydrocarbon components in oil. See Note 1.
x	BGMCG	Generalized mobilities of hydrocarbon components in gas. See Note 1.
x	BGMCW	Generalized mobilities of water
x	BGMRV	Generalized mobilities of reservoir volume

Table 2.33: Block mobility quantities (ECLIPSE 300)

The inter-region flow quantity options are given in the table below. Notes for this table:

- See [“Format of SUMMARY keywords with a component index”](#).

E100	E300	Region	Information
	x	ROFR	Inter-region oil flow rate
	x	ROFR+	Inter-region oil flow rate (+ve contributions)
	x	ROFR-	Inter-region oil flow rate (-ve contributions)
	x	ROFT	Inter-region oil flow total (in liquid and wet gas phases)
	x	ROFT+	Inter-region oil flow total (in liquid and wet gas phases) (+ve contributions)
	x	ROFT-	Inter-region oil flow total (in liquid and wet gas phases) (-ve contributions)

E100	E300	Region	Information
x		ROFTL	Inter-region oil flow total (liquid phase)
x		ROFTG	Inter-region oil flow total (gas phase)
	x	RGFR	Inter-region gas flow rate
	x	RGFR+	Inter-region gas flow rate (+ve contributions)
	x	RGFR-	Inter-region gas flow rate (-ve contributions)
x	x	RGFT	Inter-region gas flow total (in liquid and gas phases)
	x	RGFT+	Inter-region gas flow total (in liquid and gas phases) (+ve contributions)
	x	RGFT-	Inter-region gas flow total (in liquid and gas phases) (-ve contributions)
		RGFTL	Inter-region gas flow total (liquid phase)
		RGFTG	Inter-region gas flow total (gas phase)
	x	RWFR	Inter-region water flow rate
	x	RWFR+	Inter-region water flow rate (+ve contributions)
	x	RWFR-	Inter-region water flow rate (-ve contributions)
x	x	RWFT	Inter-region water flow total
	x	RWFT+	Inter-region water flow total (+ve contributions)
	x	RWFT-	Inter-region water flow total (-ve contributions)
	x	RKFR	Inter-region component flow rates. See Note 1.
	x	RKFR+	Inter-region component flow rates (+ve contributions). See Note 1.
	x	RKFR-	Inter-region component flow rates (-ve contributions). See Note 1.
	x	RKFT	Inter-region component flow totals. See Note 1.
	x	RKFT+	Inter-region component flow totals (+ve contributions). See Note 1.
	x	RKFT-	Inter-region component flow totals (-ve contributions). See Note 1.
	x	REFR	Inter-region energy flow rates (Thermal option)
	x	REFR+	Inter-region energy flow rates (+ve contributions) (Thermal option)
	x	REFR-	Inter-region energy flow rates, (-ve contributions) (Thermal option)
	x	REFT	Inter-region energy flow totals (Thermal option)
	x	REFT+	Inter-region energy flow totals (+ve contributions) (Thermal option)
	x	REFT-	Inter-region energy flow totals (-ve contributions) (Thermal option)
	x	RNLFR	Inter-region NGL flow rates
	x	RNLFR+	Inter-region NGL flow rates (+ve contributions)
	x	RNLFR-	Inter-region NGL flow rates (-ve contributions)
	x	RNLFT	Inter-region NGL flow totals
	x	RNLFT+	Inter-region NGL flow totals (+ve contributions)
	x	RNLFT-	Inter-region NGL flow totals (-ve contributions)

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 `OPTIONS3` 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*

E100	E300	Field	Region	Block	Information
x	x			BPR	Oil phase Pressure
				BPRESSUR	Oil phase Pressure (same as BPR)
x				BWPR	Water phase Pressure
x				BGPR	Gas phase Pressure
x	x	FPR	RPR		Pressure average value (same as FPRH/RPRH unless, for ECLIPSE 100, item 31 of OPTIONS is set)
x	x	FPRH	RPRH		Pressure average value (Hydrocarbon Pore-Volume Weighted)
x	x	FPRP	RPRP		Pressure average value (Pore-Volume Weighted)
x		FPRGZ	RPRGZ		P/Z (gas) average value weighted by gas content of blocks at surface conditions
x				BRS	Gas-oil ratio (in liquid phase)
x		FRS	RRS		Gas-oil ratio (in liquid phase) average value
x				BRV	Oil-gas ratio (in gas phase)
x		FRV	RRV		Oil-gas ratio (in gas phase) average value
	x	FCHIP			Component Hydrocarbon as Wet Gas. See Note 1.
	x	FCMIP			Component Hydrocarbon as Moles. See Note 1.
x				BPBUB	Bubble point pressure
x				BPDEW	Dew point pressure
x				BRSSAT	Saturated gas-oil ratio (in oil phase)
x				BRVSAT	Saturated oil-gas ratio (in gas phase)
x				BSTATE	Gas-oil state indicator. Hydrocarbon states are: 1 Gas only (with vaporized oil) 2 Gas and Oil 3 Oil only (with dissolved gas)
x		FPPC	RPPC	BPPC	Initial Contact Corrected Potential
x	x			BOKR	Oil relative permeability
x	x			BWKR	Water relative permeability
x	x			BGKR	Gas relative permeability
x	x			BKRO	Oil relative permeability
x	x			BKROG	Two-phase oil relative permeability to gas
x	x			BKROW	Two-phase oil relative permeability to water

E100	E300	Field	Region	Block	Information
x	x			BKRG	Gas relative permeability
	x			BKRGO	Two-phase gas relative permeability to oil (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)
	x			BKRGW	Two-phase gas relative permeability to water (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)
x	x			BKRW	Water relative permeability
	x			BKRWG	Two-phase water relative permeability to gas (BAKER1, BAKER2 and IKU3P three-phase relative permeability models)
	x			BKRWO	Two-phase water relative permeability to oil (BAKER1, BAKER2, and IKU3P three-phase relative permeability models)
x				BRK	Water relative permeability reduction factor due to polymer
x				BEWKR	Water effective relative permeability due to polymer
x	x			BWPC	Water-Oil capillary pressure
x	x			BGPC	Gas-Oil capillary pressure
	x			BPCO	Oil Capillary Pressures
	x			BPCG	Gas Capillary Pressures
	x			BPCW	Water Capillary Pressures
x	x			BGTRP	Trapped gas saturation (WAG hysteresis only)
x	x			BGTPD	Dynamic trapped gas saturation (WAG hysteresis only)
x				BGSHY	Departure saturation from drainage to imbibition for gas capillary pressure hysteresis
x				BGSTRP	Trapped gas critical saturation for gas capillary pressure hysteresis
x				BWSHY	Departure saturation from drainage to imbibition for water capillary pressure hysteresis
x				BWSMA	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.
	x			BMLSC	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.
	x			BMLST	Total hydrocarbon molar density (moles per reservoir volume).

E100	E300	Field	Region	Block	Information
	x			BMWAT	Water molar density (moles per reservoir volume). When the multi-component water option is enabled, this relates to the last component.
	x			BROMLS	Residual oil moles/ reservoir volume ( <a href="#">SOR</a> option). See Note 1.
	x			BJV	In (K) values. See Note 1.
	x			BVMF	Vapor mole fraction.
	x			BPSAT	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.
	x			BAMF	Component aqueous mole fraction. ( <a href="#">KWTABTn</a> , or <a href="#">CO2STORE</a> with <a href="#">THERMAL</a> , or <a href="#">CO2SOL</a> / <a href="#">H2SSOL</a> / <a href="#">GASSOL</a> and with multi-component water <a href="#">COMPW</a> options). See Note 1.
	x			BXMF	Liquid hydrocarbon component mole fraction. See Note 1.
	x			BYMF	Vapor hydrocarbon component mole fraction / vapor steam ( <a href="#">THERMAL</a> option) mole fraction. See Note 1.
	x			BSMF	( <a href="#">CO2STORE</a> with <a href="#">SOLID</a> option only) Solid hydrocarbon component mole fraction. See Note 1.
	x			BSTEN	Surface Tension.
	x			BFMISC	Miscibility Factor.
	x	FREAC		BREAC	Reaction rate. The reaction number is given as a component index. See Note 1.
	x	FREAT			Reaction total. The reaction number is given as a component index. See Note 1.
x				BHD	Hydraulic head.
x				BHDF	Hydraulic head at fresh water conditions.
x				BPR_X	Pressure interpolated at a defined coordinate.
x				BHD_X	Hydraulic head interpolated at a defined coordinate.
x				BHDF_X	Hydraulic head at fresh water conditions interpolated at a defined coordinate.
x				BSCN_X	Brine concentration interpolated at a defined coordinate.
x				BCTRA_X	Tracer concentration interpolated at a defined coordinate. Implies a tracer name TRA and this name needs to start with T
x				LBPR_X	Pressure interpolated at a defined coordinate within a local grid
x				LBHD_X	Hydraulic head interpolated at a defined coordinate within a local grid

E100	E300	Field	Region	Block	Information
x				LBHDF_X	Hydraulic head at freshwater conditions interpolated at a defined coordinate within a local grid
x				LBSCN_X	Brine concentration interpolated at a defined coordinate within a local grid.
x				LBCTRA_X	Tracer concentration interpolated at a defined coordinate within a local grid. Implies a tracer name TRA and this name needs to start with T
	x			BAQSP	(CO2STORE 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.
	x			BAQPH	(CO2STORE option only) Aqueous pH level. See Note 2.
	x			BBTFORG	Forchheimer beta factor for gas phase.
	x			BBTFORO	Forchheimer beta factor for oil phase.
	x			BKFRI	Inter-block component flow rate in the positive I direction. See Note 1.
	x			BKFRJ	Inter-block component flow rate in the positive J direction. See Note 1.
	x			BKFRK	Inter-block component flow rate in the positive K direction. See Note 1.

Table 2.35: Miscellaneous grid block and region quantities

The field, grid block and region quantity options are:

E300	Field	Region	Block	Information
x			BTEMP	Temperature
x			BENERGY	Internal energy/bulk volume
x			BHOIL	Molar Enthalpy of Oil
x			BHGAS	Molar Enthalpy of Gas
x			BHWAT	Molar Enthalpy of Water
x			BSPENOIL	Molar Energy of Oil
x			BSPENGAS	Molar Energy of Gas
x			BSPENWAT	Molar Energy of Water
x			BSPENROC	Specific Internal Energy of Rock
x			BGMEO	Enthalpy mobility of Oil
x			BGMEG	Enthalpy mobility of Gas
x			BGMEW	Enthalpy mobility of Water
x			BTCMULT	Saturation dependent conductivity factor
x			BSFOIL	Saturation of foamy oil. Calculated as the saturation of the components in the oil phase with gas-like compressibility types.

E300	Field	Region	Block	Information
x			BSFSOL	Saturation of foamy solids. Calculated as the saturation of the components in the solid phase with gas-like compressibility types.
x	FERCK	RERCK		Energy in Rock
x	FEOIL	REOIL		Energy in Oil
x	FEGAS	REGAS		Energy in Gas
x	FEWAT	REWAT		Energy in Water
x	FETOT	RETOT		Total Energy
x	FEAVE	REAVE		Average Energy Density
x	FERPC	RERPC		Energy Percentage in Rock
x	FEOPC	REOPC		Energy Percentage in Oil
x	FEGPC	REGPC		Energy Percentage in Gas
x	FEWPC	REWPC		Energy Percentage in Water

Table 2.36: Field, grid block, region quantities (ECLIPSE 300 Thermal)

The rock quantity options are:

E300	Field	Information
x	FHLR	Rate of heat loss to surrounding rocks.
x	FHLT	Cumulative heat loss to surrounding rocks.

Table 2.37: Rock quantities (ECLIPSE 300 Thermal)

The default average field pressure is the hydrocarbon pore volume weighted average defined as:

$$FPRH = \frac{\sum HCPV \cdot p}{\sum HCPV} \quad \text{Eq. 2.1}$$

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:

$$FPRP = \frac{\sum PV \cdot p}{\sum PV} \quad \text{Eq. 2.2}$$

The pore volumes used are those at reference conditions (that is uncompactd 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:

$$\frac{p}{Z} = \frac{\sum \left( PV \cdot \frac{S_g}{B_g} \right) \cdot \frac{T}{T_{sc}} \cdot \frac{P_{sc}}{B_g}}{\sum \left( PV \cdot \frac{S_g}{B_g} \right)} \quad \text{Eq. 2.3}$$

where  $S_g$  = gas saturation

$B_g$  = gas formation volume factor

$T$  = reservoir temperature (keyword [RTEMP](#))

$T_{sc}$  = temperature at standard conditions (288K=520R)

$P_{sc}$  = 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:

E100	Block	Information
x	BOKRX	Oil relative permeability in the X direction.
x	BOKRX-	Oil relative permeability in the -X direction.
x	BOKRY	Oil relative permeability in the Y direction.
x	BOKRY-	Oil relative permeability in the -Y direction.
x	BOKRZ	Oil relative permeability in the Z direction.
x	BOKRZ-	Oil relative permeability in the -Z direction.
x	BWKRX	Water relative permeability in the X direction.
x	BWKRX-	Water relative permeability in the -X direction.
x	BWKRY	Water relative permeability in the Y direction.
x	BWKRY-	Water relative permeability in the -Y direction.
x	BWKRZ	Water relative permeability in the Z direction.
x	BWKRZ-	Water relative permeability in the -Z direction.
x	BGKRX	Gas relative permeability in the X direction.
x	BGKRX-	Gas relative permeability in the -X direction.
x	BGKRY	Gas relative permeability in the Y direction.
x	BGKRY-	Gas relative permeability in the -Y direction.
x	BGKRZ	Gas relative permeability in the Z direction.
x	BGKRZ-	Gas relative permeability in the -Z direction.
x	BOKRI	Oil relative permeability in the I direction.
x	BOKRI-	Oil relative permeability in the -I direction.
x	BOKRJ	Oil relative permeability in the J direction.
x	BOKRJ-	Oil relative permeability in the -J direction.
x	BOKRK	Oil relative permeability in the K direction.

E100	Block	Information
x	BOKRK-	Oil relative permeability in the -K direction.
x	BWKRI	Water relative permeability in the I direction.
x	BWKRI-	Water relative permeability in the -I direction.
x	BWKRJ	Water relative permeability in the J direction.
x	BWKRJ-	Water relative permeability in the -J direction.
x	BWKRK	Water relative permeability in the K direction.
x	BWKRK-	Water relative permeability in the -K direction.
x	BGKRI	Gas relative permeability in the I direction.
x	BGKRI-	Gas relative permeability in the -I direction.
x	BGKRJ	Gas relative permeability in the J direction.
x	BGKRJ-	Gas relative permeability in the -J direction.
x	BGKRK	Gas relative permeability in the K direction.
x	BGKRK-	Gas relative permeability in the -K direction.
x	BOKRR	Oil relative permeability in the R (RADIAL) direction.
x	BOKRR-	Oil relative permeability in the -R (RADIAL) direction.
x	BOKRT	Oil relative permeability in the T (THETA) direction.
x	BOKRT-	Oil relative permeability in the -T (THETA) direction.
x	BWKRR	Water relative permeability in the R (RADIAL) direction.
x	BWKRR-	Water relative permeability in the -R (RADIAL) direction.
x	BWKRT	Water relative permeability in the T (THETA) direction.
x	BWKRT-	Water relative permeability in the -T (THETA) direction.
x	BGKRR	Gas relative permeability in the R (RADIAL) direction.
x	BGKRR-	Gas relative permeability in the -R (RADIAL) direction.
x	BGKRT	Gas relative permeability in the T (THETA) direction.
x	BGKRT-	Gas relative permeability in the -T (THETA) direction.

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:

E300	Block	Information
x	BPSWL	Processed scaled connate water saturation
x	BPSWCR	Processed scaled critical water saturation
x	BPSWU	Processed scaled maximum water saturation
x	BPSGL	Processed scaled connate gas saturation

E300	Block	Information
x	BPSGCR	Processed scaled critical gas saturation
x	BPSGU	Processed scaled maximum gas saturation
x	BPSOWCR	Processed scaled critical oil-in-water saturation
x	BPSOGCR	Processed scaled critical oil-in-gas saturation
x	BPKRW	Processed scaled water relative permeability at maximum water saturation
x	BPKRG	Processed scaled gas relative permeability at maximum gas saturation
x	BPKRO	Processed scaled oil relative permeability at maximum oil saturation
x	BPKRWR	Processed scaled water relative permeability at residual oil saturation
x	BPKRGR	Processed scaled gas relative permeability at residual oil saturation
x	BPKRORG	Processed scaled oil relative permeability at critical gas saturation
x	BPKRORW	Processed scaled oil relative permeability at critical water saturation
x	BPPCG	Processed scaled maximum gas-oil capillary pressure
x	BPPCW	Processed scaled maximum water-oil capillary pressure

Table 2.39: Processed scaled end-points

## Reservoir volumes

The reservoir volume options are:

E100	E300	Field	Region	Block	Information
x	x	FRPV	RRPV	BRPV	Pore Volume at Reservoir conditions
	x			BPORV	Cell Pore Volumes at Reference conditions
x		FOPV	ROPV	BOPV	Pore Volume containing Oil
x		FWPV	RWPV	BWPV	Pore Volume containing Water
x		FGPV	RGPV	BGPV	Pore Volume containing Gas
x	x	FHPV	RHPV		Pore Volume containing Hydrocarbon (or “Hydrocarbon Pore Volume”)
x				BHPV	Pore Volume containing Hydrocarbon (as above)
x		FRTM	RRTM	BRTM	Transmissibility Multiplier associated with rock compaction
x	x			BPERMMOD	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.
x	x			BPERMMDX BPERMMDY BPERMMDZ	Directional Transmissibility Multipliers in the x, y and z directions respectively, associated with rock compaction. These keywords are available when <a href="#">RKTRMDIR</a> is defined in both ECLIPSE 100 and ECLIPSE 300, or when <a href="#">ROCKTRMX</a> , <a href="#">ROCKTRMY</a> and <a href="#">ROCKTRMZ</a> are defined in ECLIPSE 300.
x	x			BPORVMOD	Pore Volume Multiplier associated with rock compaction

E100	E300	Field	Region	Block	Information
x	x			BSIGMMOD	Dual Porosity Sigma Multiplier associated with rock compaction
	x			BPVDP	Pressure Derivatives of Pore Volumes
	x			BSVOL	Specific Volume of Fluids

Table 2.40: Reservoir volumes

## Oil recovery efficiencies

The oil recovery options are:

E100	E300	Field	Region	Information
x	x	FOE	ROE	$(OIP(\text{initial}) - OIP(\text{now})) / OIP(\text{initial})$
x	x	FOEW	ROEW	Oil Production from Wells / OIP(initial)
x	x	FOEIW	ROEIW	$(OIP(\text{initial}) - OIP(\text{now})) / \text{Initial Mobile Oil with respect to Water}$
x	x	FOEWW	ROEWW	Oil Production from Wells / Initial Mobile Oil with respect to Water
x	x	FOEIG	ROEIG	$(OIP(\text{initial}) - OIP(\text{now})) / \text{Initial Mobile Oil with respect to Gas}$
x	x	FOEWG	ROEWG	Oil Production from Wells / Initial Mobile Oil with respect to Gas

Table 2.41: Oil recovery efficiencies

## Oil recovery mechanism

The oil recovery mechanism options are:

E100	Field	Region	Information
x	FORMR	RORMR	Total stock tank oil produced by rock compaction
x	FORMW	RORMW	Total stock tank oil produced by water influx
x	FORMG	RORMG	Total stock tank oil produced by gas influx
x	FORME	RORME	Total stock tank oil produced by oil expansion
x	FORMS	RORMS	Total stock tank oil produced by solution gas
x	FORMF	RORMF	Total stock tank oil produced by free gas influx
x	FORMX	RORMX	Total stock tank oil produced by 'traced' water influx
x	FORMY	RORMY	Total stock tank oil produced by other water influx
x	FORFR	RORFR	Fraction of total oil produced by rock compaction
x	FORFW	RORFW	Fraction of total oil produced by water influx
x	FORFG	RORFG	Fraction of total oil produced by gas influx
x	FORFE	RORFE	Fraction of total oil produced by oil expansion
x	FORFS	RORFS	Fraction of total oil produced by solution gas
x	FORFF	RORFF	Fraction of total oil produced by free gas influx
x	FORFX	RORFX	Fraction of total oil produced by 'traced' water influx

E100	Field	Region	Information
x	FORFY	RORFY	Fraction of total oil produced by other water influx

Table 2.42: Oil recovery mechanism (ECLIPSE 100)

ECLIPSE 100

**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".

E300	Field	Information
x	FNRR	NGL recovery volume rate
x	FNRT	NGL recovery volume total
x	FNRM	NGL recovery molar rate
x	FGRR	Gas remaining volume rate
x	FGRT	Gas remaining volume total
x	FGRM	Gas remaining molar rate
x	FXNR	NGL recovery composition. See Note 1.
x	FYGR	Gas remaining composition. See Note 1.

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".

E100	E300	Field	Aquifer	Aquifer List	Information
x	x	FAQR	AAQR		Aquifer influx rate (for liquid aquifer)
x				ALQR	Aquifer influx rate (for liquid aquifer) (as above)
x	x	FAQT	AAQT		Cumulative aquifer influx (for liquid aquifer)
x				ALQT	Cumulative aquifer influx (for liquid aquifer) (as above)
x	x	FAQRG	AAQRG		Aquifer influx rate (for gas aquifer)
x				ALQRG	Aquifer influx rate (for gas aquifer) (as above)
x	x	FAQTG	AAQTG		Cumulative aquifer influx (for gas aquifer)
x				ALQTG	Cumulative aquifer influx (for gas aquifer) (as above)
	x		AACMR		Aquifer component molar influx rate (for multi-component aquifer). See Note 1.

E100	E300	Field	Aquifer	Aquifer List	Information
	x		AACMT		Aquifer component molar influx totals (for multi-component aquifer). See Note 1.
x	x		AAQP		Aquifer pressure (excludes constant flux aquifer as given by <a href="#">AQUFLUX</a> ).
	x	FAQER	AAQER		Aquifer thermal energy influx rate (Thermal option)
	x	FAQET	AAQET		Cumulative aquifer thermal energy influx (Thermal option)
	x		AAQTEMP		Aquifer temperature
	x		AAQENTH		Aquifer molar enthalpy (Thermal option)
x	x		AAQTD		Aquifer dimensionless time (Carter-Tracy aquifers only)
x	x		AAQPD		Aquifer dimensionless pressure (Carter-Tracy aquifers only)

Table 2.44: Analytic aquifer quantities

## 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](#)”.

E100	E300	Field	Aquifer	Information
x	x	FNQR	ANQR	Aquifer influx rate
x	x	FNQT	ANQT	Cumulative aquifer influx
x	x		ANQP	Aquifer pressure (average pressure weighted by water volume)
	x		ANCMR	Aquifer component molar influx rate. See Note 1.
	x		ANCMT	Aquifer component molar influx totals. See Note 1.

Table 2.45: Numerical aquifer quantities

## Tracer and API tracking quantities

The tracer and API tracking options are:

E100	E300	Field	Group	Well	Connection	Region	Block	Information
x	x				CTFR			Tracer Flow Rate (+ or -)
x	x	FTPR	GTPR	WTPR	CTPR			Tracer Production Rate
x	x	FTPT	GTPT	WTPT	CTPT			Tracer Production Total
x	x	FTPC	GTPC	WTPC	CTPC			Tracer Production Concentration
x	x	FTIR	GTIR	WTIR	CTIR			Tracer Injection Rate
x	x	FTIT	GTIT	WTIT	CTIT			Tracer Injection Total
x	x	FTIC	GTIC	WTIC	CTIC			Tracer Injection Concentration

E100	E300	Field	Group	Well	Connection	Region	Block	Information
	x	FTMR	GTMR	WTMR				Traced mass Rate
	x	FTMT	GTMT	WTMT				Traced mass Total
	x	FTQR	GTQR	WTQR				Traced molar Rate
	x	FTCM	GTCM	WTCM				Tracer Carrier molar Rate
	x	FTMF	GTMF	WTMF				Traced molar fraction
	x	FTVL	GTVL	WTVL				Traced liquid volume rate
	x	FTVV	GTVV	WTVV				Traced vapor volume rate
	x	FTTL	GTTL	WTTL				Traced liquid volume total
	x	FTTV	GTTV	WTTV				Traced vapor volume total
	x	FTML	GTML	WTML				Traced mass liquid rate
	x	FTMV	GTMV	WTMV				Traced mass vapor rate
	x	FTLM	GTLM	WTLM				Traced mass liquid total
	x	FTVM	GTVM	WTVM				Traced mass vapor total
x							BTCNF	Tracer Concentration (Free)
x							BTCNS	Tracer Concentration (Solution)
	x						BTCN	Tracer Concentration
x		FTIPT				RTIPT	BTIPT	Tracer In Place (Total)
x		FTIPF				RTIPF	BTIPF	Tracer In Place (Free)
x		FTIPS				RTIPS	BTIPS	Tracer In Place (Solution)
x						RTFTF		Tracer inter-region Flow Total (Free)
x						RTFTS		Tracer inter-region Flow Total (Solution)
x						RTFTT		Tracer inter-region Flow Total (Total)
x		FAPI	GAPI	WAPI	CAPI	RAPI	BAPI	Oil API (for API tracking). (F,G,W,C)API are for production wells

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:

E100	E300	Field	Group	Well	Connection	Region	Block	Information
x	x				CSFR			Salt Flow Rate (+ or -)

E100	E300	Field	Group	Well	Connection	Region	Block	Information
x	x	FSPR	GSPR	WSPR	CSPR			Salt Production Rate
x	x	FSPT	GSPT	WSPT	CSPT			Salt Production Total
x	x	FSIR	GSIR	WSIR	CSIR			Salt Injection Rate
x	x	FSIT	GSIT	WSIT	CSIT			Salt Injection Total
x	x						BSCN	Salt Cell Concentration
x	x	FSPC	GSPC	WSPC	CSPC			Salt Production Concentration
x	x	FSIC	GSIC	WSIC	CSIC			Salt Injection Concentration
x	x	FSIP						Salt In Place
x						RSIP	BSIP	Salt In Place (as above)
x						RSFT		Salt inter-region Flow Total
x							BEWV_SAL	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.

Table 2.47: Brine option quantities

The outputs for the Multi-Component Brine are:

E100	Field	Group	Well	Connection	Block	Information
x				CTFRANI		Anion Flow Rate
x	FTPRANI	GTPRANI	WTPRANI			Anion Production Rate
x	FTPTANI	GTPTANI	WTPTANI	CTPTANI		Anion Production Total
x	FTIRANI	GTIRANI	WTIRANI			Anion Injection Rate
x	FTITANI	GTITANI	WTITANI	CTITANI		Anion Injection Total
x					BTCNFANI	Anion Flowing Concentration
x				CTFRCAT		Cation Flow Rate
x	FTPRCAT	GTPRCAT	WTPRCAT			Cation Production Rate
x	FTPTCAT	GTPTCAT	WTPTCAT	CTPTCAT		Cation Production Total
x	FTIRCAT	GTIRCAT	WTIRCAT			Cation Injection Rate
x	FTITCAT	GTITCAT	WTITCAT	CTITCAT		Cation Injection Total
x					BTCNFCAT	Cation Flowing Concentration
x					BTRADCAT	Cation Rock Associated Concentration
x					BTSADCAT	Cation Surfactant Associated Concentration.
x					BESALSUR	Effective Salinity with respect to Surfactant

E100	Field	Group	Well	Connection	Block	Information
x					BESALPLY	Effective Salinity art Polymer

Table 2.48: Outputs for Multi-Component Brine (ECLIPSE 100)

## 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'.

E100	Field	Group	Well	Region	Block	Description
x	FTPCH EA	GTPCH EA	WTPCH EA			Production Temperature
x	FTICHEA	GTICHEA	WTICHEA			Injection Temperature
x	FTPRHEA	GTPRHEA	WTPRHEA			Energy flows (Production)
x	FTPTHEA	GTPTHEA	WPTTHEA			Energy Production Total
x	FTIRHEA	GTIRHEA	WTIRHEA			Energy flows (Injection)
x	FTITHEA	GTITHEA	WTITHEA			Energy Injection Total
x					BTCNFHEA	Block Temperature
x	FTIPTHEA			RTIPTHEA	BTIPTHEA	Difference in Energy in place between current and initial time

Table 2.49: Summary output for Temperature option (ECLIPSE 100)

## Environmental tracers

The following keywords control output of data specific to tracers:

E100	E300	Field	Group	Well	Connection	Region	Block	Information
x	x				CTFR			Tracer Flow Rate (+ or -)
x	x	FTPR	GTPR	WTPR	CTPR			Tracer Production Rate
x	x	FTPT	GTPT	WTPT	CTPT			Tracer Production Total
x	x	FTPC	GTPC	WTPC	CTPC			Tracer Production Concentration
x	x	FTIR	GTIR	WTIR	CTIR			Tracer Injection Rate
x	x	FTIT	GTIT	WTIT	CTIT			Tracer Injection Total
x	x	FTIC	GTIC	WTIC	CTIC			Tracer Injection Concentration
x							BTCNF	Tracer Concentration (Free)
x							BTCNS	Tracer Concentration (Solution)
x							BTCN#	Tracer concentration in phase # (1,2,3,...)
x		FTIPT				RTIPT	BTIPT	Tracer In Place (Total)
x		FTIPF				RTIPF	BTIPF	Tracer In Place (Free)

E100	E300	Field	Group	Well	Connection	Region	Block	Information
x		FTIPS				RTIPS	BTIPS	Tracer In Place (Solution)
x		FTIP#				RTIP#	BTIP#	Tracer In Place in phase # (1,2,3,...)
x						RTFTT		Tracer inter-region Flow (Total)
x						RTFTF		Tracer inter-region Flow (Free)
x						RTFTS		Tracer inter-region Flow (Solution)
x						RTFT#		Tracer inter-region Flow in phase # (1,2,3,...)
x		FTADS				RTADS		Tracer Adsorption total
x							BTADS	Tracer Adsorption
x		FTDCY				RTDCY	BTDCY	Decayed tracer
x		FTIRF	GTIRF	WTIRF	CTIRF			Tracer Injection Rate (Free)
x		FTIRS	GTIRS	WTIRS	CTIRS			Tracer Injection Rate (Solution)
x		FTPRF	GTPRF	WTPRF	CTPRF			Tracer Production Rate (Free)
x		FTPRS	GTPRS	WTPRS	CTPRS			Tracer Production Rate (Solution)
x		FTITF	GTITF	WTITF	CTITF			Tracer Injection Total (Free)
x		FTITS	GTITS	WTITS	CTITS			Tracer Injection Total (Solution)
x		FTPTF	GTPTF	WTPTF	CTPTF			Tracer Production Total (Free)
x		FTPTS	GTPTS	WTPTS	CTPTS			Tracer Production Total (Solution)
x		FTICF	GTICF	WTICF	CTICF			Tracer Injection Concentration (Free)
x		FTICS	GTICS	WTICS	CTICS			Tracer Injection Concentration (Solution)
x		FTPCF	GTPCF	WTPCF	CTPCF			Tracer Production Concentration (Free)
x		FTPCS	GTPCS	WTPCS	CTPCS			Tracer Production Concentration (Solution)

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 adsorped 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 adsorped 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:

E100	Field	Group	Well	Region	Block	Information
x	FMPR	GMPR	WMPR			Methane Production Rate
x	FMPT	GMPT	WMPT			Methane Production Total
x	FMIR	GMIR	WMIR			Methane Injection Rate
x	FMIT	GMIT	WMIT			Methane Injection Total
x	FCGC			RCGC	BCGC	Bulk Coal Gas Concentration (concentration in the coal matrix)
x	FCSC			RCSC	BCSC	Bulk Coal Solvent Concentration (concentration in the coal matrix)

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.

E100	E300	Field	Group	Well	Connection	Information
x					CTFRFOA	Flow Rate (+ or -)
x		FTPRFOA	GTPRFOA	WTPRFOA		Production Rate
x		FTPTFOA	GTPTFOA	WTPTFOA	CTPTFOA	Production Total
x		FTIRFOA	GTIRFOA	WTIRFOA		Injection Rate
x		FTITFOA	GTITFOA	WTITFOA	CTITFOA	Injection Total
x		FTIPTFOA				In Solution
x		FTADSFOA				Adsorption total
x		FTDCYFOA				Decayed tracer

E100	E300	Field	Group	Well	Connection	Information
x		FTMOBFOA				Gas mobility factor (excluding shear)

Table 2.52: Foam SUMMARY field, group, well and connection output keywords

E100	E300	Field	Region	Block	Information
x		FTPRFOA			Production Rate
x		FTPTFOA			Production Total
x		FTIRFOA			Injection Rate
x		FTITFOA			Injection Total
x				BTCNFFOA	Concentration
	x			BFOAM	Surfactant concentration
x				BTCNMFOA	Capillary number
	x			BFOAMCNM	Capillary number
x		FTIPTFOA	RTIPTFOA	BTIPTFOA	In Solution
x			RTFTTFOA		Inter-region Flow Total
x		FTADSFOA	RTADSFOA		Adsorption total
x				BTADSFOA	Adsorption
x		FTDCYFOA	RTDCYFOA	BTDCYFOA	Decayed tracer
x		FTMOBFOA	RTMOBFOA	BTMOBFOA	Gas mobility factor (excluding shear)
	x			BFOAMMOB	Gas mobility factor
x				BTHLFFOA	Decay Half life
	x			BFOAMST	Gas-water surface tension

Table 2.53: Foam SUMMARY, field, region and block output keywords

## Gas Field Operations Model

The following items are relevant to the Gas Field Operations Model:

E100	E300	Field	Group	Well	Information
x	x	FSGR	GSGR		Sales Gas Rate
x	x	FGSR	GGSR		Sales Gas Rate (as above)
x	x	FSGT	GSGT		Sales Gas Total
x	x	FGST	GGST		Sales Gas Total (as above)
x	x	FGDC	GGDC	WGDC	Gas Delivery Capacity
x	x	FGDCQ	GGDCQ		Field/Group Gas DCQ
x			GMCPL		Group Multi-level Compressor Level
x	x		GPR		Group nodal PPressure in network When using the Extended Network Model, the names must correspond to network nodes instead of groups

E100	E300	Field	Group	Well	Information
x			GPRDC		Group PPressure at Delivery Capacity When using the Extended Network Model, the names must correspond to network nodes instead of groups
x		FGCR	GGCR		Gas consumption rate, at and below this group
x		FGCT	GGCT		Gas consumption cumulative total, at and below this group
	x	FFGR	GFGR		Fuel Gas rate, at and below this group
	x	FFGT	GFGT		Fuel Gas cumulative total, at and below this group
x		FGIMR	GGIMR		Gas import rate, at and below this group
x		FGIMT	GGIMT		Gas import cumulative total, at and below this group
x	x			NGOPAS	Number of iterations to converge DCQ in first pass
x				WGPRFP	Well Gas Production Rate from end of First Pass
x				WTHPFP	Well Tubing Head Pressure from end of First Pass
x				WBHPFP	Well Bottom Hole Pressure from end of First Pass
x			GPRFP		Group or node PPressure in network from end of First Pass
x			GGPRNBFP		Gas flow rate along Group's or node's outlet branch in network, from end of First Pass

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, WTHPFP, 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:

E100	E300	Field	Group	Well	Information
x	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			WOGLR	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:

E100	E300	Field	Group	Well	Information
x		FGCV	GGCV	WGCV	Gas Calorific Value
x	x	FGQ	GGQ	WGQ	Gas molar Quality
x	x	FEPR	GEPR	WEPR	Energy Production Rate
x	x	FEPT	GEPT	WEPT	Energy Production Total (cumulative)
x		FESR	GESR		Energy Sales Rate
x		FEST	GEST		Energy Sales Total (cumulative)
x		FEDC	GEDC	WEDC	Energy Delivery Capacity
x		FEDCQ	GEDCQ		Energy DCQ

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:

E100	Block	Information
x	BGI	Block Gi value

Table 2.57: Gi pseudo-compositional SUMMARY output controls

## Local grid refinement and coarsening

ECLIPSE 100

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

ECLIPSE 100 only

NOSUMLGR Prevents the creation of local timestep summary files.

### Plotting local timestep results in GRAF

ECLIPSE 100 only

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](#)”.

E100	E300	Segment	Information
x	x	SOFR	Segment Oil Flow Rate
x		SOFRF	Segment Free Oil Flow Rate
x		SOFRS	Segment Solution Oil Flow Rate (vaporized oil)
x	x	SWFR	Segment Water Flow Rate
x	x	SGFR	Segment Gas Flow Rate
x		SGFRF	Segment Free Gas Flow Rate
x		SGFRS	Segment Solution Gas Flow Rate
	x	SKFR	Segment Component Flow Rate. See Note 1.
	x	SCWGFR	Segment Component Flow Rate as Wet Gas. See Note 1.
	x	SHFR	Segment Enthalpy Flow Rate (Thermal option)
x	x	SWCT	Segment Water Cut
x	x	SGOR	Segment Gas Oil Ratio
x	x	SOGR	Segment Oil Gas Ratio
x	x	SWGR	Segment Water Gas Ratio
x	x	SPR	Segment PPressure
x	x	SPRD	Segment PPressure Drop
x	x	SPRDF	Segment PPressure Drop component due to Friction
x	x	SPRDH	Segment PPressure Drop component due to Hydrostatic head
x	x	SPRDA	Segment PPressure drop due to Acceleration head
x	x	SPRDM	Segment frictional PPressure Drop Multiplier (keyword <a href="#">WSEGMULT</a> )
x		SPPOW	Working power of a pull through pump (keyword <a href="#">WSEGPULL</a> )
x	x	SOFV	Segment Oil Flow Velocity
x	x	SWFV	Segment Water Flow Velocity

E100	E300	Segment	Information
x	x	SGFV	Segment Gas Flow Velocity
x	x	SOHF	Segment Oil Holdup Fraction
x	x	SWHF	Segment Water Holdup Fraction
x	x	SGHF	Segment Gas Holdup Fraction
x	x	SDENM	Segment fluid mixture density
x		SOVIS	Segment oil viscosity
x		SWVIS	Segment water viscosity (pure water)
x		SGVIS	Segment gas viscosity
x		SEMVIS	Segment effective mixture viscosity (water/polymer)
x	x	SGLPP	Segment Gas-Liquid Profile Parameter, C0 (drift flux slip model)
x	x	SGLVD	Segment Gas-Liquid Drift Velocity, Vd (drift flux slip model)
x	x	SOWPP	Segment Oil-Water Profile Parameter, C0 (drift flux slip model)
x	x	SOWVD	Segment Oil-Water Drift Velocity, Vd (drift flux slip model)
	x	SOIMR	Segment Oil Import Rate (keyword <a href="#">WSEGEXSS</a> )
x	x	SGIMR	Segment Gas Import Rate (keyword <a href="#">WSEGEXSS</a> )
x	x	SWIMR	Segment Water Import Rate (keyword <a href="#">WSEGEXSS</a> )
	x	SHIMR	Segment Enthalpy Import Rate (keyword <a href="#">WSEGEXSS</a> ) (Thermal option)
	x	SORMR	Segment Oil Removal Rate (keyword <a href="#">WSEGEXSS</a> )
	x	SGRMR	Segment Gas Removal Rate (keyword <a href="#">WSEGEXSS</a> )
	x	SWRMR	Segment Water Removal Rate (keyword <a href="#">WSEGEXSS</a> )
	x	SHRMR	Segment Enthalpy Removal Rate (keyword <a href="#">WSEGEXSS</a> ) (Thermal option)
	x	SOIMT	Segment Oil Import Total (keyword <a href="#">WSEGEXSS</a> )
x	x	SGIMT	Segment Gas Import Total (keyword <a href="#">WSEGEXSS</a> )
x	x	SWIMT	Segment Water Import Total (keyword <a href="#">WSEGEXSS</a> )
	x	SHIMT	Segment Enthalpy Import Total (keyword <a href="#">WSEGEXSS</a> ) (Thermal option)
	x	SORMT	Segment Oil Removal Total (keyword <a href="#">WSEGEXSS</a> )
	x	SGRMT	Segment Gas Removal Total (keyword <a href="#">WSEGEXSS</a> )
	x	SWRMT	Segment Water Removal Total (keyword <a href="#">WSEGEXSS</a> )
	x	SHRMT	Segment Enthalpy Removal Total (keyword <a href="#">WSEGEXSS</a> ) (Thermal option)
x		SAPI	Segment API value
x		SCFR	Segment polymer flow rate (Polymer flood option)
x		SCCN	Segment polymer concentration (Polymer flood option)
x		SSFR	Segment brine flow rate (Surfactant and Polymer flood option)
x		SSCN	Segment brine concentration (Surfactant and Polymer flood option)
x		STFR	Segment tracer flow rate

E100	E300	Segment	Information
x		STFC	Segment tracer concentration
x		SFD	Segment diameter for Karst Conduit Calcite Dissolution
	x	SPSAT	Segment $P_{sat}$ . The value depends on the segment 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 segments it will be the segment pressure.
	x	STEM	Segment Temperature
	x	SENE	Segment Energy Density (Thermal option)
	x	SSQU	Segment Steam Quality (Thermal option)
	x	SCVPR	Segment Calorific Value Production Rate
	x	SGQ	Segment Gas Quality
	x	SCSA	Segment Cross Sectional Area. This is specified using the <a href="#">WSEGVALV</a> keyword, and can change if the cross sectional area is used as a reservoir optimization parameter using the <a href="#">OPTPARS</a> keyword.
x	x	SSTR	Strength of ICD on segment (see keywords <a href="#">WSEGVALV</a> , <a href="#">WSEGAICD</a> and <a href="#">WSEGSICD</a> )
x	x	SFOPN	Setting of segment (see keywords <a href="#">WSEGVALV</a> , <a href="#">WSEGAICD</a> , <a href="#">WSEGSICD</a> , <a href="#">WSEGTABL</a> , <a href="#">WSEGLABY</a> and <a href="#">WSEGFLIM</a> )

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.

ECLIPSE 100

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

E100	Segment	Information
x	SRRQR	Reach flow at current time
x	SRRQT	Reach cumulative flow
x	SRBQR	Branch flow at current time
x	SRBQT	Branch cumulative flow

E100	Segment	Information
x	SRTQR	River total flow at current time
x	SRTQT	River total cumulative flow
x	SRRFLOW	Reach flux through cross-sectional area at current time
x	SRRAREA	Reach area at current time
x	SRRDEPTH	Reach depth at current time
x	SRREXCH	Exchange flux at current time
x	SRRFRODE	Reach Froude number at current time
x	SRRHEAD	Reach hydraulic head at current time
x	SRTFR	Reach tracer flow rate
x	SRTFC	Reach tracer concentration
x	SRSFR	Reach brine flow rate through connections
x	SRSFC	Reach brine concentration

Table 2.59: River SUMMARY output controls

The river reach controls are:

E100	Connection	Information
x	CRREXCH	Exchange flux at current time
x	CRRPROT	Connection cumulative water production
x	CRRINJT	Connection cumulative water injection

Table 2.60: River reach SUMMARY controls

For example,

```

SRRFLOW
-- 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 /

SRTFRFFF
-- 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 /
    
```

MYRIVER 12 /  
/

## 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.

E100	E300	Group	Information
x	x	GPR	Group or node PResure in the production network
x	x	GPRG	Group or node PResure in the gas injection network
x	x	GPRW	Group or node PResure in the water injection network
x	x	GPRB	Pressure drop along the group's or node's outlet branch in the production network
x	x	GPRBG	Pressure drop along the group's or node's inlet branch in the gas injection network
x	x	GPRBW	Pressure drop along the group's or node's inlet branch in the water injection network
x	x	GALQ	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.
x	x	GOPRNB	Oil flow rate along the group's or node's outlet branch in the production network
x	x	GWPRNB	Water flow rate along the group's or node's outlet branch in the production network
x	x	GGPRNB	Gas flow rate along the group's or node's outlet branch in the production network
	x	GLPRNB	Liquid flow rate along the group's or node's outlet branch in the production network
x	x	GWIRNB	Water flow rate along the group's or node's inlet branch in the water injection network
x	x	GGIRNB	Gas flow rate along the group's or node's inlet branch in the gas injection network
x		GOMNR	Group or node minimum oil rate as specified with <b>GNETDP</b> in the production network
x		GGMNR	Group or node minimum gas rate as specified with <b>GNETDP</b> in the production network
x		GWMNR	Group or node minimum water rate as specified with <b>GNETDP</b> in the production network
x		GLMNR	Group or node minimum liquid rate as specified with <b>GNETDP</b> in the production network
x		GOMXR	Group or node maximum oil rate as specified with <b>GNETDP</b> in the production network

E100	E300	Group	Information
x		GGMXR	Group or node maximum gas rate as specified with GNETDP in the production network
x		GWMXR	Group or node maximum water rate as specified with GNETDP in the production network
x		GLMXR	Group or node maximum liquid rate as specified with GNETDP in the production network
x		GMNP	Group or node minimum pressure as specified with GNETDP in the production network
x		GMXP	Group or node maximum pressure as specified with GNETDP in the production network
x		GPRINC	Group or node pressure increment as specified with GNETDP in the production network
x		GPRDEC	Group or node pressure decrement as specified with GNETDP in the production network

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:

E100	Field	Group	Well	Connection	Region	Block	Information
x				CCFR			Polymer Flow Rate (+ or -)
x	FCPR	GCPR	WCPR	CCPR			Polymer Production Rate
x	FCPC	GCPC	WCPC	CCPC			Polymer Production Concentration
x	FCPT	GCPT	WCPT	CCPT			Polymer Production Total
x	FCIR	GCIR	WCIR	CCIR			Polymer Injection Rate
x	FCIC	GCIC	WCIC	CCIC			Polymer Injection Concentration
x	FCIT	GCIT	WCIT	CCIT			Polymer Injection Total
x						BCCN	Polymer Concentration
x	FCIP				RCIP	BCIP	Polymer In Solution
x					RCFT		Polymer inter-region Flow Total
x						BEPVIS	Effective polymer solution viscosity
x						BVPOLY	Effective polymer solution viscosity (as above)
x						BEMVIS	Effective mixture (water/polymer) viscosity
x						BEWV_POL	Effective water viscosity
x	FCAD				RCAD		Polymer Adsorption total
x						BCAD	Polymer Adsorption concentration

E100	Field	Group	Well	Connection	Region	Block	Information
x						BCDCS	Polymer thermal degradation - total mass degraded in previous timestep
x						BCDCR	Polymer thermal degradation - total degradation rate
x						BCDCP	Polymer thermal degradation - solution degradation rate
x						BCDCA	Polymer thermal degradation - adsorbed degradation rate
x						BCABnnn	Adsorbed polymer by highest temperature band at which RRF was calculated - see keyword <a href="#">PLYTRRFA</a> . The band number nnn can range from 001 to 999, but must be less than or equal to the argument of <a href="#">PLYTRRFA</a>
x				CSFR			Salt Flow Rate (+ or -)
x	FSPR	GSPR	WSPR	CSPR			Salt Production Rate
x	FSPT	GSPT	WSPT	CSPT			Salt Production Total
x	FSIR	GSIR	WSIR	CSIR			Salt Injection Rate
x	FSIT	GSIT	WSIT	CSIT			Salt Injection Total
x						BSCN	Salt Cell Concentration
x	FSIP						Salt In Place
x					RSIP	BSIP	Salt In Place (as above)
x					RSFT		Salt inter-region Flow Total
x						BFLOW0I	Inter-block water flow rate in the positive I direction multiplied by the corresponding shear multiplier ( <a href="#">PLYSHLOG</a> and <a href="#">PLYSHEAR</a> options only)
x						BFLOW0J	Inter-block water flow rate in the positive J direction multiplied by the corresponding shear multiplier ( <a href="#">PLYSHLOG</a> and <a href="#">PLYSHEAR</a> options only)
x						BFLOW0K	Inter-block water flow rate in the positive K direction multiplied by the corresponding shear multiplier ( <a href="#">PLYSHLOG</a> and <a href="#">PLYSHEAR</a> options only)

E100	Field	Group	Well	Connection	Region	Block	Information
x						BVELW0I	Water velocity in the positive I direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)
x						BVELW0J	Water velocity in the positive J direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)
x						BVELW0K	Water velocity in the positive K direction multiplied by the corresponding shear multiplier (PLYSHLOG and PLYSHEAR options only)
x						BPSHLZI	Viscosity multiplier due to sheared water flow in the positive I direction (PLYSHLOG and PLYSHEAR options only)
x						BPSHLZJ	Viscosity multiplier due to sheared water flow in the positive J direction (PLYSHLOG and PLYSHEAR options only)
x						BPSHLZK	Viscosity multiplier due to sheared water flow in the positive K direction (PLYSHLOG and PLYSHEAR options only)
x						BSRTW0I	Water shear rate in the positive I direction prior to shear effects (PLYSHLOG option with keyword SHRATE)
x						BSRTW0J	Water shear rate in the positive J direction prior to shear effects (PLYSHLOG option with keyword SHRATE)
x						BSRTW0K	Water shear rate in the positive K direction prior to shear effects (PLYSHLOG option with keyword SHRATE)
x						BSRTWI	Water shear rate in the positive I direction following shear effects (PLYSHLOG option with keyword SHRATE)

E100	Field	Group	Well	Connection	Region	Block	Information
x						BSRTWJ	Water shear rate in the positive J direction following shear effects (PLYSHLOG option with keyword SHRATE)
x						BSRTWK	Water shear rate in the positive K direction following shear effects (PLYSHLOG option with keyword SHRATE)
x						BSHWVISI	Shear viscosity of the water/polymer solution due to shear thinning/thickening in the positive I direction (PLYSHLOG and PLYSHEAR options only)
x						BSHWVISJ	Shear viscosity of the water/polymer solution due to shear thinning/thickening in the positive J direction (PLYSHLOG and PLYSHEAR options only)
x						BSHWVISK	Shear viscosity of the water/polymer solution due to shear thinning/thickening in the positive K direction (PLYSHLOG and PLYSHEAR options only)

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:

E300	Block	Information
x	BESALTP	Effective salinity
x	BPOLY	Polymer concentration
x	BPOLYVM	Polymer viscosity multiplier

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.

E100	Field	Description
x	PSSPR	Log of the pressure change per unit time, see <a href="#">PSTEADY</a>
x	PSSSO	Log of the oil saturation change per unit time, see <a href="#">PSTEADY</a>
x	PSSSW	Log of the water saturation change per unit time, see <a href="#">PSTEADY</a>
x	PSSSG	Log of the gas saturation change per unit time, see <a href="#">PSTEADY</a>

E100	Field	Description
x	PSSSC	Log of the salt concentration change per unit time, see PSTEADY

Table 2.64: Steady state SUMMARY output controls

## Reservoir coupling facility (ECLIPSE 100 only)

The following keywords control output of data relevant to the Reservoir Coupling facility:

E100	Group	Description
x	GOPRL	Group Oil Production Rate Target
x	GOIRL	Group Oil Injection Rate Target
x	GWPRL	Group Water Production Rate Target
x	GWIRL	Group Water Injection Rate Target
x	GGPRL	Group Gas Production Rate Target
x	GGIRL	Group Gas Injection Rate Target
x	GLPRL	Group Liquid Production Rate Target
x	GVPRL	Group reservoir Volume Production Rate Target
x	GVIRL	Group reservoir Volume Injection Rate Target

Table 2.65: Reservoir coupling SUMMARY output controls (ECLIPSE 100 only)

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

ECLIPSE 100 only

The following keywords control output of data specific to the Solvent Model.

E100	Field	Group	Well	Connection	Region	Block	Information
x				CNFR			Solvent Flow Rate (+ or -)
x	FNPR	GNPR	WNPR				Solvent Production Rate
x	FNPT	GNPT	WNPT	CNPT			Solvent Production Total
x	FNIR	GNIR	WNIR				Solvent Injection Rate
x	FNIT	GNIT	WNIT	CNIT			Solvent Injection Total
x						BNSAT	Solvent SATuration
x	FNIP				RNIP	BNIP	Solvent In Place

E100	Field	Group	Well	Connection	Region	Block	Information
x					RNFT		Solvent inter-region Flow
x						BNKR	Solvent relative permeability

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.

E100	E300	Field	Group	Well	Connection	Information
x					CTFRSUR	Flow Rate (+ or -)
x		FTPRSUR	GTPRSUR	WTPRSUR		Production Rate
x		FTPTSUR	GTPTSUR	WTPTSUR	CTPTSUR	Production Total
x		FTIRSUR	GTIRSUR	WTIRSUR		Injection Rate
x		FTITSUR	GTITSUR	WTITSUR	CTITSUR	Injection Total

Table 2.67: Surfactant SUMMARY field, group, well and connection output controls

E100	E300	Field	Region	Block	Information
x				BTCNFSUR	Concentration
	x			BSURF	Concentration in solution
x		FTIPTSUR	RTIPTSUR	BTIPTSUR	In Solution
x			RTFTTSUR		Inter-region Flow Total
x		FTADSUR	RTADSUR		Adsorption total
x				BTADSUR	Adsorption
x				BTCASUR	Log (Capillary Number)
	x			BSURFCNM	Log (Capillary Number)
x				BTSTSUR	Surface tension
	x			BSURFST	Surface tension
x				BEWV_SUR	Effective water viscosity due to surfactant concentration
	x			BSURFKR	Relative permeability interpolation factor
	x			BSURFCP	Capillary pressure multiplier
	x			BESALTS	Effective salinity

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.

E100	Field	Group	Well	Connection	Block	Information
x				CTFRALK		Flow Rate (+ or -)
x	FTPRALK	GTPRALK	WTPRALK			Production Rate
x	FTPTALK	GTPTALK	WTPTALK	CTPTALK		Production Total
x	FTIRALK	GTIRALK	WTIRALK			Injection Rate
x	FTITALK	GTITALK	WTITALK	CTITALK		Injection Total
x					BTCNFALK	Concentration
x					BTADSALK	Adsorption
x					BTSTMALK	Surface tension multiplier
x					BTSADALK	Surfactant adsorption multiplier
x					BTPADALK	Polymer adsorption multiplier

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 LBTCNFSU and LBTIPTAL.

E300	Block	Information
x	BALSURF	Alkaline concentration
x	BALSTML	Surface tension multiplier

Table 2.70: Alkaline SUMMARY output block controls (ECLIPSE 300 only)

## Wellbore friction model

ECLIPSE 100 only

The following items are relevant to the Wellbore Friction Model:

E100	Connection	Information
x	COFRU	Sum of connection oil flow rates upstream of, and including, this connection
x	CWFRU	Sum of connection water flow rates upstream of, and including, this connection
x	CGFRU	Sum of connection gas flow rates upstream of, and including, this connection
x	LCOFRU	As COFRU but for local grids
x	LCWFRU	As CWFRU but for local grids
x	LCGFRU	As CGFRU but for local grids

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:

E300	Conductive Fault	Information
x	CFOWC	Oil-water contact depth in the fault
x	CFGOC	Gas-oil contact depth in the fault
x	CFPRES	Pressure in the fault
x	CFOSAT	Oil saturation in the fault
x	CFGSAT	Gas saturation in the fault
x	CFWSAT	Water saturation in the fault

Table 2.72: Conductive Fault SUMMARY output controls (ECLIPSE 300)

## User defined quantities

The user defined summary output options are:

E100	E300	User defined quantity	Information
x	x	CUxxxxxxx	User defined connection quantity
x	x	FUxxxxxxx	User defined field quantity
x	x	GUxxxxxxx	User defined group quantity
x	x	RUxxx	User defined region quantity
x	x	SUxxxxxxx	User defined segment quantity
x	x	WUxxxxxxx	User defined well quantity

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:

E300	Block	Information
x	BSONRM	Normalized oil saturation
x	BSGNRM	Normalized gas saturation
x	BSWNRM	Normalized water saturation
x	BSDROW	Oil phase saturation direction indicator for oil-water system
x	BSDRWO	Water phase saturation direction indicator for water-oil system
x	BSDROG	Oil phase saturation direction indicator for oil-gas system

<b>E300</b>	<b>Block</b>	<b>Information</b>
x	BSDRGO	Gas phase saturation direction indicator for gas-oil system
x	BSDRGW	Gas phase saturation direction indicator for gas-water system
x	BSDRWG	Water phase saturation direction indicator for water-gas system
x	BSOWNT	Normalized oil turning point saturation for oil-water system.
x	BSWONT	Normalized water turning point saturation for water-oil system
x	BSOGNT	Normalized oil turning point saturation for oil-gas system
x	BSGONT	Normalized gas turning point saturation for gas-oil system
x	BSGWNT	Normalized gas turning point saturation for gas-water system
x	BSWGNT	Normalized water turning point saturation for water-gas system
x	BSOWNH	Normalized oil hysteresis saturation for oil-water system
x	BSWONH	Normalized water hysteresis saturation for water-oil system
x	BSOGNH	Normalized oil hysteresis saturation for oil-gas system
x	BSGONH	Normalized gas hysteresis saturation for gas-oil system
x	BSGWNH	Normalized gas hysteresis saturation for gas-water system
x	BSWGNH	Normalized water hysteresis saturation for water-gas system
x	BSOWNE	Normalized oil equivalent opposite direction saturation for oil-water system
x	BSWONE	Normalized water equivalent opposite direction saturation for water-oil system
x	BSOGNE	Normalized oil equivalent opposite direction saturation for oil-gas system
x	BSGONE	Normalized gas equivalent opposite direction saturation for gas-oil system
x	BSGWNE	Normalized gas equivalent opposite direction saturation for gas-water system
x	BSWGNE	Normalized water equivalent opposite direction saturation for water-gas system
x	BSORWB	Residual oil saturation used to calculate normalized saturations in oil-water system
x	BSWROB	Residual water saturation used to calculate normalized saturations in water-oil system
x	BSORGB	Residual oil saturation used to calculate normalized saturations in oil-gas system
x	BSGROB	Residual gas saturation used to calculate normalized saturations in gas-oil system
x	BSGRWB	Residual gas saturation used to calculate normalized saturations in gas-water system
x	BSWRGB	Residual water saturation used to calculate normalized saturations in water-gas system
x	BSORWP	Process dependent residual oil saturation in oil-water system
x	BSWROP	Process dependent residual water saturation in water-oil system
x	BSORGP	Process dependent residual oil saturation in oil-gas system
x	BSGROP	Process dependent residual gas saturation in gas-oil system
x	BSGRWP	Process dependent residual gas saturation in gas-water system
x	BSWRGP	Process dependent residual water saturation in water-gas system
x	BKROWR	Representative oil-water relative permeability
x	BKRWOR	Representative water-oil relative permeability
x	BKROGR	Representative oil-gas relative permeability

<b>E300</b>	<b>Block</b>	<b>Information</b>
x	BKRGOR	Representative gas-oil relative permeability
x	BKRGWR	Representative gas-water relative permeability
x	BKRWGR	Representative water-gas relative permeability
x	BKROWH	Turning point relative permeability to oil for oil-water system
x	BKRWOH	Turning point relative permeability to water for water-oil system
x	BKROGH	Turning point relative permeability to oil for oil-gas system
x	BKRGOH	Turning point relative permeability to gas for gas-oil system
x	BKRGWH	Turning point relative permeability to gas for gas-water system
x	BKRWGH	Turning point relative permeability to water for water-gas system
x	BKROWE	Equivalent relative permeability to oil for oil-water system
x	BKRWOE	Equivalent relative permeability to water for water-oil system
x	BKROGE	Equivalent relative permeability to oil for oil-gas system
x	BKRGOE	Equivalent relative permeability to gas for gas-oil system
x	BKRGWE	Equivalent relative permeability to gas for gas-water system
x	BKRWGE	Equivalent relative permeability to water for water-gas system
x	BKROWT	Opposite saturation direction turning point relative permeability to oil for oil-water system
x	BKRWOT	Opposite saturation direction turning point relative permeability to water for water-oil system
x	BKROGT	Opposite saturation direction turning point relative permeability to oil for oil-gas system
x	BKRGOT	Opposite saturation direction turning point relative permeability to gas for gas-oil system
x	BKRGWT	Opposite saturation direction turning point relative permeability to gas for gas-water system
x	BKRWGT	Opposite saturation direction turning point relative permeability to water for water-gas system
x	BIFTOW	Oil-water interfacial tension
x	BIFTWO	Water-oil interfacial tension (alias for BIFTOW)
x	BIFTGO	Oil-gas interfacial tension (alias for BIFTGO)
x	BIFTGO	Gas-oil interfacial tension
x	BIFTGW	Gas-water interfacial tension
x	BIFTWG	Water-gas interfacial tension (alias for BIFTGW)
x	BPCOWR	Representative oil-water capillary pressure
x	BPCWOR	Representative water-oil capillary pressure
x	BPCOGR	Representative oil-gas capillary pressure
x	BPCGOR	Representative gas-oil capillary pressure
x	BPCGWR	Representative gas-water capillary pressure

E300	Block	Information
x	BPCWGR	Representative water-gas capillary pressure

Table 2.74: ODD3P SUMMARY output controls (ECLIPSE 300 only)

## Asphaltene model

The following outputs are specific to the Asphaltene model activated using the [ASPALTE](#) 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](#)".

E300	Block	Information
x	BAREAC	Asphaltene reaction rates (this keyword requires the I,J,K values for the cell as well as the reaction rate number M). See Note 1.
x	BASPADS	Asphaltene adsorption volume fraction (pre-2011.1 asphaltene model only)
x	BASPDOT	Asphaltene net deposition volume fraction (pre-2011.1 asphaltene model only)
x	BASPENT	Asphaltene entrainment volume fraction (pre-2011.1 asphaltene model only)
x	BASPFLT	Average oil flow rate at the previous timestep (2011.1 asphaltene model only)
x	BASPRD	Asphaltene fraction precipitated and dissolved
x	BASPKDM	Asphaltene permeability damage multiplier (pre-2011.1 asphaltene model only)
x	BASPLIM	Asphaltene precipitate fraction limit
x	BASPLUG	Asphaltene plugging volume fraction (pre-2011.1 asphaltene model only)
x	BASPRET	Asphaltene precipitation fraction including flocs
x	BASPREW	Asphaltene precipitation fraction
x	BASPVEL	Average oil velocity at the previous timestep (2011.1 asphaltene model only)
x	BASPVOM	Asphaltene oil viscosity multiplier

Table 2.75: Asphaltene SUMMARY output controls (ECLIPSE 300)

## Simulator performance keywords

The keywords are:

E100	E300	Keyword	Description
x	x	ELAPSED	Elapsed time in seconds.
x		HLINEARS	Number of linear iterations for each gradient calculation ( <i>Gradient Option</i> ).
x		HSUMLINS	Total number of linear iterations for each gradient pressure ( <i>Gradient Option</i> ).
	x	MAXCFL	Maximum explicit CFL value for each timestep.
	x	MAXDE	Maximum change in energy density for each timestep.
x	x	MAXDPR	Maximum change in pressure for each timestep.
x	x	MAXDSG	Maximum change in gas saturation for each timestep.

E100	E300	Keyword	Description
x	x	MAXDSO	Maximum change in oil saturation for each timestep.
x	x	MAXDSW	Maximum change in water saturation for each timestep.
	x	MAXDT	Maximum change in temperature for each timestep.
x	x	MEMORYTS	Maximum current memory usage across processors as reported by the operating system.
x	x	MLINEARP	Number of pressure iterations for each timestep (CPR or JALS solver only).
x	x	MLINEARS	Number linear iterations for each timestep.
x	x	MSUMLINP	Total number of pressure iterations since the start of the run (CPR or JALS solver only).
x	x	MSUMBUG	Total number of 'bug' messages since the start of the run.
x	x	MSUMCOMM	Total number of 'comment' messages since the start of the run.
x	x	MSUMERR	Total number of 'error' messages since the start of the run.
x	x	MSUMLINS	Total number of linear iterations since the start of the run.
x	x	MSUMMESS	Total number of 'message' messages since the start of the run.
x	x	MSUMNEWT	Total number of Newton iterations since the start of the run.
x	x	MSUMPROB	Total number of 'problem' messages since the start of the run.
x	x	MSUMWARN	Total number of 'warning' messages since the start of the run.
	x	NAIMFRAC	Fraction of implicit cells in AIM run.
	x	NBAKFL	Number of calls made to scalar fallback routines in the flash calculation for each timestep.
x	x	NBYTOT	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.
x	x	NCPRLINS	Average number of pressure iterations per linear iteration for each timestep (CPR or JALS solver only).
	x	NEWTFL	Cumulative average Newton iterations per cell in two phase flash calculations.
x	x	NEWTON	Number of Newton iterations used for each timestep
x	x	NLINEARP	Average number of pressure iterations per Newton iteration for each timestep (CPR or JALS solver only).
x	x	NLINEARS	Average number of linear iterations per Newton iteration for each timestep.
x	x	NLINSMIN	Actual minimum number of linear iterations in the Newton iterations within each timestep.
x	x	NLINSMAX	Actual maximum number of linear iterations in the Newton iterations within each timestep.

E100	E300	Keyword	Description
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 <a href="#">CVCRT</a> 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 <a href="#">CVCRT</a> 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 <a href="#">CVCRT</a> 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 <a href="#">PSTEADY</a> , 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, NBAKFL, TCPUTS, TCPUDAY.  If the <a href="#">AIM</a> 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.

E100	E300	Keyword	Description
x		STEPTYPE	<p>Outputs an integer representing the reason for selecting the timestep length (see "<a href="#">Convergence Reports</a>" in the <i>ECLIPSE Technical Description</i>):</p> <p>1=INIT 2=TRNC 3=MINF 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</p> <p>For initial time, the STEPTYPE will remain blank.</p> <p>When a neat tabulated output of the summary file data is printed in the Print file at the end of a run (see <a href="#">RUNSUM</a>) or in a separate RSM file (see <a href="#">SEPARATE</a>, <a href="#">EXCEL</a> and <a href="#">LOTUS</a>), then the output is rendered as mnemonics.</p>
x	x	TCPU	<p>Current CPU usage in seconds.</p> <p>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.</p>
x	x	TCPUDAY	CPU time per day (or hour in lab units).
x		TCPUH	Total CPU time for each gradient calculation ( <i>Gradient Option</i> ).
x		TCPUHT	Total CPU time for all gradient calculations ( <i>Gradient Option</i> ).
x		TCPUSCH	Total CPU time used in SCHEDULE section.
x	x	TCPUTS	CPU time per timestep in seconds.
x		TCPUTSH	CPU time per timestep for each gradient calculation ( <i>Gradient Option</i> ).
x		TCPUTSHT	CPU time per timestep for all gradient calculations ( <i>Gradient Option</i> ).
	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	x	TIMESTEP	Timestep length.
x	x	TUNE	<p>Requests the following set of vectors for the run:</p> <p>NLRESSUM, NLRESMAX, NTS, NTSPCL, NTSMCL and NTSECL in a THERMAL simulation.</p>
x	x	WNEWTON	<p>Number of well Newton iterations taken in the last global Newton iteration.</p> <p>A negative value indicates that the well failed to converge.</p>
x		ZIPEFF	Predicted efficiency of the timestep (mainly for developer use).

E100	E300	Keyword	Description
x		ZIPEFFC	Predicted efficiency of the timestep divided by the actual efficiency (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 WGPR FGPT GGPT WGPT
- 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 FGIPL FGIPG FPR
- FAQR FAQRG AAQR AAQRG
- FAQT FAQTG AAQT AAQTG

E100	E300	Keyword	Description
x	x	DATE	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
x	x	EXCEL	Requests RUNSUM output to be in Microsoft Excel format.

E100	E300	Keyword	Description
x	x	GMWSET	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.
x	x	FMWSET	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.
x		LOTUS	Requests RUNSUM output to be in Lotus 1-2-3 format.
x		MONITOR	Outputs information required by GRAF for run-time monitoring.
x	x	NOMONITO	Disables run-time monitoring.
x	x	NARROW	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.
x	x	RPTONLY	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.
x	x	RPTONLY	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.
x		RPTSMRY	Requests printed output of keywords in the SUMMARY section
x		RUNSUM	Requests a neat tabulated output of the summary file data at the end of the normal ECLIPSE 100 printed output.
		RUNSUM	Requests a neat tabulated output of the summary file data at the end of the run, this output goes to a separate RSM file.
x		SEPARATE	Requests RUNSUM output to go to a separate RSM file.
x	x	SUMTHIN	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.

Table 2.77: Other SUMMARY keywords

## Deprecated keywords

The following SUMMARY keywords have been deprecated due to naming convention irregularities. The following replacements should be used:

E300	Deprecated Keyword	Replacement Keyword	Deprecation release
x	CCFR	CKFR	2011.1
x	CCFT	CKFT	2011.1
x	RCFR	RKFR	2011.1
x	RCFR+	RKFR+	2011.1
x	RCFR-	RKFR-	2011.1

E300	Deprecated Keyword	Replacement Keyword	Deprecation release
x	RCFT	RKFT	2011.1
x	RCFT+	RKFT+	2011.1
x	RCFT-	RKFT-	2011.1
x	BSWL	BPSWL	2011.1
x	BSWCR	BPSWCR	2011.1
x	BSWU	BPSWU	2011.1
x	BSGL	BPSGL	2011.1
x	BSGCR	BPSGCR	2011.1
x	BSGU	BPSGU	2011.1
x	BSOWC	BPSOWCR	2011.1
x	BSOGC	BPSOGCR	2011.1
x	BCFRI	BKFRI	2012.1
x	BCFRJ	BKFRJ	2012.1
x	BCFRK	BKFRK	2012.1
x	SCFRn	SCWGFR	2012.1
x	BPORVT	BRPV	2014.1
x	LBPORVT	LBRPV	2014.1

Table 2.78: Deprecated SUMMARY keywords

## Deleted keywords

The following SUMMARY keywords have been deleted. The following replacements must be used:

E300	Deleted Keyword	Replacement Keyword	Deprecation release	Deletion Release
x	BASPFL	BAREAC	2011.1	2011.1
x	WNPR	WNLPR	2008.1	2012.1
x	WNPT	WNLPT	2008.1	2012.1
x	GNPR	GNLPR	2008.1	2012.1
x	GNPT	GNLPT	2008.1	2012.1
x	FNPR	FNLPR	2008.1	2012.1
x	FNPT	FNLPT	2008.1	2012.1
x	RNIP	RNLIP	2008.1	2012.1
x	RNPR	RNLPR	2008.1	2012.1
x	RNPT	RNLPT	2008.1	2012.1

Table 2.79: Deleted SUMMARY keywords

## 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 flow 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: **WELLSPEC**, **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 **GRUPTREE**. Any other keywords that refer to a particular group must be positioned after the group has been introduced.

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

Restriction	Keyword	Description
Required	<a href="#">WELSPECS</a>	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.

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

Restrictions	Keyword	Description
	COMPDAT	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.
Hysteresis option only	COMPIMB	Imbibition table numbers for well connections
	COMPMOBI	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.
	COMPORD	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.
Cannot be used with the Vertical Equilibrium option.	COMPRP	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.
	COMPLUMP	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.
	WDFAC	Sets the well D-factor (flow-dependent skin for free gas).
	WDFACCOR	Calculates well connection D-factors (flow-dependent skin for free gas).
	WELPI	Scales the connection transmissibility factors to give the well a specified steady-state productivity/injectivity index at the current reservoir conditions.

Restrictions	Keyword	Description
	WPIMULT	Multiplies the connection transmissibility factors by a specified value.
	PICOND	Controls the calculation of the generalized pseudo-pressure option for well inflow in gas condensate runs.
	WINJMULT	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.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
COMPINJK	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.
COMPVE	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.
WPITAB	Assigns PI multiplier tables (input with keyword PIMULTAB) to wells.
PIMULTAB	Inputs a set of PI multiplier tables, which scale well connection factors as a function of the well's maximum water cut.
WBOREVOL	Sets the wellbore volume, for modeling wellbore storage in well tests.

### ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
COMPKRI	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.
COMPAGH	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.
PSEUPRES	Activates the Generalized Pseudo Pressure option for all wells.

## Define multisegment wells

### Common keywords

Keyword	Description
COMPSEGL	Defines the location of multisegment well completions in a local grid.
COMPSEGS	Defines the location of completions in a multisegment well.
WELSEGS	Defines the segment structure of a multisegment well.
WSEGAICD	Defines segments to represent an 'autonomous' inflow control device.

Keyword	Description
WSEGDFIN	Inputs a table of inclination scaling factors versus deviation angle for the Drift Flux slip model
WSEGDFMD	Defines the Drift Flux slip model
WSEGDFPA	Changes parameter values in Drift Flux slip model.
WSEGEXSS	Specifies fluid import/removal through an external source/sink.
WSEGFLIM	Defines segments to represent a flow limiting valve.
WSEGFMOD	Specifies the multi-phase flow model and related parameters.
WSEGLABY	Defines segments to represent a 'labyrinth' inflow control device.
WSEGLINK	Defines the chord segment links for specifying looped flow paths.
WSEGMULT	Imposes constant or variable multipliers for segment frictional pressure drops.
WSEGPROP	Modifies the properties of individual well segments.
WSEGSICD	Defines segments to represent a 'spiral' inflow control device.
WSEGTABL	Designates segments to have their pressure drops calculated from a VFP table.
WSEGVALV	Defines segments to represent a sub-critical valve.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
WFRICSEG	Converts WFRICTN keyword data for a friction well into data for a multisegment well.
WFRICSGL	Converts WFRICTNL keyword data for a friction well into data for a multisegment well.
WSEGINIT	Sets initial pressure and fluid contents of well segments.
WSEGITER	Activates the robust iteration scheme for solving multisegment wells.
WSEGPULL	Defines a pull through pump for a down-hole water separator.
WSEGSEP	Defines a segment to represent a down-hole separator.

### ECLIPSE 300 only keywords

Restrictions	ECLIPSE 300 only keyword	Description
	SCONINJE	Specifies secondary well injection controls.
	SCONPROD	Specifies secondary well production controls.
	STEST	Tests whether a closed secondary well should be reopened.
Thermal only	WSEGHEAT	Specifies heat loss from a segment.
	WSEGWELL	Associates a segment of a multisegment well with a secondary well name

## 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.

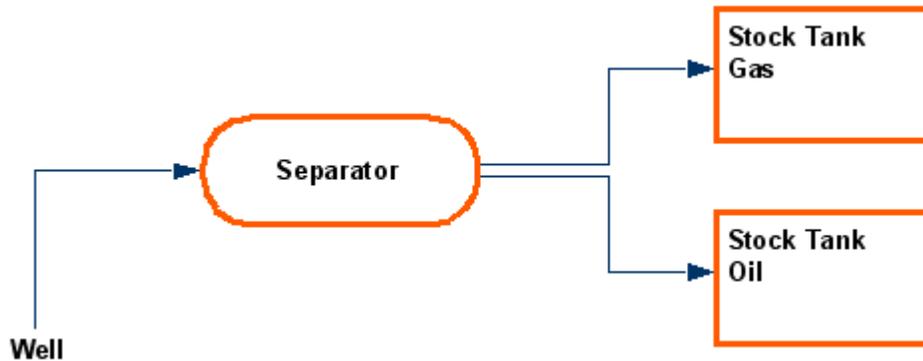


Figure 2.1. Splitting well production into stock tank oil and gas

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

Keyword	Description
<code>WEFAC</code>	Sets well efficiency factors, to take account of downtime when wells are taken down individually on a regular basis.
<code>WELOPEN</code>	Opens or closes wells or connections.
<code>WLIST</code>	Constructs well lists for use with other well keywords.

## Production well control data

### Common keywords

Restriction	Keyword	Description
Either WCONPROD or WCONHIST should be used to specify production wells once they have been introduced with WELSPECS and COMPDAT.	WCONPROD	Control data for production wells.
Either WCONPROD or WCONHIST should be used to specify production wells once they have been introduced with WELSPECS and COMPDAT.	WCONHIST	Observed production rates for history matching wells.
	WHISTCTL	Sets control mode instructions for all history matching wells.
	WELTARG	Resets an individual operating target or limit.
	WELCNTL	Resets the control mode and operating target on wells.
	WELDRAW	Sets a maximum allowable drawdown for production wells.
	WTADD	Adds to a well target or limit by a given amount.
	WTMULT	Multiplies a well target or limit by a given factor.
	WLISTARG	Resets individual operating targets or limits for a list of wells, constructed with the WLIST keyword.
	WREGROUP	Switches wells automatically between groups as certain limits are violated.
	LINCOM	Specifies coefficients for a linear combination of oil, water and gas.

### ECLIPSE 100 only keywords

- [WGORPEN](#) Sets data for the GOR Penalty option

## Injection well control data

### Common keywords

Restriction	Keyword	Description
	WCONINJE	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.
	WCONINJH	Observed injection rates for history matching wells. be used to specify injection wells once they have been introduced with WELSPECS and COMPDAT.

Restriction	Keyword	Description
	WCONINJP	Sets the rate of an injection well in terms of the voidage of the production wells that surround it.
	WHISTCTL	Sets control mode instructions for all history matching wells.
	WELTARG	Resets an individual operating target or limit.
	WELCNTL	Resets the control mode and operating target on wells
	WTMULT	Multiplies a well target or limit by a given factor.
	WLISTARG	Resets individual operating targets or limits for a list of wells, constructed with the <a href="#">WLIST</a> keyword.
Tracer option only	WTRACER	Defines the concentration of tracers in the well injection streams
Brine option only	WSALT	Defines the salt concentration in water injectors

### ECLIPSE 100 only keywords

Restriction	ECLIPSE 100 only keyword	Description
	WAPI	Defines the API of injected oil in API tracking cases.
Temperature option only	WTEMP	Defines the temperature of injected water

### ECLIPSE 300 only keywords

Restriction	ECLIPSE 300 only keyword	Description
	WINJGAS	Defines the composition of the fluid injected by wells.
	WINJOIL	Defines the composition of the fluid injected by wells.
Thermal	WINJTEMP	Defines the temperature of the fluid injected by wells
Water component model	WINJW	Defines the composition of the water injected by wells
	WELLSTRE	Sets the composition of a named wellstream for injection.
Water component model	WELLSTRW	Sets the composition of a named wellstream for water injection
	WINJEDET	Describes detailed composition of a lumped fluid for an injection well.
	WINJMIX	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.
	WELLWAG	Specifies WAG well injection targets.

## Vertical flow performance keywords

### Common keywords

Keyword	Description
<a href="#">VFPPROD</a>	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 <a href="#">WHTEMP</a> keyword). A VFP table must be entered before it is referenced in <a href="#">WCONPROD</a> or <a href="#">WHTEMP</a> .
<a href="#">VFPINJ</a>	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 <a href="#">WCONINJE</a> .
<a href="#">VFPCHK</a>	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).
<a href="#">WVFPDP</a>	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).
<a href="#">WVFPEXP</a>	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.
<a href="#">WHTEMP</a>	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.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
<a href="#">VFPTABL</a>	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.
<a href="#">WALQCALC</a>	Defines the well ALQ to be set equal to the surface density of the well's produced oil or gas.

### ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">VFPCHECK</a>	Contains data that ECLIPSE 300 uses to check for consistency between the simulator fluid model and that used in VFP <i>i</i> to generate the <a href="#">VFPPROD</a> and/or <a href="#">VFPINJ</a> keywords.

## Well economic limits and coning control

### Common keywords

Keyword	Description
WECON	Economic operating limits on well rates and production ratios.
CECON	Economic operating limits on individual well connections.
WCUTBACK	Instructions to cut back well rates to limit coning.
WLIMTOL	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.
WORKTHP	Instructions to perform a workover if the well is unable to produce at its THP limit.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
WECONINJ	Sets a minimum economic rate for injection wells.
WTHPMAX	Sets a maximum THP design limit.
WBHGLR	Sets upper limits on the gas-liquid ratio at bottom hole conditions, which will trigger workovers or rate cutbacks.
CECONT	Economic limits on well connection tracer rates and concentrations.
WCUTBACT	Well rate cutbacks and reversals based on tracer rates and concentrations.
WECONT	Economic limits on well tracer rates and concentrations.

### ECLIPSE 300 only keywords

- [WECONCMF](#) Economic operating limits on component mole fractions.

## Group controls and limits

Keyword	Description
NUPCOL	Sets the number of iterations in each timestep for which well flow targets are recalculated for group control.
GCONTROL	Sets tolerance fractions and iteration limits for group control targets and (in ECLIPSE 300) gas injection calculations.
WLIMTOL	Tolerance fraction for workover limits.

## Group production control by guide rate

### Common keywords

Keyword	Description
GCONPROD	Group production rate targets and limits for control by guide rate.
GRUPTARG	Resets an individual production rate target or limit for a group.

Keyword	Description
GPMAINT	Sets group production or injection target to maintain the average pressure in a fluid-in-place region.
WGRUPCON	Sets or scales well guide rates for group control.
GUIDERAT	Coefficients for a general formula for variable well guide rates.
GTADD	Adds to a group rate target or limit by a given factor.
GTMULT	Multiplies a group rate target or limit by a given factor.

### 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
- [GCONPRI](#) 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

ECLIPSE 100 only keyword	Description
<a href="#">GCUTBACK</a>	Instructions to cut back group rates to limit water/gas breakthrough.
<a href="#">GCUTBACT</a>	Instructions to cut back group rates based on tracer rates and concentrations.
<a href="#">GECONT</a>	Economic limits on group tracer rates and concentrations.

### Group injection control

#### Common keywords

Keyword	Description
<a href="#">GCONINJE</a>	Group injection rate targets and limits.

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.
WELLSTRE	Sets the composition of a named wellstream for injection.
WINJMIX	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.

- [GCONSALE](#) 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

Keyword	Description
<a href="#">WDRILTIM</a>	Time taken to drill each well. Limits the rate at which new wells can be drilled.
<a href="#">QDRILL</a>	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 ( <a href="#">GCONPROD</a> , <a href="#">GCONINJE</a> , <a href="#">GCONSALE</a> ) or when needed to maintain a group's production potential ( <a href="#">GDRILPOT</a> ).
<a href="#">WDRILPRI</a>	Places wells in the prioritized drilling queue. This operates in a similar way to the sequential drilling queue (keyword <a href="#">QDRILL</a> ) 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 <a href="#">DRILPRI</a> keyword.
<a href="#">DRILPRI</a>	Defines the default priority formula for the prioritized drilling queue set up with keyword <a href="#">WDRILPRI</a>
<a href="#">GDRILPOT</a>	Causes wells to be drilled from the drilling queue when a group's production rate potential falls below a specified value.
<a href="#">WORKLIM</a>	Time taken to perform a workover. Limits the rate of doing workovers.
<a href="#">GRUPRIG</a>	Assigns workover rigs to groups. Together with <a href="#">WORKLIM</a> , it limits the rate at which workovers can be performed within particular groups.
<a href="#">COMPLUMP</a>	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.

## ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
WDRILRES	Prevents two wells being drilled in the same grid block
WELSOMIN	Prevents new connections being opened in grid blocks having less than a specified minimum oil saturation.
PRORDER	Defines order of applying group production rules, including drilling and re-perforating, to maintain group production at its target rate.
WPLUG	Well plug-back lengths, for automatic plug-backs.

## Action keyword facility

Keyword	Description
ACTION	Initiates a set of actions (defined by SCHEDULE section keywords) to be executed when a specified field condition is satisfied.
ACTIONG	Defines actions to be triggered by group quantities.
ACTIONR	Defines actions to be triggered by region quantities.
ACTIONW	Defines actions to be triggered by well quantities.
ACTIONS	Defines actions to be triggered by well segment quantities.
ACTIONX	Defines actions to be triggered by a set of conditions on various quantities
DELAYACT	Defines actions to be triggered a given time after another action has been triggered.
ENDACTIO	Indicates the end of the set of actions initiated with the ACTION keyword family.

## 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)

ECLIPSE 100 only keyword	Description
WLIFT	Instructions for automatic retubing, lift switching and THP reduction, when a well or group production rate falls below a specified value.

<b>ECLIPSE 100 only keyword</b>	<b>Description</b>
GLIFTLIM	Limits on artificial lift capacity for groups. Prevents further automatic lift switching when group lift capacity would be exceeded.
WLIMTOL	Tolerance fraction for lift switching limits. If violated by more than the tolerance, the timestep will be repeated after retubing or lift switching.
WELTARG	Change VFP table number or artificial lift quantity, for manual retubing or lift switching.
PRORDER	Defines order of applying group production rules, including artificial lift operations, to maintain group production at its target rate.
VFPTABL	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.

## Scale deposition model

<b>Keyword</b>	<b>Description</b>
SCDATAB	Defines scale damage tables which govern the reduction in the connection transmissibility factor caused by deposited scale.
SCDPTAB	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.
SCDPTRAC	Sets a passive water tracer to be associated with the sea water fraction.
WSCTAB	Specifies which scale deposition table and which scale damage table will be used to model scale effects for a well.
WSCCLEAN	Reduces the amount of scale deposited around well connections
WSCCLENL	Reduces the amount of scale deposited around local grid well connections

## Flash transformation and separator conditions

### Common keywords

<b>Keyword</b>	<b>Description</b>
COMPFLSH	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.

### ECLIPSE 100 only keywords

<b>Restriction</b>	<b>ECLIPSE 100 only keyword</b>	<b>Description</b>
	GSEPCOND	Assigns separators to groups, for use with SEPVALS

Restriction	ECLIPSE 100 only keyword	Description
Use of SEPVALS is not allowed in runs using the COMPFLSH keyword.	SEPVALS	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.

## 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

Keyword	Description
<a href="#">AQUFLUX</a>	Modifies the water influx rate from a constant flux aquifer.
<a href="#">AQUCHGAS</a>	Modifies the gas head for a constant head gas aquifer.
<a href="#">AQUCHWAT</a>	Modifies the water head for a constant head water aquifer.
<a href="#">AQUCT</a>	Modifies the properties of a Carter-Tracy aquifer. <a href="#">AQUFETP</a> Modifies the properties of a Fetkovich aquifer.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
<a href="#">RAINFALL</a>	Specifies a monthly flux rate for constant flux aquifers.
<a href="#">AQUCWFAC</a>	Modifies the properties of all <a href="#">AQUCHWAT</a> type aquifers.

### ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">AQSTREAM</a>	Sets composition of gas stream for <a href="#">AQUCHGAS</a> aquifer. For the <a href="#">GASWAT</a> and <a href="#">CO2STORE</a> options, sets composition of water <a href="#">AQUCHWAT</a> , <a href="#">AQUCT</a> , <a href="#">AQUFETP</a> and <a href="#">AQUFLUX</a> aquifers.
<a href="#">AQSTREAMW</a>	For the multi-component water option ( <a href="#">COMPW</a> ), sets composition of water <a href="#">AQUCHWAT</a> , <a href="#">AQUCT</a> , <a href="#">AQUFETP</a> and <a href="#">AQUFLUX</a> aquifers.

## Dissolved gas and vaporized oil constraints (ECLIPSE 100)

Restriction	ECLIPSE 100 only keyword	Description
If not specified, gas redissolves instantly in under-saturated oil	DRSDT	Gas re-solution rate.
If not specified, oil vaporizes instantly in under-saturated wet gas	DRVDT	Oil vaporization rate
Only for systems with both wet gas and live oil	VAPPARS	Parameters controlling vaporization of oil in under-saturated gas.

## Geomechanics option (ECLIPSE 300)

ECLIPSE 300 only keyword	Description
DISPBC	Permits the application of displacement (Dirichlet) or velocity boundary conditions.
GMDISBC	Permits the application of displacement (Dirichlet) boundary conditions.
GMPSTBC	Principal stresses can be applied as boundary conditions
GMTRABC	Permits application of traction (Neumann) boundary conditions.
PSTRBC	Principal stresses can be applied as boundary conditions.
STRESEBC	Permits the application of a load or a velocity (rigid) boundary condition on all external and internal surfaces of the simulation grid.
TRACTBC	Permits application of traction (Neumann) boundary conditions.
WSCTAB	Links a well to a scale damage table.

## Grid block property modifications

### Common keywords

Keyword	Description
BOX	Defines the current input box, for use with MULTX, MULTX-, MULTY, MULTY-, MULTZ, MULTZ- and MULTPV.
MULTFLT	Modifies the transmissibility across a named fault.
MULTPV	Multiplies the pore volumes of cells within the current input box.
MULTX	Multiplies X-direction transmissibilities of cells within the current input box
MULTX-	Multiplies negative X-direction transmissibilities of cells within the current input box.
MULTY	Multiplies Y-direction transmissibilities of cells within the current input box.
MULTY-	Multiplies negative Y-direction transmissibilities of cells within the current input box.
MULTZ	Multiplies Z-direction transmissibilities of cells within the current input box.

Keyword	Description
<a href="#">MULTZ-</a>	Multiplies negative Z-direction transmissibilities of cells within the current input box.
<a href="#">MULTR</a>	Multiplies R-direction transmissibilities of cells within the current input box.
<a href="#">MULTR-</a>	Multiplies negative R-direction transmissibilities of cells within the current input box.
<a href="#">MULTTHT</a>	Multiplies Theta-direction transmissibilities of cells within the current input box.
<a href="#">MULTTHT-</a>	Multiplies negative Theta-direction transmissibilities of cells within the current input box.

## ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
<a href="#">EXCAVATE</a>	Removes cells during a run.

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
<a href="#">OVERTR</a>	Overwrites R direction transmissibilities.
<a href="#">OVERTTHT</a>	Overwrites theta direction transmissibilities.
<a href="#">OVERTX</a>	Overwrites X direction transmissibilities.
<a href="#">OVERTY</a>	Overwrites Y direction transmissibilities.
<a href="#">OVERTZ</a>	Overwrites Z direction transmissibilities.

## Dynamic Regions Facility

Keyword	Description
<a href="#">DYNAMICR</a>	Defines regions based on when a set of conditions are satisfied. These regions may be static or allowed to change dynamically.
<a href="#">ENDDYN</a>	Indicates the end of a DYNAMICR keyword sequence

## 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)

ECLIPSE 300 only keyword	Description
LAGTRUPD	Defines the streamline update conditions and solver method for the Lagrangian tracer solver.
LTRACE	Forces a streamline trace at the end of the next timestep.
NOLTRACE	Prevents streamline tracing until the next LTRACE or LAGTRUPD.
RPTSLN	Controls the output of streamline data as SLN files.

## Simulator controls

The keywords are:

### Common keywords

Keyword	Description
NEXTSTEP (or NEXT)	Restricts the next timestep to a specified maximum size.
TUNING	Timestep and convergence controls.
SKIPREST	Makes ECLIPSE skip the subsequent keywords in a restart run until the restart time is reached.
NSTACK	Sets the linear solver stack size.
ZIPPY2	Experimental feature to automatically select timesteps in poorly converging runs.
ZIPP2OFF	Turns off ZIPPY2 feature.

### ECLIPSE 100 only keywords

ECLIPSE 100 only keyword	Description
TUNINGDP	Supplies a modified set of convergence parameters for some high throughput cases.
TIGHTEN	Enables convergence parameters to be tightened
TIGHTENP	Enables convergence parameters to be tightened.
DIMPES	Sets the <code>IMPES</code> solution procedure.
DIMPLICIT	Restores the fully implicit solution procedure.
MATCORR	Activates the material balance correction option, to reduce the accumulated material balance error.
READDATA	Reads files of SCHEDULE data that may be constructed externally during the course of the run.
OPTIONS	Activate special options within particular facilities in ECLIPSE 100.

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
TOPT	Preset tuning options.
TSCRIT	Timestep controls.
CVCRIT	Convergence criteria.
FCCRIT	Flash convergence criteria.
LSCRIT	Linear solution convergence criteria.
LSCRITS	Linear solution convergence criteria for each individual sub-grids.
AIMFRAC	Sets the target implicit fraction in AIM mode.
AIMPVI	Specifies the minimum pore volume for an explicit cell in AIM mode.
AIMCON	Controls implicit cells within an AIM run.
CHANDIMS	Increase well array dimensions.
OPTIONS3	Activate special options within particular facilities in ECLIPSE 300.
WCOMPRES	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

## User defined values

- [UDQ](#) Specify user defined quantities.

## Output

### Common keywords

Keywords	Description
<a href="#">RPTSCHED</a>	Report switches to select which simulation results are to be printed at report times.
<a href="#">RPTRST</a>	Controls data written to the Restart file.
<a href="#">SAVE</a>	Controls solution output for plotting and flexible restarts.
<a href="#">OUTSOL</a>	Writes the current solution to a Save file for fast restarts.
<a href="#">BOUNDARY</a>	Defines section of grid for which output maps are required.
<a href="#">WPAVE</a>	Controls the calculation of well block average pressures that may be output to the Summary file.
<a href="#">WWPAVE</a>	Controls the calculation of well block average pressures for an individual well. Average pressures may be output to the Summary file.
<a href="#">WPAVEDEP</a>	Specifies a reference depth for the calculation of well block average pressures.
<a href="#">WRF'TPLT</a>	Requests output of RFT, PLT and segment data to the RFT file.
<a href="#">WELDEBUG</a>	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.)

Keywords	Description
WELEVNT	Sets SUMMARY keyword WPWEM output for individual wells.
WTEMPQ	Queries the well name and well list template matching algorithm
RPTONLY	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.
RPTONLYO	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.
RPRKPAME	Outputs the rock compaction
SUMTHIN	Reduces the number of data written to summary output files.

## ECLIPSE 100 only keywords

- [WRFT](#) Requests output of well block data to the RFT file.

## ECLIPSE 300 only keywords

ECLIPSE 300 only keyword	Description
DBGODD3P	Debug output for <a href="#">ODD3P</a> option.
RPTPRINT	Controls the output of printed simulation results.
AUTOSAVE	Writes the solution to a Save file every 'n' steps.
SAVEEND	Requests that the solution is written to a Save file whenever the run ends.
GSEPREPT	Specifies the separator chain used for sum-then-separate summary section keywords such as <a href="#">FGPRG</a>
ROCKPAME	pore volume and the permeability multipliers versus pressure for a given cell.

## 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

Keyword	Description
TSTEP	Advances the simulation over specified intervals between reporting times.
DATES	Advances the simulation to specified dates at which reports are required.
SIMULATE	Counteracts a NOSIM keyword and turns on the timestep simulation.

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

ECLIPSE 100 only keyword	Description
PLYADS	Polymer adsorption functions (tabulated).
PLYDHFLF	Polymer thermal degradation half life.
PLYMAX	Polymer/salt concentrations for mixing calculations.
PLYROCKM	Polymer-rock properties.
PLYSHEAR	Polymer shear thinning/thickening data.
PLYSHLOG	Activates a logarithm-based polymer shear thinning/thickening model that may be used as an alternative to PLYSHEAR. By default, this model is based on water velocity; if used in conjunction with keyword SHRATE then a switch to shear rate form is applied.
SHRATE	Activates the shear rate form of the alternative polymer shear thinning/thickening logarithmic model.

### 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.

Keyword	Description
<a href="#">OPTCONS</a>	Defines constraints for the reservoir optimization problem.
<a href="#">OPTDIMS</a>	Specifies dimensions for the reservoir optimization problem.
<a href="#">OPTFUNC</a>	Specifies the objective function for reservoir optimization.
OPTIMIZE	Defines the start of the optimization section.
<a href="#">OPTLOAD</a>	Hot-starts a reservoir optimization simulation.
<a href="#">OPTOPTS</a>	Specifies options that control the reservoir optimization.
<a href="#">OPTPARS</a>	Defines parameters for the reservoir optimization problem
<a href="#">OPTTUNE</a>	Defines tuning parameters for reservoir optimization.
<a href="#">RPTOPT</a>	Reservoir optimization report options.

Table 2.80: OPTIMIZE section keywords

# 3

## *Keywords*

---

### **A**

This section contains an alphabetic listing of keywords starting with the letter A.

## ACF

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## ACFDET

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 0.001 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (CO<sub>2</sub>)

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is used to specify the activity coefficient model for CO<sub>2</sub> 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 CO<sub>2</sub> 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 CO<sub>2</sub>:

```
ACTCO2S
  2 /
```

## ACTDIMS

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

*ECLIPSE 100 only*

## 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 (MNL PAC), 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

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 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 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 ACTION keyword and its corresponding ENDACTIO keyword (including the ENDACTIO keyword itself) is limited to MNLPAK 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

	FGPR	Field gas production rate
	FGOR	Field gas oil ratio
	FWCT	Field water cut
	FPR	Field average pressure.
<i>ECLIPSE 100 only</i>	FOPT	Field oil production cumulative total
<i>ECLIPSE 100 only</i>	FWPT	Field water production cumulative total
<i>ECLIPSE 100 only</i>	FGPT	Field gas production cumulative total
<i>ECLIPSE 100 only</i>	FOIR	Field oil injection rate
<i>ECLIPSE 100 only</i>	FWIR	Field water injection rate
<i>ECLIPSE 100 only</i>	FGIR	Field gas injection rate
<i>ECLIPSE 100 only</i>	FOIT	Field oil injection cumulative total
<i>ECLIPSE 100 only</i>	FWIT	Field water injection cumulative total
<i>ECLIPSE 100 only</i>	FGIT	Field gas injection cumulative total
<i>ECLIPSE 100 only</i>	FCPR	Field polymer production rate
<i>ECLIPSE 100 only</i>	FCPC	Field polymer production concentration
<i>ECLIPSE 100 only</i>	FSPR	Field salt production rate
<i>ECLIPSE 100 only</i>	FSPC	Field salt production concentration
	FTPR	Field tracer production rate (the tracer name must be added)
	FTPC	Field tracer production concentration (the tracer name must be added).
<i>ECLIPSE 300 only</i>	FNLPR	Field NGL production rate
<i>ECLIPSE 300 only</i>	FCNMR	Field component molar rates in the NGL phase
	FUxxxxxx	User defined field quantity

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.

```

ACTION
ACT1 FGOR > 3.0 /
WELOPEN
PROD5 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 ACTIONX 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 be 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 MNLPAK 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 (/):

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

<i>ECLIPSE 300 only</i>	GWGPR	Group wet gas production rate
<i>ECLIPSE 300 only</i>	GWGPT	Group wet gas production cumulative total
<i>ECLIPSE 300 only</i>	GWGIR	Group wet gas injection rate
<i>ECLIPSE 300 only</i>	GWGIT	Group wet gas injection cumulative total
	GGOR	Group gas oil ratio
	GWCT	Group water cut
	GWGR	Group water gas ratio
	GGLR	Group gas liquid ratio
<i>ECLIPSE 100 only</i>	G CPR	Group polymer production rate
<i>ECLIPSE 100 only</i>	G CPC	Group polymer production concentration
<i>ECLIPSE 100 only</i>	G SPR	Group salt production rate
<i>ECLIPSE 100 only</i>	G SPC	Group salt production concentration
	G TPR	Group tracer production rate (the tracer name must be added)
	G TPC	Group tracer production concentration (the tracer name must be added)
<i>ECLIPSE 300 only</i>	G EXGR	Group excess gas rate
<i>ECLIPSE 300 only</i>	G NLPR	Group NGL Production rate
<i>ECLIPSE 300 only</i>	G CNMR	Group component molar rates in NGL phase
	GUxxxxxx	User defined group quantity

- The operator for the triggering condition.

This must be one of:

‘>’ Greater than

‘<’ Less than

- The value of the triggering condition.

The units depend upon the quantity selected in item 3.

- 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 MNL PAC 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 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 (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.

4. The quantity to which the triggering condition applies.

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)
- ECLIPSE 100 only* RGIPG Region gas in place (in gas phase only)
- RUxxx User defined region 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 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 be 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 MNLPAK 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 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 NSEGMAX (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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

---

## ECLIPSE 100

Practically any SCHEDULE section keyword may be placed between ACTIONW 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.

## ECLIPSE 300

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 MNLPAK 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 (/):

## 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:

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
<i>ECLIPSE 300 only</i>	WWGPR Well wet gas production rate

<i>ECLIPSE 300 only</i>	WWGPT	Well wet gas production cumulative total
<i>ECLIPSE 300 only</i>	WWGIR	Well wet gas injection rate
<i>ECLIPSE 300 only</i>	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
<i>ECLIPSE 100 only</i>	WBGLR	Well gas liquid ratio at bottom hole conditions
<i>ECLIPSE 300 only</i>	WXMF_nn	Well liquid mole fraction in component nn
<i>ECLIPSE 300 only</i>	WYMF_nn	Well vapor mole fraction in component nn
<i>ECLIPSE 300 only</i>	WZMF_nn	Well total mole fraction in component nn
<i>ECLIPSE 100 only</i>	WCPR	Well polymer production rate
<i>ECLIPSE 100 only</i>	WCPC	Well polymer production concentration
<i>ECLIPSE 100 only</i>	WSPR	Well salt production rate
<i>ECLIPSE 100 only</i>	WSPC	Well salt production concentration
	WTPR	Well tracer production rate (tracer name needs to be added)
	WTPC	Well tracer production concentration (tracer name needs to be added)
<i>ECLIPSE 300 only</i>	WNLPR	Well component molar rates in NGL phase
<i>ECLIPSE 300 only</i>	WCNMR	Well NGL Production rate
	WUxxxxxxx	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.

```

ACTIONW
ACT5 'PR*' WBHP < 3500 10000 /
WTMULT
 '?' LRAT 0.8 /
/
ENDACTIO
ACTIONW
ACT6 'PR-A*' WOPR < 1000.0 /
WELOPEN
PR-A5 OPEN /

```

/  
ENDACTIO

## ACTIONX

## Initiates a set of keywords to be processed when a set of conditions are satisfied

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## ECLIPSE 300

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, ACTIONW, ACTIONX), 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

'>=' or '.GE.' Greater than or equals

'<=' or '.LE.' Less than or equals

'=' or '.EQ.' Equals

'!=' 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:

AND applies a logical 'and' to this condition with the following

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 AND and 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 "ALWAYS" condition

This is an exception to the above format. This condition has no operators and no quantities; it consists of the single word "ALWAYS" (including the double quotes). The record and keyword are terminated as for all other conditions.

"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 [DYNAMICR](#) keyword, but is not restricted to it.

## Quantity types and definitions

### 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

(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.

### 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 NSEGMX (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.

*ECLIPSE 300 only*

- 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.

```

WLISTDYN
'*HIGHWCT' '*' WWCT > 0.8 /
/

ACTIONX
WORKOVER 10000 /

```

ECLIPSE 300 only

```

MNTH .GE. MAR AND /
MNTH .LE. OCT AND /
GMWL '*HIGHWCT' .GE. 4 /
/
WECON
 '*HIGHWCT' 2* 0.75 2* CON /
/
ACTIONX
 WECONOFF 1 /
 MNTH .GT. OCT /
/
WECON
 '* ' 2* 1.0 2* CON /
/
ENDACTIO
ENDACTIO

```

### 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.

```

ACTIONX
 DELAY 1 /
 GWCT 'FIELD' > 0.8 AND /
 DAY > 1 AND /
 MNTH > JUN AND /
 YEAR >= 2021 /
/
WELOPEN
 PR-A5 OPEN /
/
ENDACTIO

```

### Example 5

ECLIPSE 300 only

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.

```

ACTIONX
 ACT5 100000 /
 WCGMR_2 'PROD1' > WCGMR_5 'PROD2' AND /
 GCGMR_3 'G1' > GCGMR_7 'G2' OR /
 FCGMR_1 > FCGMR_7 /
/
WTMULT
 'PR*' LRAT 0.8 /
/
ENDACTIO

```

## Example 6

ECLIPSE 300 only

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.

```
UDQ
-- A COUPLE OF CONNECTION QUANTITIES
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 100000 /
WUMW 'PROD1' > WUMW 'PROD2' AND /
GUMW 'G1' > GUMW 'G2' OR /
FUMW < 1 /
/

WTMULT
'?' LRAT 0.5 /
/

ENDACTIO
```

## ACTNUM

## Active grid block identification

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 IOCN 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

```

```
0 0 0 1 1 1 0 0
0 0 0 0 0 1 0 1
1 1 0 0 0 0 0 1
1 1 1 1 1 1 1 1
1 1 1 1 1 1 1 1
0 0 1 0 0 1 1 1 /
```

## ACTPARAM Parameters for the ACTION facility

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ADD

## Adds a constant to the specified array in current box

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

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.

## 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
FLUXNUM	MULTNUM	MPFANUM
<i>ECLIPSE 300 only</i>	MIDS	

### EDIT section arrays

PORV	DEPTH	
TRANX (TRANR)	TRANY (TRANTHT)	TRANZ
DIFFX (DIFFR)	DIFFY (DIFFTHT)	DIFFZ
<i>ECLIPSE Thermal only</i>	HEATTX (HEATTR)	HEATTY (HEATTTHT) HEATTZ (HEATTZ)
<i>ECLIPSE Thermal only</i>	ROCKV	

### PROPS section arrays

SWL	SWCR	SWU
SGL	SGCR	SGU
KRW	KRO	KRG
PCG	PCW	
<i>ECLIPSE 300 only</i>	PSORG	PSORW PSGRW
<i>ECLIPSE 300 only</i>	PSGRO	PSWRO PSWRG
<i>ECLIPSE 300 only</i>	HSORG	HSORW HSGRW
<i>ECLIPSE 300 only</i>	HSGRO	HSWRO HSWRG

### REGIONS section arrays

SATNUM	PVTNUM	EQLNUM
IMBNUM	FIPNUM	ENDNUM

ROCKNUM MISCNUM WH2NUM

WH3NUM

*ECLIPSE 300 only*

EOSNUM

*ECLIPSE 300 only*

PSTNUM ISTDNUM DSTNUM

*ECLIPSE 300 only*

SDROW SDRG 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The name of the array to be modified
- 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
- 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.
- 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.

---

**Note:** In the SCHEDULE section, regions (pre-defined by the `FIPxxxx` keyword in the REGIONS section) can be used in item 4.

---

## 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	
<i>ECLIPSE 300 only</i>	FLUXNUM	MULTNUM	MPFANUM
<i>ECLIPSE 300 only</i>	MIDS		

### EDIT section arrays

PORV	DEPTH		
TRANX (TRANR)	TRANY (TRANTHT)	TRANZ	
DIFFX (DIFFR)	DIFFY (DIFFTHT)	DIFFZ	
<i>ECLIPSE Thermal only</i>	HEATTX (HEATTR)	HEATTY (HEATTTHT)	HEATTZ (HEATTZ)
<i>ECLIPSE Thermal only</i>	ROCKV		

### PROPS section arrays

SWL	SWCR	SWU	
SGL	SGCR	SGU	
KRW	KRO	KRG	
PCG	PCW		
<i>ECLIPSE 300 only</i>	PSORG	PSORW	PSGRW
<i>ECLIPSE 300 only</i>	PSGRO	PSWRO	PSWRG
<i>ECLIPSE 300 only</i>	HSORG	HSORW	HSGRW
<i>ECLIPSE 300 only</i>	HSGRO	HSWRO	HSWRG

**REGIONS section arrays**

SATNUM PVTNUM EQLNUM

IMBNUM FIPNUM ENDNUM

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 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* PERMZ 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 ADDRREG 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 ADDREG keyword is not available for use with local grid systems.

ECLIPSE 100 only

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:

Keyword	Description
<a href="#">DIFFMMF</a>	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.
<a href="#">MULSGGD</a> <a href="#">MULSGGDV</a>	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.
<a href="#">MULTSIG</a> <a href="#">MULTSIGV</a>	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.
<a href="#">MULTX</a> <a href="#">MULTY</a> <a href="#">MULTZ</a>	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.
<a href="#">PERMX</a> <a href="#">PERMY</a> <a href="#">PERMZ</a>	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.
<a href="#">PORO</a>	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.
<a href="#">SIGMA</a> <a href="#">SIGMAV</a>	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.
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 3.1: 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
.....
    
```

```
2 2 2 3
2 2 3 3
2 3 3 3
3 3 3 3 /
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.
/
```

## 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 /
/
```

## ADDZCORN Adds a constant to the corner point depth array

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$0 \leq IX1A \leq NDIVIX \quad 0 \leq IX2A \leq NDIVIX$$

$$0 \leq JY1A \leq NDIVIY, \quad 0 \leq JY2A \leq NDIVIY$$

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 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 NDIVIZ=5 in RUNSPEC keyword DIMENS:

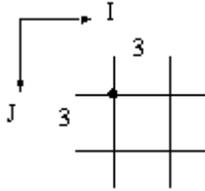
```

ADDZCORN
 10.0  3  4  3  4  1  5           / Add 10ft to all layers
-20.0  7  9  6  9  1  5  7  9  6  9 / Raise a fault block 20ft
-15.0 10 13 15 15  1  1  4* TOP   / Raise the top surface only
/

```

### Example 2

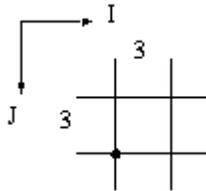
Add 10 ft. to the top-left corner of cells (3,3,1) to (3,3,5):



```
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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (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 ≥ 4:

```

ADSALNOD
  1.0
  5.0
 10.0
 30.0 /
  5.0
 30.0 /

```

## ADSORP

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [PLYADS](#), [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:

$$C_{ads} = \frac{aC^m}{1+bC} \quad a = (a_1 + a_2 C_{SE}) \left( \frac{K_{ref}}{K} \right)^n,$$

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  $K = \sqrt[3]{K_x K_y K_z}$

$K_{ref}$  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:

```

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  /

```

## AIM

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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), [IMPSTAT](#) 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- An integer which controls the implicit treatment of the wells.
  - Wells are not given any special treatment.
    - All the inner solver direction lines of cells containing the completions of open wells are set to be implicit.
    - All the cells containing the completions of open wells are set to be implicit.
    - All the cells containing the completions of open wells and their neighbors are set to be implicit.

DEFAULT: 2

Thermal DEFAULT: 3

- An integer which controls the implicit treatment of LGRs (this item does not override the [NOAIMLGR](#) keyword).
  - All cells in LGRs are set implicit
  - LGRs are not given any special treatment.

DEFAULT: 1

- All cells below the specified pore volume threshold are made implicit. A negative value indicates that no pore volume limit is enforced.

UNITS:  $\text{rm}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{rm}^3$  (PVT-M)

DEFAULT: -1.0

- 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

- 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

*Thermal option*

DEFAULT: 0.1

*CO2STORE option*

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*

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.

DEFAULT: 100.0

Thermal only

13. Threshold for gas saturation above which a cell will be designated as implicit. A negative value turns this option off.

DEFAULT: -1.0

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:

$$S^{(n+*)} = S^{(n)} + \delta \times \Delta t^{(n)} \max \left( \left[ \frac{(S^{(n)} - S^{(n-1)})}{\Delta t^{(n-1)}} \right], 0 \right) \quad \text{Eq. 3.1}$$

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--extrap
   2   1*   1*   1   0.5           5           0.125           5           5           1
/

```

## AIMFRAC

### Set target implicit fraction in AIM mode

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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:  $\text{rm}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{rm}^3$  (PVT-M)

### Example

```
AIMPVI
12 /
```

## AJGPARAM Specify static parameters for adjoint gradient calculation

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE
	OPTIMIZE

**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

TRANX 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' 'PERMX' /
```

## AJGRADNT Request sensitivities with respect to grid properties using the adjoint method

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
	OPTIMIZE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE
	OPTIMIZE

**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 ' /
/
```

## ALKADS

## Alkaline adsorption functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The local alkaline concentration in the solution surrounding the rock.  
The values should increase monotonically down the column.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)
- 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

```

ALKADS
.0000 .00000
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0005 .00007
.0008 .00008
.0010 .00010 /

```

## ALKALINE **Enables the Alkaline model**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

## ALKROCK Specifies the alkaline-rock properties

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## ALPHA

## Transport coefficient tables

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_c$  oil transport coefficients, followed by  $N_c$  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
/
```

## ALPHAI

## Imbibition transport coefficient tables

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
--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
/

```

## ALPHANUD **ALPHAD** region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 **TABDIMS**). The data must be terminated by a slash (/).

The **ALPHANUD** region number specifies which set of transport coefficient tables (input using **ALPHAD** in the **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 **NTALPHA=4**; **NDIVIX=8**, **NDIVIY=6** and **NDIVIZ=5** specified in **RUNSPEC** keyword **DIMENS**; and no input **BOX** set:

```
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 /
```

## ALPHANUI **ALPHAI** region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ALPHANUM ALPHA region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ALPOLADS Polymer adsorption multipliers as a function of alkaline concentration

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (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 /

```

## ALSURFST

## Water/oil surface tension multipliers as a function of alkaline concentration

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The local alkaline concentration.  
The values should increase monotonically down the column.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)
- 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

```

ALSURFST
.0000 1.0000
.0003 0.90005
.0005 0.90007
.0008 0.90008
.0010 0.90008 /
.0000 1.00000
.0010 0.90010 /

```

## AMALGAM Specify LGR amalgamations

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 300

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`.

ECLIPSE 100

---

**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`).

---

ECLIPSE 100

---

**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

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 H<sub>2</sub>O in the list of components set by [COMPS](#), using the name H<sub>2</sub>O, 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<sub>2</sub> 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<sub>2</sub> data followed by the water fraction. Usually it is preferable to use the [RSW](#) keyword to input the CO<sub>2</sub> concentration in the aqueous phase.

In the [H2SSOL](#) case only two components, H<sub>2</sub>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<sub>2</sub>S data followed by the water fraction.

If both [CO2SOL](#) and [H2SSOL](#) are present three components, CO<sub>2</sub>, H<sub>2</sub>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<sub>2</sub> data followed by H<sub>2</sub>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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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,  $\text{CO}_2$ ,  $\text{H}_2\text{S}$  and  $\text{H}_2\text{O}$ ) 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  $\text{H}_2\text{O}$  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 ( $\text{H}_2\text{O}$ ).

If AMFVD is not used in equilibrated cases ([EQUIL](#) keyword present) then water will be entirely composed of  $\text{H}_2\text{O}$ .

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,  $\text{CO}_2$  or  $\text{H}_2\text{S}$ ) 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

### 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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## APIGROUP **Allows grouping of oil PVT tables for API tracking**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 **PVTNUM** is used to specify which API group is to be used for each grid block. For example, a cell in **PVTNUM** region 2 will use the second API group to obtain the oil properties. For water and gas PVT properties, the **PVTNUM** regions correspond to the actual table numbers, as in runs that do not use API tracking. So a cell in **PVTNUM** region 2 still uses table number 2 for its water and gas properties.

### Example

```
APIGROUP
2 /
```

## APILIM

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 the 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [AQUDIM5](#).
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: m<sup>2</sup> (`METRIC`), ft<sup>2</sup> (`FIELD`), cm<sup>2</sup> (`LAB`), m<sup>2</sup> (`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 `AQUDIMS` 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 1 10 1 2 'I+' 1000 /
/
```

## AQANNC

## Set connection values explicitly for analytic aquifers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (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 /
/
```

## AQANTRC

## Sets initial tracer concentrations for analytic aquifers

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

ECLIPSE 100

---

**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 1<sup>st</sup> 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 /
```

```
2 WT1 0.0 /  
2 WT2 1.0 /  
/
```

## Example 2

Set 60% concentration of TR6 in the first aquifer.

```
AQANTRC  
1 TR6 0.6 /  
/
```

## AQCOEF

**A-coefficient terms for water component**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

In [GASWAT](#) mode the normal temperature dependence of the Peng-Robinson equation for the water component is modified to take the form:

$$\alpha^{1/2} = A_1 + A_2 T + A_3 T^{-3} \quad \text{Eq. 3.2}$$

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
.4530      1.0   .34E-02 /
```

## AQSTREAM Set composition of the injected gas stream or water analytic aquifer

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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  $N_c^{\text{th}}$  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 /
/
```

## AQSTREAW Set composition of multi-component water analytic aquifer

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.
- ...
- ...
- $N_w + 1$ .  $a_{N_w}$ , the mole fraction of the  $N_w^{\text{th}}$  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 [AQSTREAW](#) 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

```
AQSTREAW
1 0.5 0.3 0.1 0.05 0.05 /
/
```

## AQUALIST Sets up lists of aquifers for output

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 AQUUDIMS.

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 AQUUDIMS).

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

AQUANCON connects analytic aquifers, declared using [AQUCT](#), [AQUFETP](#), [AQUFLUX](#), [AQUCHWAT](#) or [AQUCHGAS](#), 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  $Z = \text{NDIVIZ}$  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:  $\text{m}^2$  (`METRIC`),  $\text{ft}^2$  (`FIELD`),  $\text{cm}^2$  (`LAB`),  $\text{m}^2$  (`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 `AQUDIMS` 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 the argument `AQUFET`, `AQUCT` or `AQUANCON` in the `RPTSOL` keyword to 2.

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

The number of analytic aquifers is specified using the [AQUDIMs](#) 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 [AQUDIMs](#).
  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](#)), atma ([LAB](#)), atma ([PVT-M](#))
  4. Aquifer productivity index  
(Total influx rate per unit pressure difference.)  
UNITS: sm<sup>3</sup>/day/bars ([METRIC](#)), Mscf/day/psi ([FIELD](#)), scc/hr/atm ([LAB](#)), sm<sup>3</sup>/day/atm ([PVT-M](#))
  5. Table number for gas pressure properties (for example see keyword [PVTG](#)).  
DEFAULT: 1
- ECLIPSE 300* 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.
- ECLIPSE 300*
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.

*ECLIPSE 100*

---

**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 [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/pressure gas ‘aquifer’ defined by this keyword are read in by either the [AQUANCON](#) or [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 reentering the [AQUCHGAS](#) keyword in the [SCHEDULE](#) section.

ECLIPSE 300 only

**Note:** For compositional runs, the `AQSTREAM` keyword must be used together with the `AQUCHGAS` keyword.

The aquifer inflow is given by the equation:

$$Q_{ai} = \frac{d}{dt} (G_{ai}) = J\alpha_i [p_a + p_c - p_i + \rho g (d_i - d_a)] \quad \text{Eq. 3.3}$$

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:

```

AQUCHGAS
 1 0.0 1.0 2.0E9 1 /
/
AQUANCON
 1 1 10 10 10 1 2 'J+' /
 1 10 10 1 10 1 2 'I+' /
/

```

## AQUCHWAT Specifies the property data for constant head/pressure water aquifers

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 AQU DIMS.

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:  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{stb}/\text{day}/\text{psi}$  (FIELD),  $\text{scc}/\text{hr}/\text{atm}$  (LAB),  $\text{sm}^3/\text{day}/\text{atm}$  (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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), atma (LAB), atma (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), atma (LAB), atma (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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: -infinity

12. Maximum allowed flow per aquifer connection

UNITS: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: infinity

13. Do you want to remove the depth term,  $J\alpha_i \rho g (d_i - d_a)$ , 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

ECLIPSE 300

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 (/).

ECLIPSE 100

**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 [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 entering the [AQUCHWAT](#) keyword again in the [SCHEDULE](#) section.

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 [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 aquifer inflow is given by the equation:

$$Q_{ai} = \frac{d}{dt} (G_{ai}) = J\alpha_i [p_a + p_c - p_i + \rho g (d_i - d_a)] \quad \text{Eq. 3.4}$$

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  $i$

- $J$  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 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:

```
AQUCHWAT
 1 0.0 'PRESSURE' 1.0 2.0E9 1 /
/
AQUANCON
 1 1 10 10 10 1 2 'J+' /
 1 10 10 1 10 1 2 'I+' /
/
```

## AQUCON

## Specifies connection data for numerical aquifers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

AQUCON connects a numerical aquifer (declared using the [AQUNUM](#) 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.

DEFAULT: 1.0

ECLIPSE 100 only

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 AQUNUM) or the aquifer connection cell area. A value of 0 uses the cross-sectional area specified in item 5 of AQUNUM, 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

ECLIPSE 100

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

ECLIPSE 100

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:

$$T = \frac{CDARCY}{(1/T_a) + (1/T_c)} \tag{Eq. 3.5}$$

where

$T_c$  = Component of transmissibility from the grid block

$T_a$  = Component of transmissibility from the aquifer

In the Cartesian case:

$$T_c = \frac{K_c \cdot A_c \cdot NTG}{D_c} \tag{Eq. 3.6}$$

where

$K_c$  = 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:

$$T_a = \frac{K_a \cdot A_a}{D_a} \quad \text{Eq. 3.7}$$

where

$K_a$  = Aquifer permeability from AQUNUM

$A_a$  = Aquifer area from AQUNUM

$D_a$  = Half the aquifer length from AQUNUM

If transmissibility option 1 (item 10) is used:

$$T_a = \frac{K_a \cdot A_c}{D_a} \quad \text{Eq. 3.8}$$

ECLIPSE 100

The NNCs that both connect the aquifer cells and connect the aquifer to the reservoir can be listed by using the ALLNNC mnemonic (24<sup>th</sup> control) in the RPTGRID keyword.

ECLIPSE 300

For this simulator, the NNCs can be output using the NNC mnemonic in the RPTGRID keyword.

In addition, an array with the name AQUIFERN 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  $2^{(N-1)}$  will appear in the 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 *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 NX=20, NY=20, NZ=10.

```
AQUCON
 2 1 3 1 20 1 10 'I-' 1.0 /
/
```

### 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 NDIVIX (specified in RUNSPEC keyword [DIMENS](#)) is 5.

```
AQUCON  
1 5 5 1 1 1 3 'I+' 1.0 /  
/
```

## AQUCT

## Specifies the property data for Carter-Tracy aquifers

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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), atma (LAB), atma (PVT-M)

4. Permeability of the aquifer

UNITS: mD (METRIC), mD (FIELD), mD (LAB), mD (PVT-M)

5. Porosity of the aquifer

UNITS: m<sup>3</sup>/m<sup>3</sup> (METRIC), ft<sup>3</sup>/ft<sup>3</sup> (FIELD), cc/cc (LAB), m<sup>3</sup>m<sup>3</sup> (PVT-M)

DEFAULT: 1.0

6. Total (rock + water) compressibility of the aquifer

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (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](#))

ECLIPSE 300

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](#). 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/sm<sup>3</sup> ([METRIC](#)), lb/stb ([FIELD](#)), gm/scc ([LAB](#)), kg/sm<sup>3</sup> ([PVT-M](#))

DEFAULT: 0.0

ECLIPSE 300

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 [AQUDIMS](#). 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.

ECLIPSE 100

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:

```

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+' /
/

```

## AQUCWFAC **Changes the property data for all constant head aquifers**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword can be used to change the datum depth and the datum pressures in **all** the aquifers defined using the [AQUCHWAT](#) 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:

```
AQUCWFAC
10.0 1.0 /
```

# AQUDIMS Dimensions for aquifers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

ECLIPSE 100 only

7. MXNALI The maximum number of aquifer lists.

DEFAULT: 0

ECLIPSE 100 only

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.

```
AQUUDIMS  
0 0 1 36 2 100 /
```

## AQUFET

## Specification data for Fetkovich aquifers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [AQUANCONL](#), instead of [AQUFET](#).

The number of analytic aquifers is specified using the [AQUDIMs](#) 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:

- Datum depth  
UNITS: m (METRIC), ft (FIELD), cm (LAB)
- 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)
- Initial volume of water in the aquifer  
UNITS: sm<sup>3</sup> (METRIC), stb (FIELD), scc (LAB)
- Total (rock + water) compressibility of the aquifer  
UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB)
- Aquifer productivity index  
(total influx rate per unit pressure difference)  
UNITS: sm<sup>3</sup>/day/bars (METRIC), stb/day/psi (FIELD), scc/hr/atm (LAB)
- Table number for water pressure properties  
(See keyword [PVTW](#))
- Lower I-location of connecting grid blocks
- Upper I-location of connecting grid blocks
- Lower J-location of connecting grid blocks
- Upper J-location of connecting grid blocks
- Lower K-location of connecting grid blocks
- Upper K-location of connecting grid blocks
- 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 face

'J-' J = 1 face

'J+' J = NDIVIY face

'K-' K = 1 face (top)

'K+' K = NDIVIZ face (bottom)

#### 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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (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 [AQUDIMS](#) 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 [AQUFETP](#), [AQUCT](#), [AQUANCON](#) or [AQANCONL](#)

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> ([METRIC](#)), stb ([FIELD](#)), scc ([LAB](#)), sm<sup>3</sup> ([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<sup>3</sup>/day/bars ([METRIC](#)), stb/day/psi ([FIELD](#)), scc/hr/atm ([LAB](#)), sm<sup>3</sup>/day/atm ([PVT-M](#))

7. 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.

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<sup>3</sup> ([METRIC](#)), lb/stb ([FIELD](#)), gm/scc ([LAB](#)), kg/sm<sup>3</sup> ([PVT-M](#))

DEFAULT: 0.0

*ECLIPSE 300*

9. 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 (/).

## ECLIPSE 100

**Note:** The slash could be omitted for ECLIPSE 100, especially if the number of records entered using the [AQUFETP](#) 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.

Connections to a Fetkovich 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 Fetkovich aquifer.

## ECLIPSE 100

This keyword [AQUFETP](#) cannot be used together with the keyword [AQUFET](#).

The aquifer properties can be modified at any time during the simulation by entering the [AQUFETP](#) keyword again in the [SCHEDULE](#) section.

## 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 or 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.

See also "[Aquifer modeling facilities](#)" in the *ECLIPSE Technical Description* for further information.

**Note:** [AQUFETP](#) must not be used in the [SOLUTION](#) section of restarted runs, as the aquifer specification data cannot be changed.

## Example

One Fetkovich aquifer, connected to two vertical sides of a 10 x 10 x 2 grid:

```

AQUFETP
 1 7000.0 4000.0 2.0E9 1.0E-5 500.0 1 /
/
AQUANCON
 1 1 10 10 10 1 2 'J+' /

```

```
1 10 10 1 10 1 2 'I+' /  
/
```

# AQUFLUX

## Specifies a constant flux aquifer

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB),  $\text{sm}^3/\text{day}/\text{m}^2$  (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:  $\text{kg}/\text{sm}^3$  (METRIC),  $\text{lb}/\text{stb}$  (FIELD),  $\text{gm}/\text{scc}$  (LAB),  $\text{kg}/\text{sm}^3$  (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), atma (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 `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 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:

$$Q_w = F_w \cdot A \cdot M \quad \text{Eq. 3.9}$$

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 `AQUANCONL` 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 `AQUANCONL` keyword.

The aquifer flux can be modified at any time during the simulation by reentering the `AQUFLUX` keyword in the `SCHEDULE` section.

*ECLIPSE 100 only*

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

```

AQUFLUX
1 0.0003 /
2 0.0004 /
/

AQUANCON
-- Aquifer 1
1 1 10 10 10 10 1 4 'J+' /
1 10 10 6 10 1 4 'I+' /
--Aquifer 2
2 10 10 6 10 1 4 'I+' /
/

```

## AQUNNC

## Set non-neighbor connection values explicitly for numerical aquifers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.rm<sup>3</sup>/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

```
AQUNNC
-- 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 /
/
```

## AQUNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 AQUNUM 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 AQUNUM keyword must be less than or equal to the value set by the 1<sup>st</sup> item in the [AQUDIMS](#) 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 AQUNUM are as follows:

1. Aquifer identification number

This should be a number between 1 and the maximum number of aquifers, set in the keyword [AQUDIMS](#).

2. I-coordinate of grid block to represent aquifer
3. J-coordinate of grid block to represent aquifer
4. K-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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (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: m<sup>3</sup>/m<sup>3</sup> (METRIC), ft<sup>3</sup>/ft<sup>3</sup> (FIELD), cc/cc (LAB), m<sup>3</sup>/m<sup>3</sup> (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.

UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

#### 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)

ECLIPSE 100 only

#### 11. PVT table number for aquifer

DEFAULT: Block `PVTNUM` value.

ECLIPSE 100 only

#### 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

$$PORV = Porosity \cdot Length \cdot Area \quad \text{Eq. 3.10}$$

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 /
/

```

## AQUTAB

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

## ASPDEPO

## Asphaltene deposition data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is set to provide data for the asphaltene deposition process. The required data consists of:

1. A static deposition or adsorption coefficient  $\alpha$   
UNITS: 1/day (METRIC), 1/day (FIELD), 1/hr (LAB), 1/day (PVT-M)
2. A plugging coefficient  $\gamma_1$ , used in the pre-2011.1 deposition model  
UNITS: 1/m<sup>3</sup> (METRIC), 1/ft<sup>3</sup> (FIELD), 1/cc (LAB), 1/m<sup>3</sup> (PVT-M)
3. An entrainment coefficient  $\beta$   
UNITS: 1/m (METRIC), 1/ft (FIELD), 1/cm (LAB), 1/m (PVT-M)
4. A critical velocity  $U_{cr}$  above which entrainment takes place  
UNITS: m/day (METRIC), ft/day (FIELD), cm/hr (LAB), m/day (PVT-M)
5. A plugging coefficient  $\gamma_2$  used in the three-component deposition model  
UNITS: 1/m (METRIC), 1/ft (FIELD), 1/cm (LAB), 1/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

```

ASPDEPO
-- adsorp      plug1      entrain      Vcr      plug2
   0.001        0          1.e-5        0.01     0.1/

```

## ASPFLOC

## Asphaltene flocculation components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $1, 2, \dots, i, i+1, \dots, j, \dots, k, \dots, N$ , 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:

```

ASPFLOC
-- cplast  cplast  cpflocs
   4       6      7 /

```

## ASPFLRT

**Asphaltene flocculation/dissociation rates**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [ASPFLOC](#) 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
-- floc rates      followed by dissociation rates
  0.010  0.05    0.10
  0.005  0.04    0.08 /
```

## ASPHALTE Asphaltene option

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

Asphaltene default: weight, no permeability damage and no viscosity change:

```
ASPHALTE
-- charact. crit Perm damage Viscosity model
/
```

## ASPKDAM Asphaltene permeability damage

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [SWFN](#), [SOF3](#) or [SWOF](#)). Interpolation depends on the wettability factor  $F$  which is given by the [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 [TABDIMS](#) keyword), and the maximum number of rows in each table is set by item 3 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

```

ASPKROW
-- Sw   krw   kro
  0.00  0.0   1.0
  0.50  0.31  0.7
  1.00  0.95  0.0/

```

## ASPLCRT

## Asphaltene plugging control

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\varepsilon_{cr} = \phi \cdot S_{cr}$  is required here.

2. A critical asphaltene flocs molar concentration above which plugging can happen.

UNITS: kg-M/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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      /
```

## ASPP1P

## Asphaltene one parameter precipitation

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
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:

```
ASPP1P
-- concentration 2nd component
'Z' 2 /
```

## ASPP2P

## Asphaltene two parameters precipitation

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

## Asphaltene two parameters precipitation data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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. /
/

```

## ASPREWG Asphaltene as percentage weight

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```

ASPP1P
-- pressure
  'P' /
ASPREWG
-- pressure      % weight
   3000          30.0
   3500          70.0
   4000         100.0 /

```

#### 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.

```

ASPP1P
-- molar fraction 2nd component
  'Z' 2 /
ASPREWG
-- Z2      % weight
  0.001    100.0

```

0.005	70.0
0.010	10.0 /

## ASPVISO

## Asphaltene oil viscosity change

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ASPWETF Asphaltene wettability factor data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\varepsilon = \phi \cdot S_s$  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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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. I1 Lower I coordinate of the box.
2. I2 Upper I coordinate of the box.
3. J1 Lower J coordinate of the box.
4. J2 Upper J coordinate of the box.
5. K1 Lower K coordinate of the box.
6. K2 Upper K coordinate of the box.
7. NX Number of coarsened cells along the X direction in this box.  
NX must be an exact factor of I2-I1+1.
8. NY Number of coarsened cells along the Y direction in this box.  
NY must be an exact factor of J2-J1+1.
9. NZ Number of coarsened cells along the Z direction in this box.  
NZ must be an exact factor of K2-K1+1.

This divides the box I1-I2, J1-J2, K1-K2 into  $NX * NY * NZ$  coarse cells. Coarsening boxes specified using COARSEN or AUTOCOAR must not overlap one another, but they may join together.

The parameter 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

```
AUTOCOAR
-- 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.

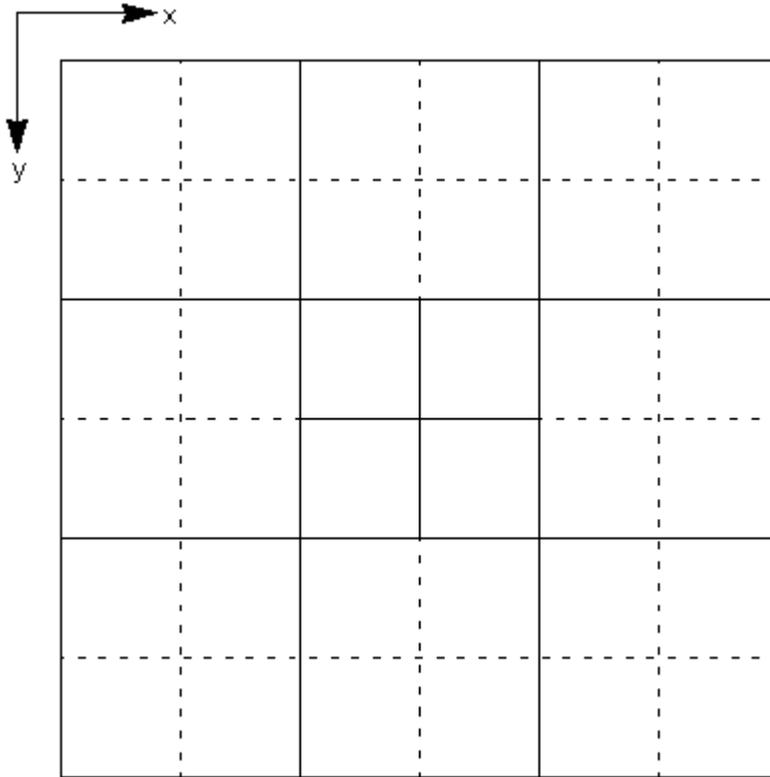


Figure 3.1. The effect of the COARSEN keyword

Now with

```
AUTOREF
3 3 1 /
```

the grid is refined as shown in figure 3.2.

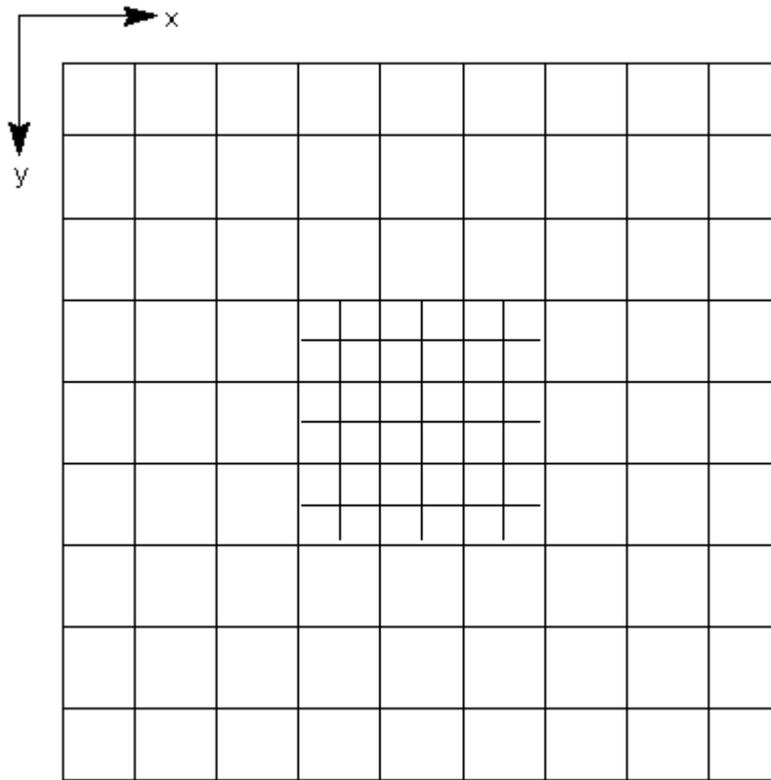


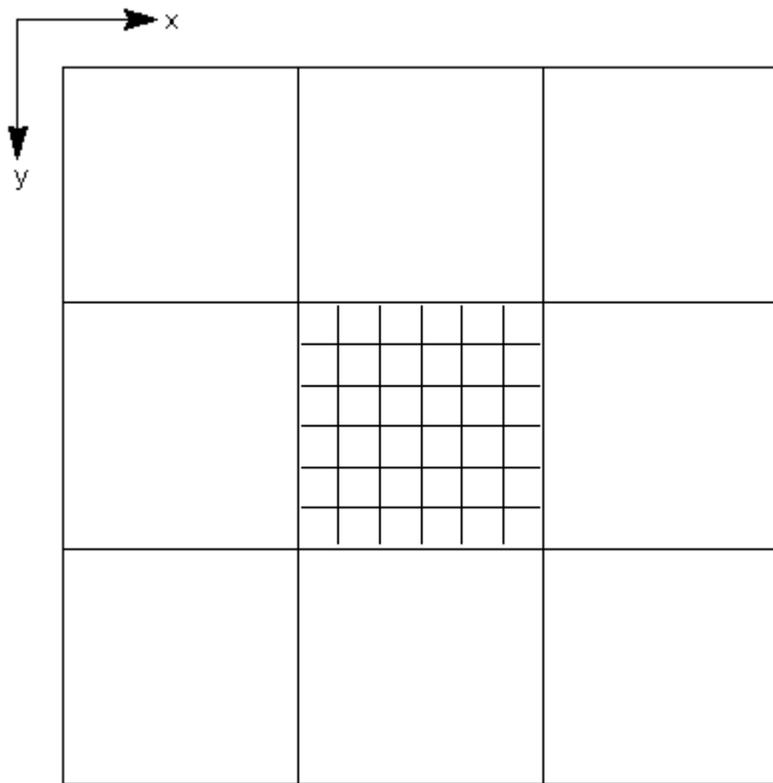
Figure 3.2. The effect of the AUTOREF keyword

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 AUTO COAR keyword*

## AUTOREF Set options for auto refinement

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## AUTOSAVE Specify automatic writing of SAVE files

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

## **B**

This section contains an alphabetic listing of keywords starting with the letter B.

## BAKER1

## Request Baker three-phase relative permeability model (General)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

```
BAKER1
WATER GAS/
```

#### Example 2

Example 2 shows how to use the BAKER1 model for gas and oil three-phase relative permeability.

```
BAKER1  
GAS OIL /
```

### 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)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

```
BAKER2  
WATER GAS OIL/
```

Example 3 describes the [BAKER1](#) model for water and oil three-phase relative permeability, and BAKER2 for gas three-phase relative permeability.

```
BAKER1  
WATER OIL/  
  
BAKER2  
GAS/
```

## BDENSITY **Brine surface density**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

See also keywords [PVTWSALT](#) and [DENSITY](#).

### Example

With NTPVT =2 and NPPVT ≥ 4 in [TABDIMS](#):

```

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 /

```

## BGGI

## Variation of saturated gas FVF with pressure and $G_i$

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $G_i$  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,  $G_i$ ) function which multiplies the saturated gas FVF value at (P,  $G_i=0$ ) to give the true saturated gas FVF at (P,  $G_i$ ). The saturated gas FVF value at (P,  $G_i=0$ ) 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  $G_i$  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,  $G_i=0$ ) (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  $G_i$  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

```
BGGI
100.0  1.0
        0.99  /
200.0  1.0
        0.992 /
/
```

## BIC

## Binary interaction coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## BICAQ1

## First binary interaction coefficient for aqueous phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$k_{jw}^a = bq_1 + bq_2T_{rj} + bq_3T_{rj}^2 \quad \text{Eq. 3.11}$$

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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## BICS

**Binary interaction coefficients for surface**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

---

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

*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

## Biot's constant for rock/fluid interaction

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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       $\sigma_{1,1} \quad \sigma_{1,2} \quad \dots \quad \sigma_{1,NGMBTC}$

The next NGBMTR 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       $\sigma_{3,1} \quad BM_{1,1} \quad \dots \quad BM_{1,NGMBTC}$

Record 3       $\sigma_{3,2} \quad BM_{2,1} \quad \dots \quad BM_{2,NGMBTC}$

... ..

... ..

NGBMTR+1     $\sigma_{3,NGBMTR} \quad BM_{NGBMTR,1} \quad \dots \quad BM_{NGBMTR,NGMBTC}$

Records 2 through NGBMTR 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, NGBMTR, 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, NGBMTR=3:

```
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 /
/
```

## BLACKOIL Requests black oil mode

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 $G_i$

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $G_i$  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,  $G_i$ ) function which multiplies the saturated oil FVF value at (P,  $G_i=0$ ) to give the true saturated oil FVF at (P,  $G_i$ ). The saturated oil FVF value at (P,  $G_i=0$ ) 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 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  $G_i$  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 FVF at (P,  $G_i=0$ ) (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  $G_i$  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

```
BOGI
100.0  1.0
        0.97  /
200.0  1.0
        0.952  /
/
```

## BOUNDARY Defines area of grid to be printed

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

- ECLIPSE 100 only*
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 /
```

## BOX

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## Redefines the current input box

The keyword should be followed by six integers that redefine the current input box. Subsequent operations using [EQUALS](#), [ADD](#), [MULTIPLY](#) and [COPY](#) only alter grid blocks within the current input box. Similarly, data read into an array (for example, using [PERMX](#), [PORV](#) or [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 (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:

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword } \text{DIMENS})$$

The default cell limits are the current BOX values.

The data should be terminated by with a slash (/).

Note that at the end of each input section, the program effectively supplies an [ENDBOX](#) and resets the input box.

This keyword can be used in the RUNSPEC and [SUMMARY](#) sections.

See also keywords [ADD](#), [COPY](#), [ENDBOX](#), [EQUALS](#), [MULTIPLY](#), [MINVALUE](#) and [MAXVALUE](#).

### Example

```

-- IX1-IX2  JY1-JY2  KZ1-KZ2
BOX
   3   7   4  13   3   5 /

```

*ECLIPSE 300 only*

## BPARA

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Block Parallel license

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

```

BPARA
PARALLEL
--NDMAIN MACHINE TYPE
      8      DISTRIBUTED /

```

## BPIDIMS

## Dimensions for interpolated block quantities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#)).

*ECLIPSE 300*

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.

*ECLIPSE 300*

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.

4. Artificial Lift Quantity (ALQ) used in the pressure loss calculations for the branch.

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

```

BRANPROP
-- DnTre   UpTre   VFP   ALQ   ALQ=
-- node   node   table  value  dens?
  GB2     N4         3 /
  GA2     N3         2 /
  GB1     N2         1 /
  GA1     N1       9999 /
  N4      N3         4 /
  N3      N2         4 /
  N2      N1         5 /
  N1      FIELD    6   100 /
/

```

## BRINE

### Brine tracking is required

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Salt name	Chemical formula
Sodium Chloride	NaCl
Potassium Chloride	KCl
Calcium Chloride	CaCl <sub>2</sub>
Magnesium Chloride	MgCl <sub>2</sub>
Sodium Carbonate	Na <sub>2</sub> CO <sub>3</sub>
Potassium Carbonate	K <sub>2</sub> CO <sub>3</sub>
Calcium Carbonate	CaCO <sub>3</sub>
Magnesium Carbonate	MgCO <sub>3</sub>
Sodium Sulfate	Na <sub>2</sub> SO <sub>4</sub>
Potassium Sulfate	K <sub>2</sub> SO <sub>4</sub>
Calcium Sulfate	CaSO <sub>4</sub>
Magnesium Sulfate	MgSO <sub>4</sub>

Table 3.2: List of available salts in the Multi-Component Brine model

### Example

ECLIPSE 100 only

Multi-component brine model with three salts.

```
BRINE
NACL  CACL2  MGCO3  /
```

## BTOBALFA Dual porosity block to block matrix-fracture coupling multiplier

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
X	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
X	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 0.5 /
```

## **C**

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $\text{kJ}/\text{sm}^3$  (METRIC),  $\text{Btu}/\text{Mscf}$  (FIELD),  $\text{J}/\text{scc}$  (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 /
```

## CALVAL

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
--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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
4. J1 Lower J-coordinate of the box in the parent grid.
5. J2 Upper J-coordinate of the box in the parent grid.
6. K1 Lower K-coordinate of the box in the parent grid.
7. K2 Upper K-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.
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 only*

*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

```

## CART

### 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## CATYPE

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

- Use the Langmuir scaling value for the volumetric strain term in the Palmer-Mansoori model ([ROCKPAMA](#) or [ROCKPAME](#)).

NOPMSCAL or PMSCAL

DEFAULT: PMSCAL

- 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*

- Pressure chop if passing through the rebound pressure.

PMPCHP or NOPMPCHP

DEFAULT: NOPMPCHP

- Minimum pore volume of the Palmer-Mansoori method ([ROCKPAMA](#) or [ROCKPAME](#)).

DEFAULT: 5.0D-6

## Example

```
CBMOPTS
-- model water-flow
INSTANT NO/
```

## CCTYPE

## Component compressibility type

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$V_c = [1 - c_{P_c} (P - P_{ref_c})] \cdot [1 + C_{T1_c} (T - T_{ref_c})] \cdot MW_c / \rho_{ref_c} \quad \text{Eq. 3.12}$$

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:

$$V_c = Z_c RT / P \quad \text{Eq. 3.13}$$

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

DEFAULT: No limit

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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](#))

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:  $\text{sm}^3/\text{day}$  (METRIC), stb/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 [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.

ECLIPSE 100

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  2  1* 3.5 / GOR limit for layer-2 connections
  /                               in all wells with names beginning with P
```

## CECONT

## Tracer economic limits for production well connections

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
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

```
CECONT
  PROD4 4* CON NO /
TR5 1000.0 0.5 /
TR1 2* 1* 0.8 2* /
TR3 1500.0 0.1 /
/
/
```

### Example 3

```
CECONT
  PRODUCER 8 9 1 1 CON+ NO /
BRI 1000.0 0.1 /
/
/
```

## CFLLIMIT

## Stable timestepping

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- Turns on (or off) the stable timestepping option.  
ON Stable timestepping turned on.  
OFF Stable timestepping turned off.  
DEFAULT: OFF
- 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.

---

- In AIM runs only, the CFL value above which cells will be made implicit (subject to item 5)  
DEFAULT: 0.75
- 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
- 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.

---

- 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
- An integer which controls the implicit treatment of the wells.
  - Wells are not given any special treatment.
  - All the inner solver direction lines of cells containing the completions of open wells are set to be implicit.
  - 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*/
```

## CGDTYPE

## Component gas density type

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component
NO	Not used. The molar volume for this component is computed as if this keyword were not present.
EMIX	An equation of state is used for the EMIX components. The vapor root of the cubic equation of state is always used.
EGFE	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.

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 /
```

## CGVTYPE

## Component gas viscosity type

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component
NO	Not used. The viscosity for this component is computed as if this keyword was not present.
LMIX	The LBC viscosity model is applied with these components' gas phase molar density.
LEMIX	The LBC viscosity model is applied on the LEMIX components densities where an EoS is used to compute the phase molar density.

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 /
```

## CHANDIMS **Increase well dimensions**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- The maximum number of wells  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of completions per well.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of groups.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of wells in a group, or child groups in a group.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of stages per separator including the last, which represents the stock tank.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of wellstreams.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of mixtures.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of separator conditions.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- The maximum number of mixture items.  
DEFAULT: Same as the value set in [WELLDIMS](#)
- Item not used.
- 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

```
CIRCLE
```

## CNAMES

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 CO<sub>2</sub> to dissolve in the aqueous phase**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword turns on the CO<sub>2</sub> Solution option. The CO<sub>2</sub> 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:

[SOLUBILI](#) This allows values in the default solubility table to be entered by the user.

[SOLUBILS](#) This allows the user to enter a solubility table with salt dependencies.

[SOLUBILT](#) 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 CO<sub>2</sub> gas-water ratio versus depth.

[RSW](#) This specifies the initial variation of aqueous CO<sub>2</sub> gas-water ratio by enumeration.

[AMF](#) This specifies the initial composition of aqueous CO<sub>2</sub> system.

For further information see "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

## CO2STORE **CO<sub>2</sub> storage option**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword turns on the CO<sub>2</sub> storage option. The mutual solubility and fluid properties of CO<sub>2</sub> and H<sub>2</sub>O are considered, where the H<sub>2</sub>O rich phase might include salts.

CO2STORE is the only keyword required for this option.

For further information see "[CO<sub>2</sub> storage](#)" in the *ECLIPSE Technical Description*.

## COAL

### 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## COALADS

## Gas/solvent relative adsorption data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\text{Surface concentration of gas} = C_g(P) \cdot F_g(f_g)$$

$$\text{Surface concentration of solvent} = C_s(P) \cdot F_s(f_g)$$

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 ( $= S_g / (S_g + S_{\text{solv}})$ )

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  $\geq 4$ .

COALADS	Fg	Fs
--		
0.0	0.0	1.0
1.0	1.0	0.0
/		
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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
/
```

## Example 2

In this example we assume that the instant desorption model is used, see [CBMOPTS](#). 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $S_{\text{gas}} / (S_{\text{gas}} + S_{\text{solvent}})$ . 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:

$$\text{Surface concentration of gas} = C_g(P \cdot F_g(f_g))$$

$$\text{Surface concentration of solvent} = C_s(P \cdot F_s(f_g))$$

where

$C_g(P \cdot F_g(f_g))$  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 NTSFUN (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.

```
COALPP
--      Fg   Fs
   0.0  0.0  1.0
   1.0  1.0  0.0
/
```

## COARSEN Specifies a box of cells for grid coarsening

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$1 \leq I1 \leq I2 \leq NX$$

$$1 \leq J1 \leq J2 \leq NY$$

$$1 \leq K1 \leq K2 \leq NZ.$$

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.

ECLIPSE 100

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  /
/
```

## CODTYPE

## Component oil density type

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component
NO	Not used. The molar volume for this component is computed as if this keyword was not present.
EMIX	An equation of state is used for the EMIX components. The liquid root of the cubic equation of state is always used.
EGFE	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.

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:

```
CODTYPE
EMIX EMIX EMIX EMIX /
```

## COLLAPSE Identifies collapsible cells in the compressed VE option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
- $PERMZ \leq$  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
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**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

COMPAGH is used to set the gravity head for a completion. The pressure at each completion,  $P_{comp}$ , is calculated by:

$$P_{comp} = P_{BH} + AGH \cdot (D_{comp} - D_{BH}) \quad \text{Eq. 3.14}$$

where:

$P_{BH}$  Pressure at bottom hole reference depth

$AGH$  Average gravity head between  $D_{comp}$  and  $D_{BH}$

$$AGH = \left( \int_{D_{comp}}^{D_{BH}} \rho(\delta) g d\delta \right) / \left( \int_{D_{comp}}^{D_{BH}} d\delta \right) \quad \text{Eq. 3.15}$$

$D_{comp}$  Depth of completion

$D_{BH}$  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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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,  $P_{comp}$ , is calculated by:

$$P_{comp} = P_{BH} + AGH \cdot (D_{comp} - D_{BH}) \quad \text{Eq. 3.16}$$

where

$P_{BH}$  Pressure at bottom hole reference depth

$AGH$  Average gravity head between  $D_{comp}$  and  $D_{BH}$ ,

$$AGH = \left( \int_{D_{comp}}^{D_{BH}} \rho(\delta) g d\delta \right) / \left( \int_{D_{comp}}^{D_{BH}} d\delta \right) \quad \text{Eq. 3.17}$$

$D_{comp}$  Depth of completion

$D_{BH}$  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 will override those values calculated internally by ECLIPSE 300, based on completion inflows (see item 13 of keyword [WELSPECL](#)).

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 /  
/
```

## COMPDAT Well completion specification data

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

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<sup>3</sup>/day/bars (`METRIC`), cP.rb/day/psi (`FIELD`), cP.rcc/hr/atm (`LAB`), cP.rm<sup>3</sup>/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 [WELSPEC3](#))
- 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<sup>3</sup> (METRIC), day/Mscf (FIELD), hr/scc (LAB), day/sm<sup>3</sup> (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

ECLIPSE 300 only

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  $r_o$  from Peaceman's formula

## Notes

ECLIPSE 100

- 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 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 0 1*  0.333 3* X /
PROD3  4 5 2 3 OPEN 0 1*  0.333 3* X /
'INJ*'  0 0 3 3 OPEN 0 2.3 /
/

```

## COMPDATL Completion data for wells in local grids

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

*ECLIPSE 100*

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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 ( $I = \text{NDIVIX}$  , 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 [WELSPEDS](#) 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.

ECLIPSE 100

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.

ECLIPSE 100

UNITS: m ([METRIC](#)), ft ([FIELD](#)), cm ([LAB](#)), m ([PVT-M](#))

ECLIPSE 300

DEFAULT: 1 ft = 0.3048 m

ECLIPSE 100

DEFAULT: Undefined.

## 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/sm<sup>3</sup> (METRIC), day/Mscf (FIELD), hr/scc (LAB), day/sm<sup>3</sup> (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

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 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

ECLIPSE 100

- 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

```

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  /
/

```

## COMPDATM **Completion data for wells in amalgamated local refined grids**

COMPDATM is an alias for [COMPDATL](#).

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## COMPFLSH Flash transformation ratios for well connections

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$F_1 = \frac{B_{ob}(\text{flash}) - 1}{B_{ob}(\text{diff}) - 1} \quad \text{Eq. 3.18}$$

DEFAULT: 1.0

7. Transformation ratio ( $F_2$ ) for solution gas - oil ratio, where

$$F_2 = \frac{R_s(\text{flash})}{R_s(\text{diff})} \quad \text{Eq. 3.19}$$

DEFAULT: 1.0

8. 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 `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 `PVCO` or `PVTO`) will be used in the calculation. The other PVT property keywords (for example `PVTW`, `PVTG`, `DENSITY`, `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 `PVTNUM`.

## Example

```

COMPFLSH
-- 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:

$$F_1 = \frac{B_{ob}(\text{flash}) - 1}{B_{ob}(\text{diff}) - 1} \quad F_2 = \frac{R_s(\text{flash})}{R_s(\text{diff})} \quad \text{Eq. 3.20}$$

where

$B_{ob}$  is the formation volume factor of oil at bubble point pressure, and

$R_G$  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:

$$\frac{B_o \text{ (flash)}}{B_o \text{ (diff)}} = \frac{B_{ob} \text{ (flash)}}{B_{ob} \text{ (diff)}} = F_1 + \frac{1-F_1}{B_{ob} \text{ (diff)}} \quad \text{Eq. 3.21}$$

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  $Q_o^*$  and  $Q_g^*$ ,

where

$$Q_o^* = T_{\text{oil-oil}} Q_o$$

$$Q_g^* = T_{\text{gas-oil}} Q_o + Q_g$$

with the transformation coefficients

$$T_{\text{(oil-oil)}} = \frac{1}{F_1 + \left( \frac{1-F_1}{B_{ob} \text{ (diff)}} \right)} \quad \text{Eq. 3.22}$$

$$T_{\text{(gas-oil)}} = (T_{\text{(oil-oil)}} F_2 - 1) R_s \text{ (diff)} \quad \text{Eq. 3.23}$$

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  $B_o$  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  $B_{ob}$  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

## Imbibition table numbers for well connections

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /  
/
```

## COMPINJK User-defined injection well relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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*),

$$M(p, j) = \frac{\left[ \frac{k_r(o, j)}{\mu(o, j)} + \frac{k_r(w, j)}{\mu(w, j)} + \frac{k_r(g, j)}{\mu(g, j)} \right]}{B(p, j)} \quad \text{Eq. 3.24}$$

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

$$M(p, j) = \frac{k_r(p, *)}{B(p, j)\mu(p, j)} \quad \text{Eq. 3.25}$$

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:

$k_{rw}(S_w = 1 - S_{owcr} - S_{gco})$  for water injectors

$k_{rg}(S_g = 1 - S_{ogcr} - S_{wco})$  for gas injectors

$k_{ro}(S_w = S_{wco}, S_g = S_{gco})$  for oil injectors.

If a default or negative value is supplied, the well must already have been declared an injector with keyword `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 `COMPINJK`.

---

**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 YES (item 10 in keyword `WELSPECS`), its flag will be reset to NO and a warning message is printed.

---

## Example

This example causes all connections in well INJ1 to have a constant injected phase relative permeability set to the appropriate saturation table end point. The connection in layer 3 of well INJ2 has an injected phase relative permeability of 1.0.

```
COMPINJK
  INJ1      /
  INJ2  2* 3 3 1.0 /
  /
```

## COMPKRI

## Relative permeabilities for injector

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

COMPKRI is used to set the phase relative permeabilities for an injection well completion. The total mobility is

$$\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g} \quad \text{Eq. 3.26}$$

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 is 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
'WINJ' 4* 0.0 0.0 0.8 /
'CYCLIC' 4* 0.4 0.0 0.2 /
/
```

## COMPKRIL Relative permeabilities for injector completions in local refined grids

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g} \quad \text{Eq. 3.27}$$

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. K<sub>rw</sub>, 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 COMPKR1 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 is 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 COMPKR1. 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 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

```
COMPKR1L
CYCLIC LGR1 4* 0.4 0.2 0.1 /
/
```

## COMPLMPL Lumps connections in local grids for automatic workovers

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 first have 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, [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.

### ECLIPSE 100

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.

---

### ECLIPSE 100

---

**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 [WELOPENL](#), [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

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](#).

ECLIPSE 100

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.

ECLIPSE 100

---

**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

```

PROD2 CARF3 2\* 5 5 3 /  
/

## COMPLUMP Lumps connections for automatic workovers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 100

---

**Note:** It is still possible to use keyword `COMPLUMP` for wells completed in a single, non-amalgamated local grid.

---

## Example

```

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  /
/

```

## COMPMBIL Voidage mobility for injector completions in local refined grids

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g} \quad \text{Eq. 3.28}$$

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<sup>-1</sup> (METRIC), cP<sup>-1</sup> (FIELD), cP<sup>-1</sup> (LAB), cP<sup>-1</sup> (PVT-M)

DEFAULT: 10.0 cP<sup>-1</sup>

## Example

```
COMPMBIL  
ICYC5 LGR1 2 2 1 3 10.0 /  
/
```

## COMPMOBI Voidage mobility for injector

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

COMPMOBI is used to set the total mobility for an injection completion. The total mobility is

$$\frac{k_{rw}}{\mu_w} + \frac{k_{ro}}{\mu_o} + \frac{k_{rg}}{\mu_g} \quad \text{Eq. 3.29}$$

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<sup>-1</sup> (METRIC), cP<sup>-1</sup> (FIELD), cP<sup>-1</sup> (LAB), cP<sup>-1</sup> (PVT-M)

DEFAULT: 1.0 cP<sup>-1</sup>

### Example

```
COMPMOBI
ICYC5 18 6 8 23 10.0 /
/
```

## COMPOFF **Switches off automatic compressors in the network**

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WRFPTPLT](#)) 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

```
COMPORD  
HWELL1    INPUT /  
HWELL2    INPUT /  
/
```

## COMPRIV

**Specifies which grid blocks connect to a river**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## COMPRP

## Data for rescaling saturations at well connections

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 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.

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,  $SS_w$  and  $SS_g$ ,

$$k_{rw} = k_{rw}(SS_w) \quad \text{Eq. 3.30}$$

$$k_{rg} = k_{rg}(SS_g) \quad \text{Eq. 3.31}$$

$$k_{ro} = k_{ro}(SS_w, SS_g) \quad \text{Eq. 3.32}$$

**Water saturation scaling**

In three-phase and oil-water problems, the water saturation is scaled as follows:

If

$$SWMIN < S_w < SWMAX \quad \text{Eq. 3.33}$$

Then

$$SS_w = (1-f_w)S_{wco} + f_w(1-S_{owcr})$$

where

$$f_w = (S_w - SWMIN) / (SWMAX - SWMIN)$$

If

$$S_w < SWMIN \quad \text{Eq. 3.34}$$

If

$$S_w > S_{wco},$$

$$\text{Then } SS_w = S_{wco}$$

$$\text{If } S_w < S_{wco},$$

$$\text{Then } SS_w = \frac{(S_w S_{wco})}{\text{MIN}(S_{wco}, SWMIN)}$$

If

$$S_w > SWMAX \quad \text{Eq. 3.35}$$

$$\text{If } S_w < 1 - S_{owcr},$$

$$\text{Then } SS_w = 1 - S_{owcr}$$

$$\text{If } S_w > 1 - S_{owcr},$$

$$\text{Then } 1 - SS_w = \frac{(1 - S_w) S_{owcr}}{\text{MIN}(S_{owcr}, \{1 - SWMAX\})}$$

ECLIPSE 100

In gas-water problems, the water saturation is scaled as follows:

If

$$SWMIN < S_w < SWMAX \quad \text{Eq. 3.36}$$

$$\text{Then } SS_w = (1-f_w)S_{wco} + f_w(1-S_{gcr})$$

$$\text{Where } f_w = \frac{S_w - SWMIN}{SWMAX - SWMIN}$$

If

$$S_w < SWMIN \quad \text{Eq. 3.37}$$

$$\begin{aligned}
&\text{If } S_w > S_{wco}, \\
&\text{Then } SS_w = S_{wco} \\
&\text{If } S_w < S_{wco}, \\
&\text{Then } SS_w = \frac{S_w S_{wco}}{\text{MIN}(S_{wco}, \text{SWMIN})} \\
&\text{If} \\
&S_w > \text{SWMAX} \tag{Eq. 3.38} \\
&\text{If } S_w < 1 - S_{gcr}, \\
&\text{Then } SS_w = 1 - S_{gcr} \\
&\text{If } S_w > 1 - S_{gcr}, \\
&\text{Then } 1 - SS_w = \frac{(1 - S_w) S_{gcr}}{\text{MIN}(S_{gcr}, \{1 - \text{SWMAX}\})}
\end{aligned}$$

### Gas saturation scaling

In three-phase and oil-gas problems, the gas saturation is scaled as follows:

$$\begin{aligned}
&\text{If} \\
&\text{SGMIN} < S_g < \text{SGMAX} \tag{Eq. 3.39} \\
&\text{Then } SS_g = (1 - f_g) S_{gcr} + f_g (1 - S_{ogcr} - S_{wco}) \\
&\text{Where } f_g = \frac{(S_g - \text{SGMIN})}{\text{SGMAX} - \text{SGMIN}}
\end{aligned}$$

$$\begin{aligned}
&\text{If} \\
&S_g < \text{SGMIN} \tag{Eq. 3.40} \\
&\text{If } S_g > S_{gcr}, \\
&\text{Then } SS_g = S_{gcr} \\
&\text{If } S_g < S_{gcr}, \\
&\text{Then } SS_g = \frac{S_g S_{gcr}}{\text{MIN}(S_{gcr}, \text{SGMIN})}
\end{aligned}$$

$$\begin{aligned}
&\text{If} \\
&S_g > \text{SGMAX} \tag{Eq. 3.41} \\
&\text{If } S_g < 1 - S_{ogcr} - S_{wco}, \\
&\text{Then } SS_g = 1 - S_{ogcr} - S_{wco} \\
&\text{If } S_g > 1 - S_{ogcr} - S_{wco},
\end{aligned}$$

$$\text{Then } 1-SS_g = \frac{(1-S_g)(S_{ogcr} + S_{wco})}{\text{MIN}(\{S_{ogcr} + S_{wco}\}, \{1-SGMAX\})}$$

In gas-water problems, the scaled gas saturation is given by

$$SS_g = 1-SS_w \quad \text{Eq. 3.42}$$

## 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  $S_w = 1-S_{owcr}$ . 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,

$$SWMIN = l_b(1-S_{owcr}) + (1-l_b)S_{wco} \quad \text{Eq. 3.43}$$

and SWMAX is the water saturation in the grid block when the contact reaches the top of the perforations,

$$SWMAX = (1-l_t)(1-S_{owcr}) + l_t S_{wco} \quad \text{Eq. 3.44}$$

Now consider the grid block when it has instead gas encroaching from the top. Above the gas oil contact the gas saturation is  $1-S_{ogcr} - S_{wco}$ . 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,

$$SGMIN = l_t(1-S_{ogcr} - S_{wco}) + (1-l_t)S_{gcr} \quad \text{Eq. 3.45}$$

and SGMAX is the gas saturation in the grid block when the contact reaches the bottom of the perforations,

$$SGMAX = (1-l_b)(1-S_{ogcr} - S_{wco}) + l_b S_{gcr} \quad \text{Eq. 3.46}$$

In gas-water problems where water encroaches from the bottom, the scaled water saturation end points are

$$SWMIN = l_b(1-S_{gcr}) + (1-l_b)S_{wco} \quad \text{Eq. 3.47}$$

$$SWMAX = (1-l_t)(1-S_{gcr}) + l_t S_{wco}$$

## COMPRPL Scaled saturations for local grid well connections

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- 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](#).
- 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](#).
- I - location of connecting grid block(s)
 

DEFAULT: 0 (allows any I-value)
- J - location of connecting grid block(s)
 

DEFAULT: 0 (allows any J-value)
- K - location of upper connecting block in this set of data
 

DEFAULT: 0 (corresponds to top connection of well)
- 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.
- 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

ECLIPSE 100

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 [WELSPECL](#).

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.

```

COMPSEGL
-- Name
  PROD /
-- Local I J K Brn Start End Dirn End Connection

```

--	Grid			No	Length	Length	Penet	Range	Depth
--	Top Branch								
	UPPER	2	3	3	2	30	1*	X	10 /
--	Bottom Branch								
	LOWER	2	3	3	3	230	1*	X	6 /
/									

## COMPSEGS Defines location of completions in a multisegment well

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

### ECLIPSE 100

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

### ECLIPSE 300

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*).

ECLIPSE 100

See also [item 63](#) of the `OPTIONS` keyword.

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
-- Name
  PROD /
-- I J K Brn Start End Dirn End Connection
--      No Length Length Penet Range Depth
-- Top Branch
  2 5 2 2 30 1* X 6 /
-- Middle Branch
  2 5 5 3 170 1* X 6 /
-- Bottom Branch
  2 5 8 4 230 1* X 6 /
/

```

## COMPSOL Sets number of components in a SOLID, BLACKOIL run

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## COMPVD

## Total composition with respect to depth

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, $P_{sat}$ , at this depth. UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M) The $P_{sat}$ 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

## Well connection depth data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- 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**.
- I - location of connecting grid block(s)  
DEFAULT: 0 (Allows any I-value)
- J - location of connecting grid block(s)  
DEFAULT: 0 (Allows any J-value)
- K - location of upper connecting block in this set of data  
DEFAULT: 0 (Corresponds to top connection of well)
- K - location of lower connecting block in this set of data
- 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  $(1.0 - \text{CVEFRAC})$  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

$$S = \frac{h_t}{h_p} S_d + S_p \quad \text{Eq. 3.48}$$

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):

$$S_p = 1.35 \left( \frac{h_t}{h_p} - 1 \right)^{0.825} \{ 1n(h_{tpr} + 7) - 1.9 \} - [0.49 + 0.1 \ln(h_{tpr}) \ln(r_{wc})] \quad \text{Eq. 3.49}$$

where

$h_{tpr} = h_t$  divided by the square root of the vertical to horizontal permeability ratio  $K_v/K_h$

$$r_{wc} = r_w e^{0.2126(z_m/(h_t+2.753))}$$

$z_m$  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  $Kh$  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

$$h_t = \text{GBOT-GTOP}$$

$$h_p = \text{DBOT-DTOP}$$

$$z_m = \text{MIN}\{0.5(\text{DTOP} + \text{DBOT}) - \text{GTOP}, \text{GBOT} - 0.5(\text{DTOP} + \text{DBOT})\}$$

Eq. 3.50

## COMPVEL Well connection depth data in local grids

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 **COMPRP** 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  $(1.0 - \text{CVEFRAC})$  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.  $G_{TOP}$  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.  $G_{BOT}$  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  $D_{TOP}$  and  $D_{BOT}$  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 /  
/
```

## COMPW

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

---

**Note:** The following solution method options are also available, but are only recommended for the use of developers.

---

OFF: calculates the geometry but performs no calculations

COARSEN: assumes the solution is identical in all cells of the fault (VE is not applied).

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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' /
```

```
8 8 6 8 1 4 'X' /  
8 9 8 8 1 4 'Y' /  
9 9 8 9 1 4 'X' /  
/
```

## COORD

## Coordinate lines

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, DTHETA\_V, 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 NDIVIZ=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
2000 2000 1000 2000 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.  $(NDX + 1) * (NDY + 1) * NUMRES$  coordinate lines are expected, where NDX, 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 NDX, NDY, the number of coordinate lines expected is  $(NDX + 1) * (NDY + 1)$ .

ECLIPSE 100

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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (NDIVY + 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

- ECLIPSE 100 only*
  5. Lower bound for reservoir number for lateral connection to cells in different reservoirs (integer) (R1)

DEFAULT: Current reservoir number

- ECLIPSE 100 only*
  6. Upper bound for reservoir number for lateral connection to cells in different reservoirs (integer) (R2)

DEFAULT: Current reservoir number

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).

ECLIPSE 100

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,k1) 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.

ECLIPSE 100

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

ECLIPSE 100

Here `NDIVIX=5`, `NDIVIY=3`, `NDIVIZ=4`, as specified in `RUNSPEC` keyword `DIMENS`, and `NUMRES=2`.

```

----- K1 K2  completed      join below      lateral limits
COORDSYS
   1  2  COMP           /
   3  4  INCOMP        /

```

## COPY

## Copies data from one array to another

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

## ECLIPSE 100

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:

## ECLIPSE 100

This keyword may not be used with the [PBUB](#) or [PDEW](#) arrays as either **From** or **To**.

## 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
MIDS		

*ECLIPSE 300 only*

### EDIT section arrays

PORV	DEPTH	
TRANX (TRANR)	TRANY (TRANHT)	TRANZ
DIFFX (DIFFR)	DIFFY (DIFFHT)	DIFFZ
HEATX (HEATR)	HEATY (HEATHT)	HEATZ (HEATZ)
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

### REGIONS section arrays

SATNUM	PVTNUM	EQLNUM
IMBNUM	FIPNUM	ENDNUM
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](#), [TRANX](#) 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  1  19  1  4  /
    
```

```
          PERMX  PERMZ  / defaults to last specified box
/
----- ARRAY  FACTOR
MULTIPLY
          PERMZ  0.1   /
/
```

In the REGIONS section:

```
COPY
SATNUM  FIPNUM  /
/
```

## COPYBOX

## Copies an array of grid data from one box to another

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
x	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- ARRAY ----- SOURCE ----- DESTINATION -----
COPYBOX
  PERMX          2 3 2 3 1 1          2 3 2 3 4 4  /
  PORO           1 3 1 3 1 3          5 7 1 3 1 3  /
/

```

### Example 2

```

CARFIN
LGR1 1 2 1 2 3 4 4 4 4 /
----- ARRAY ----- SOURCE ----- DESTINATION -----
COPYBOX
  PERMX          2 3 2 3 1 1          2 3 2 3 4 4  /
  PORO           1 2 1 3 1 3          3 4 1 3 1 3  /
/
ENDFIN

```

## COPYREG

## Copy by FLUXNUM region, MULTNUM region or OPERNUM region

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

ECLIPSE 100

This keyword may not be used with the **PBUB** or **PDEW** arrays as either **From** or **To**.

### 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*

FLUXNUM            MULTNUM            MPFANUM

*ECLIPSE 300 only*

MIDS

### **EDIT section arrays**

PORV                    DEPTH

TRANX (TRANR)    TRANY (TRANHT)    TRANZ

DIFFX (DIFFR)    DIFFY (DIFFHT)    DIFFZ

HEATX (HEATR)    HEATY (HEATHT)    HEATZ (HEATHT)

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

*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

*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.
- 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.

*ECLIPSE 100 only*

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component
NO	Not used. The viscosity for this component is computed as if this keyword was not present.
LMIX	The LBC viscosity model is applied with these components' oil phase molar density.
LEMIX	The LBC viscosity model is applied on the LEMIX components densities where an EoS is used to compute the phase molar density.

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:

```
COVTYPE
LMIX LMIX LMIX LMIX /
```

## CPR

## CPR Linear solver

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
/
```

## CREF

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$\rho_c = \frac{\rho_{ref,c}}{\left[1 - c_{P_c}(P - P_{ref,c})\right] \left[1 + c_{T1_c}(T - T_{ref,c})\right]} \quad \text{Eq. 3.51}$$

where

$\rho_{ref,c}$  is the reference density, specified with keyword [DREF](#)

$c_{P_c}$  is the oil component liquid compressibility, specified with this keyword

$P_{ref,c}$  is the reference pressure specified, with keyword [PREF](#)

$T_{ref,c}$  is the reference temperature, specified with keyword [TREF](#)

$c_{T1_c}$  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 /
```

## CREFW

## Water component compressibility

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

CREFW specifies the water compressibility for each of the  $N_w$  water components.

The water component densities, at pressure  $P$ , are calculated from:

$$\rho_w = \rho_{ref_w} \cdot (1 + X + X^2/2)$$

$$X = c_{P_w} \cdot (P - P_{ref_w})$$

where

$\rho_{ref_w}$  is the reference density, specified with keyword [DREFW](#)

$c_{P_w}$  is the water component compressibility, specified with this keyword

$P_{ref_w}$  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
```

## CREFW

## 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*.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## CRITPERM Permeability criterion for Vertical Equilibrium cell compression

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#)
- $TRANZ = 0.0$
- $PERMZ \leq$  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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> ([METRIC](#)), lb/ft<sup>3</sup> ([FIELD](#)), gm/cc ([LAB](#)), kg/m<sup>3</sup> ([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 /
/
```

## CSKIN

## Update well connection skin factors

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /  
/
```

## CVCRT

## Convergence criteria

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>2</sup>** 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

10. Minimum rms linear solver residual for convergence of linear equations.

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

24. Convergence tolerance for Multisegment Well Model molar densities.  
UNITS: kg-M/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (PVT-M)  
DEFAULT: 0.015 kg-M/rm<sup>3</sup>
25. Convergence tolerance for Multisegment Well Model molar flow rate (homogeneous flow segments).  
UNITS: kg-M/day (METRIC), lb-M/day (FIELD), gm-M/hr (LAB), kg-M/day (PVT-M)  
DEFAULT: 50 kg-M/day
26. Convergence tolerance for Multisegment Well Model energy density.  
UNITS: kJ/rm<sup>3</sup> (METRIC), Btu/rb (FIELD), J/rcc (LAB), kJ/rm<sup>3</sup> (PVT-M)  
DEFAULT: 50 kJ/rm<sup>3</sup>
27. Convergence tolerance for Multisegment Well Model velocity (drift flux segments).  
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](#).

- DEFAULT: 1
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 /
```

## CVTYPE

## Component volatility type

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component	Option
LIVE	for components that are both volatile and condensable.	
SOLID	for components that are only in the solid phase	Solid
DEAD	for components that are only in the oil phase	Thermal
GAS	for components that are only in the gas phase	Thermal

Table 3.7: CVTYPE component values

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:

Type	Solid phase	Oil phase	Gas phase
LIVE		x	x
SOLID	x		
DEAD		x	
GAS			x

Table 3.8: Phases corresponding to each volatility type

Thermal only

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:

```
CVTYPE
GAS LIVE DEAD DEAD /
```

## CVTYPES

## Component volatility type for surface conditions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Type	Component	Option
LIVE	for components that are both volatile and condensable.	
SOLID	for components that are only in the solid phase	Solid
DEAD	for components that are only in the oil phase	Thermal
GAS	for components that are only in the gas phase	Thermal

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:

Type	Solid phase	Oil phase	Gas phase
LIVE		x	x
SOLID	x		
DEAD		x	
GAS			x

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 /
```

## CWTYPE

## 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:

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

Type	Component
SURFF	Surfactant component for foam model ( <a href="#">FOAMFSC</a> )
SURFS	Surfactant component for surface tension effects ( <a href="#">SURFST</a> )
SURF	Surfactant component for both foam and surface tension effects
POLY	Polymer component for water phase viscosity effects
ALK	Alkaline component for water phase viscosity effects

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 9<sup>th</sup> 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 /
```

## **D**

This section contains an alphabetic listing of keywords starting with the letter D.

# DATE

## Outputs the date to the summary file

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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

```
DATE
```

#### Example 2

Example RUNSUM output

```
SUMMARY OF RUN test
```

```

DATE          YEARS   DAY  MONTH  YEAR   FOPR   WGOR   WBHP
              YEARS                STB/DAY MSCF/STB PSIA
              YEARS                PRODUCER PRODUCER
-----
19-OCT-92    0          19   10   1992     0         0   4800.000
20-OCT-92    0.002738   20   10   1992  20000.00  1.270000  2892.246
21-OCT-92    0.006462   21   10   1992  20000.00  1.270000  2743.021
24-OCT-92    0.014967   24   10   1992  20000.00  1.270000  2578.314
28-OCT-92    0.026090   28   10   1992  20000.00  1.270000  2459.743
 3-NOV-92    0.043609    3   11   1992  20000.00  1.263817  2364.214
14-NOV-92    0.073839   14   11   1992  20000.00  1.241348  2287.771
    
```

## DATES

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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, 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:** 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is optional and similar to [DATUMR](#). 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 [DATUMR](#) keyword has been processed previously, in which case the datum values of any unspecified fluid-in-place set will be based on the [DATUMR](#) keyword.

---

**Note:** For ECLIPSE 100 only, an exception is the fluid-in-place set created by the keyword [FIPOWG](#) 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 [RPP0](#), [RPPW](#), [RPPG](#) and [RPPC](#) (ECLIPSE 100 only).

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)}$$

**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 IZ1 IZ2
   ON    4  4   5  5   1  3 /
```

## DCQDEFN Definition of the DCQ - gas rate or energy

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword may be used to specify whether the DCQ targets set in keyword [GASYEAR](#), [GASPERIO](#), [GDCQ](#), [GASF'TARG](#) 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 (/):

- 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 /
```

## DEADOIL

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

---

## DEBUG

## Controls on debug output

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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).

DEBUG item	Effect of DEBUG item
Item 1	Terminal output copied to debug file. Timing reports if > 1.
Item 2	Debug summary for all wells.
Item 3	Details of storage allocation. If set > 1 the local grid storage allocation is also output.
Item 4	Details of index arrays from RCINDX.
Item 5	Debug of environment variables.
Item 6	Debug from matrix set-up (see also switches 45-50). 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.
Item 7	Debug from linear solver.
Item 8	Debug from equilibrator.
Item 9	Debug from non-linear solver (chops of $S_w, S_g, S_o, P_o, R_s, R_v, BHP$ ). If >1 debug of PVT extrapolation, and state transitions.
Item 10	Debug from NNC generation.
Item 11	Debug from network option balancing calculation.
Item 12	Debug from Fetkovich or Carter-Tracy aquifer calculations.
Item 13	Debug from vertical equilibrium option.
Item 14	Debug from transmissibility calculations.
Item 15	Debug from threshold pressure calculation.

DEBUG item	Effect of DEBUG item
Item 16	<p>Debug from hysteresis option.</p> <p>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.</p>
Item 17	Debug from API tracking option.
Item 18	Debug from the ACTION keyword.
Item 19	If >0 grid displays of the non-linear residual function are produced in the debug file at each non-linear iteration.
Item 20	Debug from input parser and the I/O PUT and GET Routines.
Item 21	Debug from total compressibility check calculation.
Item 22	Debug from cascade solver.
Item 23	Debug from end-point scaling option.
Item 24	Debug from tracer tracking option.
Item 25	Debug from gravity imbibition and drainage calculation in dual porosity/dual permeability systems.
Item 26	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>0, then only the n <sup>th</sup> local grid is checked.
Item 27	If >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.
Item 28	Debug from polymer flood model.
Item 29	Debug from dynamic memory allocation module.
Item 30	Debug from Gi-condensate/volatile oil model.
Item 31	Debug from gas field operations model.
Item 32	Debug from Solvent Model.
Item 33	Subroutine indicator to detect the position of a program hang.
Item 34	Debug of internal tables generated from alternative keywords.
Item 35	Debug from the surfactant model.
Item 36	Debug from nine-point scheme.
Item 37	Debug from gas lift optimization calculation.
Item 38	<p>If &gt;0 a line search is performed along the newton direction.</p> <p>The following steps are used alpha = 10*0.1,-1,1. the search is only performed newton iteration &gt; debug(38).</p>

DEBUG item	Effect of DEBUG item
Item 39	<p>Debug for the parallel options, reservoir coupling and Open-ECLIPSE message passing routines.</p> <p>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 <code>N</code> is the process number.</p> <p>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.</p>
Item 40	Debug from local grid refinement.
Item 41	Debug from parallel LGR option.
Item 42	Debug from reservoir coupling facility.
Item 43	<p>Debug from the domain decomposition parallel option.</p> <p>When <code>set = 2</code>, 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 <code>n</code> is the slave number.</p>
Item 44	Debug information about the phase flows.
Item 45	Lower bound on X axis of matrix set-up debug output (switch 6).
Item 46	Upper bound on X axis of matrix set-up debug output (switch 6).
Item 47	Lower bound on Y axis of matrix set-up debug output (switch 6).
Item 48	Upper bound on Y axis of matrix set-up debug output (switch 6).
Item 49	Lower bound on Z axis of matrix set-up debug output (switch 6).
Item 50	Upper bound on Z axis of matrix set-up debug output (switch 6).
Item 51	Debug from the History Matching option.
Item 52	Writes the RFT file as a GRAF user file, provided the output files are formatted.
Item 53	Debug from the flow reversal code.
Item 54	Fraction of the Newton direction used if <code>DEBUG</code> switch 38 is set.
Item 55	Debug from viscous displacement.
Item 56	Replaces bands by NNCs in the solver.
Item 57	Reserved for experimental features.
Item 58	Reserved for experimental features.
Item 59	Debug from timestep selection.
Item 60	Nested LGRs.
Item 61	Debug from petro-elastic model. The cells for which output is to be written are specified by switches 45-50.
Item 76	Debug from rivers. If <code>set&gt;0</code> then a table of the river solution is output at every well iteration. If <code>set&gt;1</code> , solution obtained from river pre-solver is output.
Item 77	Debug from pseudo steady-state calculation.

DEBUG item	Effect of DEBUG item
Item 78	Statistics from SAMG solver setup and cycling.
Item 79	Outputs the norms of the residuals.
Item 80	Reserved for developer use.
Item 81	Reserved for experimental features.
Item 82	Debug from timestep selection.
Item 83	
Item 84	
Item 85	
Item 86	Reserved for developer use.
Item 87	Reserved for developer use.

Table 3.12: DEBUG keyword values

## Example

```
DEBUG
1 1 0 0 0 0 1 1 1 0 0 0 6*0 1 1 /
DEBUG
78* 1 /
```

## DEBUG3

## Controls on debug output

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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.

Item	Effect of DEBUG3 item
1	Debug from allocation/keyword reading. If 1, summary of storage allocation written to debug file. If >1, summary is also written to Print file, debug from allocation routines to Debug file.
2	Debug from file opening and closing.
3	Debug from table reading and checking.
4	Debug from equilibration. Can be set to 1 or 2.
5	Debug from timestepping.
6	Debug from residual calculation.
7	Matrix evaluation by differencing.
8	Debug from non-linear solver.
9	Debug from linear solver. Can be set to 1 or 2.
10	Clocks. Can be set to 1 or 2. 2 is for ECLIPSE 100-style output.
11	Debug from well calculations. Can be set to 1 or 2.
12	Debug from timestepping.
13	Debug of vector runs.
14	Debug from separators.
15	Debug of solution and properties. Can be set to 1 or 2.
16	Debug from flash calculations.
17	Null check of analytical and numerical flow Jacobians.

Item	Effect of DEBUG3 item
18	Set monitor flag to on (-1 sets back to off). If set to 2, will flag all subroutine entry and exit points and write message to the debug flag. If set to 3, will also write message to the screen or the log file.
19	If set to 1, check inter-array gaps at each memory release. If set to 2, also check inter-array gaps at each subroutine entry and exit point.
20	Force divide check on fatal error to get trace back.
21	ACTION debug.
22	Basic vector flash debug: If set to 1: Summary debug (includes iteration counts, cell states and max residual). If set to 2: also input/output debug for individual cells. If set to 3: also output of residuals, C/D matrices, Z-factors + derivatives.
23	Output of matrix.
24	Debug from NNC calculations. Can be set to 1 or 2.
25	Debug from Michelsen vector stability test. If set to 1: Individual cell information from SS/BFGS algorithms.
26	Not used.
27	Newton Search direction debug facility (use with IDEBUG (40)). Turned on after NEWTIT > IDEBUG (27), if IDEBUG (27)>0.
28	Debug of B matrix
29	Newton Search direction debug facility (see DEBUG (27)). Specifies the number of sub-steps (up to a max. of 100).
30	Lower x-index for debug output. DEFAULT: 1
31	Upper x-index for debug output. DEFAULT: $N_x$
32	Lower y-index for debug output. DEFAULT: 1
33	Upper y-index for debug output. DEFAULT: $N_y$
34	Lower z-index for debug output. DEFAULT: 1

Item	Effect of DEBUG3 item
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: <i>inactive</i>
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 <a href="#">SOLVDIRS.</a> )
49	Select outer internal direction. Can be set to 1,2 or 3. (See also <a href="#">SOLVDIRS.</a> )
50	Debug of nested factorization pointers.
51	Debug of LGR.
52	<a href="#">PVTO</a> and <a href="#">PVTG</a> 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 OGW, OW, GW, W, OG, O, G, R, where 5=OGW, 6=OW and so on.
58	Miscible option debug.

Item	Effect of DEBUG3 item
59	Non-equilibrium restart debug.
60	Debug from K-value generation.
61	Obtain extra properties for debug (specify before RUNSPEC).
62	Suppress space saving in matrix storage.
63	Force corner point transmissibility calculations.
64	Debug from <code>OVERTR</code> , <code>OVERTTHT</code> , <code>OVERTX</code> , <code>OVERTY</code> , <code>OVERTZ</code> / <code>TRANR</code> , <code>TRANHTHT</code> , <code>TRANX</code> , <code>TRANZ</code> 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: <ol style="list-style-type: none"> <li>1. Check of Gamma inverse for each column J in LUD</li> <li>2. Check of solution of multisegment well matrix equation by multiplying out</li> <li>3. Check of full Jacobian by differencing</li> <li>4. Statistics of variable update (including well convergence information)</li> <li>5. Check of DMXP, derivatives of M-S well primary variables with respect to XP</li> <li>6. Check domain and MPRP inter-array gaps</li> <li>7. Check derivative of WRES with respect to BHP</li> <li>8. Check derivative of WRES with respect to XP (brief)</li> <li>9. Check derivative of WRES with respect to XP (verbose)</li> <li>10. Check derivative of Q with respect to XP (brief)</li> <li>11. Check derivative of Q with respect to XP (verbose).</li> </ol>
75	Debug from flux option.

Item	Effect of DEBUG3 item
76	$P_{sat}$ debug. 1: Trap $P_{sat}$ solution failures. 2: $P_{sat}$ 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. 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 > 0, skips Young re-ordering after an Appleyard chop.
92	Debug <a href="#">WARP</a> setup.
93	Debug solution/wells output at each Newton iteration (ECLIPSE 100 DEBUG (19)).
94	Rowsum debug $v = (B^{*-1})Av$ .
95	Reserved.
96	WARP LGR debug: if set to >0 then sets up solver pointers for LGRs, blocks NNCs by to-from LGRs and sets QQWARP to true. If set to >1, the NNCs are also blocked by <a href="#">AIM</a> state within each group.
97	<a href="#">GPMAINT</a> debug.

Item	Effect of DEBUG3 item
98	Override timestepping defaults: <ol style="list-style-type: none"> <li>1. <a href="#">IMPES</a></li> <li>2. <a href="#">AIM</a></li> <li>3. <a href="#">FULLIMP</a></li> </ol>
99	Check on behavior of AIM.
100	Limited general debug.
101	Debug of multipoint flux & tensor permeabilities.
102	Debug tests via Comsim test driver.
103	Debug of <a href="#">GEOMECH</a> : <ol style="list-style-type: none"> <li>1. check full Jacobian - can be used with internal limits (value of item 30 to 35 or 36 to 37)</li> <li>2. write out matrix in TMCRWS format</li> <li>3. not used</li> <li>4. solve matrix with direct LU decomposition solver</li> <li>5. only perform direct solving if <a href="#">WARP</a> or <a href="#">ECLSOL</a> fails, then only compare answers</li> <li>6. same as 1, but only check geomech variables</li> <li>7. check that the plastic yield function is being zeroed</li> <li>8. check derivatives numerically of yield function with respect to stress</li> <li>9. statistics on plastic calculation</li> <li>10. if moving grid, delay copying latest pore volume until XGMGUD</li> <li>11. Plastic iteration summary</li> <li>12. <a href="#">SAMG</a> solver</li> <li>13. Reserved for developer use</li> <li>14. FAULT NNC summary.</li> </ol>
104	Experimental treatment of well terms for the linear solver.
105	Improved black oil formulation: <p>If set to 1: Non-linear convergence</p> <p>If set to 2: +linear convergence.</p>
106	Debug from network simulator and balancing calculations. (Network Option.)
107	Velocities and radii at pressure points in pseudo calculation (compared with velocity-dependent $K_r$ ).
108	Initialize derivatives in domain calculations. <p>If set to 1: initialized to 0.</p> <p>If set to 2: initialized to NaN.</p>

Item	Effect of DEBUG3 item
109	Compares the results from skipping a stability calculation with a rigorous stability calculation. (Fast Flash calculation.)
110	Debug from the Gradient Option.
111	Debug from PEBI grids (global/local/solver mapping).
112	For stability debug.
113	Reserved.
114	Reserved.
115	Debug of group solving. <ol style="list-style-type: none"> <li>1. Production sweep</li> <li>2. Gas injection sweep</li> <li>3. Water injection sweep.</li> </ol>
116	Detailed well debug. <ol style="list-style-type: none"> <li>1. Unable to flow wells and their revival/closure.</li> </ol>
117	Debug from conductive faults.
118	Debug for FrontSim material balance errors.
119	Debug sector model.
120	production optimization debug: >0: Basic output 2: Numerical gradients.
121	Adjoint gradient debug: <ol style="list-style-type: none"> <li>1. Basic</li> <li>2. Basic + adjoint details</li> <li>3. Full output (=basic + sim + adjoint)</li> <li>4. Full + use LU direct solver.</li> </ol>
122	Detailed (non-linear) convergence debug.
123	Derivative checker for conductive faults: Multiple of 2 for $dx / dy$ Multiple of 3 for $dy / dY$ Multiple of 5 for $dY / dX$ Multiple of 7 for $dx / dX$ .
124	Reserved.
125	Perturbation in numerical gradient computation is $10^{-(\text{value of item 125})}$ .

Item	Effect of DEBUG3 item
126	Adjoint partial gradient calculation options: Bit-packed: 1= analytic dFdp (only implemented for certain parameters) 2 = analytic dRdp, (only implemented for certain parameters) 4 = numerical dFdX. (not advised: output is extremely verbose).
127	Reserved.
128	Reserved.
129	If set to 1: dump prior to flash. If set to 2: dump after flash. If set to 3: dump prior to and after flash.
130	Time linear solver timer: iterations = mod(100, item 130 value) solver iterations = mod(10, item 130 value).
131	AIM done per color = 1.
132	Turn off WARP NNCs.
133	WARP preconditioner.
134	If set to 2 and FULLIMP then do NOT break up IVRB runs.
135	Reserved.
136	Heater derivative check.
137	Reserved.
138	ILU0 solver debug.
139	Reserved.
140	Reserved.
141	Linear system in sparse (compressed row) format (NEWTIT = (value of item 141)-1)
142	DEBUG3 items for the Gas Lift Optimization calculation are: Set to 1 for basic debug Set to 2 for above and well gas lift increment/decrement debug Set to 3 for above and dead well debug Set to 4 for above and estimated group flow rates debug Set to 6 for above and well proxy model debug.
143	Reserved for developer use.
144	Reserved for developer use.
145	Reserved for developer use.

Item	Effect of DEBUG3 item
146	Reserved for developer use.
147	Reserved for developer use.
148	Reserved for developer use.
149	Reserved for developer use.
150	Reserved for developer use.
151	Reserved for developer use.
152	Reserved for developer use.
153	Reserved for developer use.
154	Reserved for developer use.
155	Reserved for developer use.
156	Reserved for developer use.
157	If > 0 outputs extra arrays for the geomechanics data structure. Only useful for developers.
158	Reserved for developer use.
159	Reserved for developer use.
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

```
DEBUG3  
4*0 1 1 0 1 /  
DEBUG3  
120* 1 /
```

## DELAYACT Specifies a set of keywords to be processed after an ACTION is triggered

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 MNLPAK 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.

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 '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

```

```
WCONPROD
  'PRODUCER' 'SHUT' 'WRAT' 1* 0. /
/
DELAYACT
'ENDRUN20' 'ENDRUN' 20 /
END
ENDACTIO
ENDACTIO
ENDACTIO
ENDACTIO
```

## DENAQA

## Specify coefficients of Ezrokhi's method for aqua density

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 (/).

- The density of oil at surface conditions.  
 UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)  
 ECLIPSE 300 DEFAULT: 600 kg/m<sup>3</sup> = 37.457 lb/ft<sup>3</sup>
- The density of water at surface conditions.  
 UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)  
 ECLIPSE 300 DEFAULT: 999.014 kg/m<sup>3</sup> = 62.366 lb/ft<sup>3</sup>
- The density of gas at surface conditions.  
 UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)  
 ECLIPSE 300 DEFAULT: 1 kg/m<sup>3</sup> = 0.062428 lb/ft<sup>3</sup>

The keyword [GRAVITY](#) may be used as an alternative to DENSITY.

ECLIPSE 300

In compositional runs the oil and gas surface densities are obtained from the equation of state and the values entered under the density keyword are not used. A dummy value may be entered in this case. The water density is always required. Note that DENSITY always contains these items, even if not all three phases are present in the run.

ECLIPSE 100

**Note:** When using the API Tracking option (keyword [API](#) in the RUNSPEC section), the set of records must be entered in increasing order of oil surface 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 300

## DEPTH

## Grid block center depths

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  
JAN, FEB, MAR, APR, MAY, JUN, JLY, AUG, SEP, OCT, NOV or DEC.  
JUL is an acceptable alternative to JLY.
2. Year  
Input the year as a four-digit integer number.
3. Time of observation in 24 hour clock format, hh:mm  
Do not use 24:00; use 00:00 of the following day
4. 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 the 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
10000.0 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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

```

```
1.0 1.0 0.0  
10000.0 0.9 0.07 0.03 /
```

## DGRDT

## Maximum rate of change of guide rate for wells under group control

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

```
DGRDT
0.01 /
```

## DGRID

## Dump section of grid

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)}$$

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 /
```

## DIAGDISP

## Enables alternate form of tracer dispersion

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$A_1 = D_1(|v_x|, c) + D_2(|v_y|, c) + D_3(|v_z|, c)$$

$$A_2 = D_4(|v_x|, c) + D_5(|v_y|, c) + D_6(|v_z|, c) \text{ and}$$

$$A_3 = D_7(|v_x|, c) + D_8(|v_y|, c) + D_9(|v_z|, c)$$

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$D_i^a = D_i \left| \frac{\partial}{\partial \ln y_i} (\ln f_i) \right| \quad \text{Eq. 3.52}$$

where the normal diffusion coefficients are defined by the condition:

$$J_i = -cD_i \frac{\partial y_i}{\partial d} \quad \text{Eq. 3.53}$$

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:

$$F_{ig}^{\text{diff}} = TD D_{ig}^a (y_i S_g b_g^m)_u \Delta \left( \ln f_i - \frac{M_i Gh}{RT} \right) \quad \text{Eq. 3.54}$$

where  $TD$  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**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB), m<sup>2</sup>/day (PVT-M)

## DIFFC

## Molecular diffusion data for each PVT region

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/):

- The oil molecular weight.  
UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB)
- The gas molecular weight.  
UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB)
- The gas-in-gas diffusion coefficient.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)
- The oil-in-gas diffusion coefficient.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)  
DEFAULT: Gas-in-gas coefficient's value
- The gas-in-oil diffusion coefficient.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)  
DEFAULT: Gas-in-gas coefficient's value
- The oil-in-oil diffusion coefficient.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)  
DEFAULT: Gas-in-oil coefficient's value
- The gas-in-oil diffusion coefficient for cross phase diffusion.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)  
DEFAULT: 0.0
- The oil-in-oil diffusion coefficient for cross phase diffusion.  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)  
DEFAULT: 0.0

ECLIPSE calculates the diffusive flow from the following expression (in the gas-in-gas case):

$$F_{\text{diff}} = T_d D_{\text{gg}} S_g B_g (x_i - x_j) \quad \text{Eq. 3.55}$$

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  $\text{cm}^2/\text{sec}$ ; they may be converted as follows:

$$1 \text{ cm}^2/\text{s} = 8.64 \text{ m}^2/\text{day} \text{ (METRIC)}$$

$$1 \text{ cm}^2/\text{s} = 110.0976 \text{ ft}^2/\text{day} \text{ (FIELD)}$$

$$1 \text{ cm}^2/\text{s} = 3600 \text{ cm}^2/\text{hr} \text{ (LAB)}$$

ECLIPSE 100

---

**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  $\text{ft}^3$ . 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 /

```

## DIFFCBM

## Coal Bed Methane diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $d_1 \ d_2 \ d_3 \ \dots d_{ncomps}$

Diffusion coefficients for coal region number 1. The diffusion coefficients should be larger or equal to zero.

UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB), m<sup>2</sup>/day (PVT-M)

DEFAULT: 0.0

Record 2:  $d_1 \ d_2 \ d_3 \ \dots d_{ncomps}$

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 cm<sup>2</sup>/s, they may be converted as follows:

LAB units: 1 cm<sup>2</sup>/s = 3600 cm<sup>2</sup>/hr

METRIC or PVT-M units: 1 cm<sup>2</sup>/s = 8.64 m<sup>2</sup>/day

FIELD units: 1 cm<sup>2</sup>/s = 92.9979 ft<sup>2</sup>/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 / -- coalnum=1
/
```

## DIFFCGAS Gas diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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]):

$$J_i = -cD_i \frac{\partial y_i}{\partial d} \quad \text{Eq. 3.56}$$

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:

$$F_{ig}^{\text{diff}} = TD D_{ig} (S_g b_g^m) \Delta y_i \quad \text{Eq. 3.57}$$

where

$TD$  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

```

DIFFCGAS
--For      N2      C1      C25      C7+1      C7+2 (field units)
           0.16   0.15   0.04     0.013    0.012 /

```

## DIFFCGO

## Gas-oil diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\text{cm}^2/\text{s}$ , 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

```
DIFFCGO
--For      N2      C1      C25      C7+1      C7+2 (field units)
           0.016   0.015   0.004   0.0013   0.0012 /
```

## DIFFCGW

## Gas-water diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [DIFFCGW](#).

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  $\text{cm}^2/\text{s}$ , 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

```
DIFFCGW
--For      CO2      H2O      (field units)
           0.016    0.015    /
```

## DIFFCOAL Gas diffusion data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)
2. The re-adsorption fraction  
DEFAULT: 1.0
3. The solvent diffusion coefficient  
UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/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
```

## DIFFCOG

## Oil-gas diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [DIFFCOG](#).

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  $\text{cm}^2/\text{s}$ , 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

```
DIFFCOG
--For      N2      C1      C25      C7+1      C7+2 (field units)
           0.16    0.15    0.04    0.013    0.012 /
```

## DIFFCOIL

## Oil diffusion coefficients

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [DIFFCGAS](#) keyword.

UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB), m<sup>2</sup>/day (PVT-M)

DEFAULT: 0.0

## DIFFCWAT **Water diffusion coefficients**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB), m<sup>2</sup>/day (PVT-M)

DEFAULT: 0.0

## DIFFCWG

## Water-gas diffusion coefficients

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\text{cm}^2/\text{s}$ , 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

```
DIFFCWG
--For      CO2      H2O      (field units)
           0.16     0.15     /
```

## DIFFDP

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

## DIFFMMF

**Matrix-fracture diffusivity multipliers**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
X	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

ECLIPSE 100

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
      3   5   1  10   2   4   /
DIFFMR-
90*2.0 /
ENDBOX

```

## DIFFMTHT

## Diffusivity multipliers in the theta direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

ECLIPSE 100

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## DIFFMX

## Diffusivity multipliers in the X-direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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-](#).

ECLIPSE 100

---

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the DIFFMX values.

---

### Example

```

----- 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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   1  10   2   4   /
DIFFMX-
90*2.0 /
ENDBOX

```

## DIFFMY

## Diffusivity multipliers in the Y-direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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-](#).

ECLIPSE 100

---

**Note:** Any non-neighbor connections generated due to faults will have diffusivities which reflect the DIFFMY values.

---

### Example

```

----- 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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   1  10   2   4   /
DIFFMY-
90*2.0 /
ENDBOX

```

## DIFFMZ

## Diffusivity multipliers in the Z-direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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-](#).

ECLIPSE 100

---

**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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      3   5   1   10   2   4   /
DIFFMZ-
 90*2.0 /
ENDBOX

```

## DIFFR

## Diffusivities in the radial direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `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+1, J, K).

The `DIFFR` keyword is entirely optional. Any diffusivities that 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:** `DIFFR` does not affect non-neighbor connections due to faults.

---

ECLIPSE 300

---

**Note:** For ECLIPSE 100 compatibility, the `EDIT` section keyword `DIFFTR` has been aliased to `DIFFR`. However, `DIFFTY` can still be used in the `EDIT` section as documented in previous releases.

---

Diffusivities are analogous to transmissibilities for diffusive flow, and are described in "[Diffusion](#)" in the *ECLIPSE Technical Description*.

### Example

```

----- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX          1  1  1  3  1  1  /
DIFFR
3*0.0 /

```

## DIFFTGAS Thermal diffusion coefficients for gas

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$F_{ig}^{\text{diff}} = T_D D_{ig}^a \left( \frac{y_i S_g b_g^m}{RT} \right)_u \Delta \left( RT \ln f_i - M_i Gh + M_i D_{ig}^T \ln T \right) \quad \text{Eq. 3.58}$$

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `DIFFTTHT` 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

```

----- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
      1   3   1   3   1   1   /
DIFFTHT
9*0.0 /

```

## DIFFTOIL

## Thermal diffusion coefficients for oil

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 DIFFFAOIL 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

## DIFFUSE

## Enables molecular diffusion

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

# DIFFX

## Diffusivities in the X-direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

ECLIPSE 300

---

**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 /
    
```

## DIFFY

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

---

ECLIPSE 300

---

**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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (PVT-M)

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   2   /
DIFFZ
18*0.0 /

```

## DIMENS

### Specifies the dimensions of the grid

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $\theta$  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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $R_s$  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 [PVTG](#) or [PVC](#) keywords are present.

## DISPBC

## Velocity boundary conditions for coupled rock and stress calculations

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## DISPBCL

## Velocity boundary conditions for coupled rock and stress calculations in a Local Grid Refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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' /
/

```

## DISPDIMS **Dispersion dimension data**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## DISPERSE Dispersion tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 MXVVP (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<sup>2</sup>/day (`METRIC`), ft<sup>2</sup>/day (`FIELD`), cm<sup>2</sup>/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 /
```

```
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
```

## DNGL

## Partial densities for NGLs

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 4 6 7 /
```

## DPCDT

## Maximum rate of change of capillary pressure

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $dP_c / dt = 0.0099972$  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 /
```

## DPGRID

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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  $NDIVIZ / 2$  layers), the missing values for the remaining fracture layers being obtained from the corresponding matrix cell.

This operation will be performed for the [DX](#), [DY](#), [DZ](#), [PERMX](#), [PERMY](#), [PERMZ](#), [PORO](#), [TOPS](#), [MIDS](#), [NTG](#), [DZNET](#), [ZCORN](#), [PERMXY](#), [PERMYZ](#) and [PERMZ](#) 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 [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 [BOX](#) keyword may be used to specify values for the cells required.

When using an unstructured grid supplied by FloGrid, the [DEPTHS](#) entered in the [EDIT](#) section are also copied from matrix to fracture.

When using the Temperature Option, the thermal conductivity [THCONR](#) is also copied from the matrix to the fracture.

When using the Thermal Option, the thermal conductivity [THCONR](#), and the rock heat capacity data [HEATCR](#) and [HEATCRT](#) are copied from the matrix to the fracture.

When using the Thermal Option, the rock thermal conductivity [THCROCK](#), and the phase thermal conductivities [THCOIL](#), [THCGAS](#), [THCWATER](#) or [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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$K_r = \bar{K}_r + M(K_r(s) - \bar{K}_r) \quad \text{Eq. 3.59}$$

where

$\bar{K}_r$  is the relative permeability at  $\bar{s} = (S_{OWCR} + 1.0 - S_{WCO}) / 2$

$K_r(s)$  is the input table relative permeability

SOWCR is the critical oil to water saturation

SWCO is the connate water saturation

$M$  is the modification function, which is given by  $M = 4.0m_w K_r(s)^2 - 4.0m_w K_r(s) + 1$

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:

$$\bar{s} + M(s' - \bar{s}) = s \quad \text{Eq. 3.60}$$

where

$\bar{s}$  is  $(\text{SOWCR} + 1.0 - \text{SWCO}) / 2$

$M$  is the modification function, which is given by  $M = 4.0m_w K_r(s')^2 - 4.0m_w K_r(s') + 1$

$K_r$  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:

$$K_{rf}(S_w) = K_{rf}^{table}(S_w) \cdot K_{rmmax} / K_{rfmax} \quad \text{Eq. 3.61}$$

where

$K_{rf}^{table}(S_w)$  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 /

```

## DPNUM

## Identifies extent of dual porosity region

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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.

---

**Note:** Grid data for single porosity cells is only required in the first 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.

ECLIPSE 300

This keyword is not permitted if multi porosity is specified, either as a general multi porosity run using [NMATRIX](#) with value greater than 1, or as a triple porosity run using [TRPLPORO](#).

ECLIPSE 100

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.

ECLIPSE 100

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.

ECLIPSE 300

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:

```

DPNUM
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 /

```

## DR

## R direction grid block sizes

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
1 2 4 8 16 32 64 128 256 512 1000 2000 /

```

## DRAINAGE **Request drainage option in hysteresis**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This option restricts  $K_r$  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.

## DREF

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Reference densities

In a run with  $N_c$  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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.62}$$

where

$\rho_{ref,c}$  is the reference density, specified with this keyword

$c_{P_c}$  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 keyword [TREF](#)

$c_{T1_c}$  is the thermal expansion coefficient, specified with keyword [THERMEX1](#).

There should be one value for each component:

UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## DREFW

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Reference densities for water components

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:

$$\rho_w = \rho_{ref_w} \cdot (1 + X + X^2/2)$$

$$X = c_{P_w} \cdot (P - P_{ref_w})$$

where

$\rho_{ref_w}$  is the reference density, specified with this keyword

$c_{P_w}$  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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

DEFAULT: 999.014 kg/m<sup>3</sup>

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

## DRILPRI

## Defines the default priority formula for the prioritized drilling queue

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#), [GCONSALE](#)). 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

$$\text{Priority} = \frac{A + BQ_o + CQ_w + DQ_g}{E + FQ_o + GQ_w + HQ_g} \quad \text{Eq. 3.63}$$

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.

```
DRILPRI  
100 0.0 1.0 0.0 0.0 1.0 0.0 0.0 0.0 /
```

## Example 2

Recalculate drilling priorities at each timestep. Drill wells from lowest to highest water-liquid ratio.

```
DRILPRI  
0 0.0 0.0 1.0 0.0 0.0 1.0 1.0 0.0 /
```

## DRSDT

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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-resolution). 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-resolution).

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  $\text{DRSDT} = 0$  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 in to 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

```
DRSDT
0.0003 / (in Mscf/stb/day)
```

## DRV

## Radial-direction grid block sizes (vector)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $I^{\text{th}}$  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 has 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).

ECLIPSE 300

## Example

With `NDIVIX=11` as specified in `RUNSPEC` keyword `DIMENS`:

```
DRV
1 2 4 8 16 32 64 128 256 512 1024 /
INRAD
1 /
```

## DRVDT

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 ( $R_v$ ) 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,  $R_v$  cannot rise and free oil does not vaporize in undersaturated gas (no re-vaporization). At the other extreme, if DRVDT is very large,  $R_v$  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  $R_v$ .

Moreover increases in  $R_v$  are constrained by the availability of free oil and the gas saturation limit ( $R_{vsat}$ ).

---

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.

UNITS:  $\text{sm}^3/\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{scc}/\text{hr}$  (LAB)

### Example

```
DRVDT
0.009 /
```

## DSPDEINT

**Interpolation of dispersion tables by water density**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## DSTNUM

## Tertiary hysteresis saturation function region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## DTHETA

## Theta-direction grid block sizes

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IR1-IR2 JT1-JT2 KZ1-KZ2
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 /

```

## DTHETAV

**Angular sizes of grid blocks (vector)**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword specifies the size of the cells in the theta direction.

The keyword should be followed by `DIVIIY` positive real numbers, where `NDIVIIY` is the grid theta dimension entered using the `DIMENS` keyword in the `RUNSPEC` section.

The  $J^{\text{th}}$  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 `NDIVIIY=8`, specified in `RUNSPEC` keyword `DIMENS`:

```
DTHETAV
 20 20 30 40 4*50 / entire reservoir subtends 310 degrees
```

## DUALPERM **Run is to use dual permeability**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## DUMPCUPL Tells master run to write a reservoir coupling file

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [FMTOU](#) 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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

## DX

## X-direction grid block sizes

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
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)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $I^{\text{th}}$  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 ( $I$ ,  $1 - \text{NDIVIX}$ ,  $1 - \text{NDIVIX}$ ). See the `DIMENS` keyword for the specification of `NDIVIX` 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 /
```

ECLIPSE 300

# DY

## Y-direction grid block sizes

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## DYNAMICR Define regions based on when a set of conditions are satisfied

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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](#).

ECLIPSE 300

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	DYNAMICR is used to define a region family in the SOLUTION section, or once 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 <a href="#">Example 1</a> .
Dynamically - automatic	The DYNAMICR keyword is specified inside an ACTIONX keyword using the ALWAYS condition. Region values will be recomputed every timestep. See <a href="#">Example 2</a> .
Dynamically - conditional	The DYNAMICR 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 DYNAMICR 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 <a href="#">Example 3</a> .

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 DYNAMICR 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 DYNAMICR 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" /
  /
DYNAMICR
  FIPYYY  1  /
    "IN"  FIPXXX 1  /
  /
  FIPYYY  2  /
    "ALL" FIPXXX 2 FIPXXX 3 /
  /
  FIPYYY  3  /
    BOIP <= 500000 /
  /
ENDDYN
ENDACTIO
```

### 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.

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 /
  BOIP <= 400000 /
/
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 1/  
    "DIST_TO_WELLNAME" INJ 2 <= 10 /  
/  
ENDDYN
```

## DYNRDIMS

## Specify dimensions required for the dynamic region feature

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $J^{\text{th}}$  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- `NDIVX`,  $J$ , 1- `NDIVZ`). See the `DIMENS` keyword for the specification of `NDIVX` and `NDIVZ`.

`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 (/).

---

**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 `NDIVY = 11`, specified in `RUNSPEC` keyword `DIMENS`:

```
DYV
1000 500 7*200 500 1000 /
```

ECLIPSE 300

# DZ

## Z-direction grid block sizes

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $DZ \leq 10^{-6}$  for any cell, the cell will be either be made inactive or in thermal cases it may become rock-filled.

### Examples

#### Example 1

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
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
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

```

DZ
100*10 100*50 /
    
```

## DZMATRIX

# The vertical dimension of a block of matrix material

This is an alias for [DZMTRXV](#).

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## DZMTRX

## The vertical dimension of a block of matrix material

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $l_z$  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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1   4   1   2   1   1   /
DZMTRXV
  8*2.0 /

```

## DZNET

## Specifies the cell net DZ values

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$NTG_i = \frac{DZNET_i}{DZ_i} \quad \text{Eq. 3.64}$$

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

```

-- 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 /

```

## DZV

## Z-direction grid block sizes (vector)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $K^{\text{th}}$  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  $DZ \leq 10^{-6}$  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 /
```

ECLIPSE 300

## E

This section contains an alphabetic listing of keywords starting with the letter E.

## E100NOSI

**Enable NOSIM if incompatible E100 keyword**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [PLYVISC](#), is not compatible with ECLMC.

The keyword has no associated data.

See "[Brine tracking](#)" in the *ECLIPSE Technical Description*.

## EDITNNC

## Multiply a non-neighbor connection

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (IPT2)

DEFAULT: 0

*ECLIPSE 100 only*

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

*ECLIPSE 100 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

*ECLIPSE 100 only*

14. Diffusivity multiplier for the non-neighbor connection (DIFFM)

The diffusivity is set to  $DF * DIFFM$ , 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

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 `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 /
.
.
/

```

## EDITNNCR **Replace a non-neighbor connection**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
14. Diffusivity for the non-neighbor connection.  
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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

- 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
- Integer flag to select model for relative permeability hysteresis (capillary pressure hysteresis always uses the Killough model):

Water wet hysteresis models:

- Carlson's hysteresis model used for the non-wetting phase(s), drainage ([SATNUM](#)) curve used for the wetting phase.
- Carlson's hysteresis model used for the non-wetting phase(s), imbibition ([IMBNUM](#)) curve used for the wetting phase.
- Killough's hysteresis model used for the non-wetting phase(s), drainage ([SATNUM](#)) curve used for the wetting phase.
- Killough's hysteresis model used for the non-wetting phase(s), imbibition ([IMBNUM](#)) curve used for the wetting phase.
- Killough's hysteresis model used for both wetting and non-wetting phases.

Oil wet to water hysteresis models:

- Carlson's non-wetting model for gas and water phases (if present), drainage ([SATNUM](#)) curve used for the oil phase.
- Killough's non-wetting model for the gas and water phases (if present), drainage ([SATNUM](#)) curve used for the oil phase.
- Killough's hysteresis model used for non-wetting gas and water phases (if present) and the wetting oil phase.

Water wet hysteresis models:

- |                    |  |
|--------------------|--|
| <i>ECLIPSE 300</i> | 8 Jargon's hysteresis model used for the non-wetting phase(s), drainage ( <a href="#">SATNUM</a> ) curve used for the wetting phase.   |
| <i>ECLIPSE 300</i> | 9 Jargon's hysteresis model used for the non-wetting phase(s), imbibition ( <a href="#">IMBNUM</a> ) curve used for the wetting phase. |
| <i>ECLIPSE 100</i> | -1 Equilibration only option:  |

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.

---

ECLIPSE 300

DEFAULT: 2

ECLIPSE 100

DEFAULT: 0

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](#)).

ECLIPSE 100

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.

DEFAULT: RETR

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.

	water wet		oil wet	
item 8	G	O	G	O
	O	G	W	G
	G	O	O	W
	W	W	G	O

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](#).

## ECLIPSE 100

The default values will be used if 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.

## ECLIPSE 300

For the 2014.1 version, if hysteresis is specified via the HYSTER option of the SATOPTS keyword in the RUNSPEC section, the default values will be used if EHYSTR is not entered. Prior to the 2014.1 version, if the HYSTER option of SATOPTS was specified but EHYSTR omitted, no hysteresis was selected. Pre-2014.1 behavior can be restored by setting [item 320](#) of the [OPTIONS3](#) keyword to 1. Note that for ECLIPSE 300, the default value of item 2 is 2.

## Example

Killough's hysteresis model applied to both capillary pressure and relative permeability, for the non-wetting phase on a water-wet system:

```
EHYSTR  
0.07 2 1.0 0.1 BOTH RETR /
```

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:

```
EHYSTR  
0.07 7 1.0 0.1 KR RETR 6* 1 /
```

## EHYSTRR Hysteresis parameters by region

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## ENDACTIO **Indicates end of ACTION keyword sequence**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## ENDBOX

## Reset current input box to encompass the entire grid

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

## ENDFIN

## Terminates data for a local grid refinement

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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  3  4  5  5  8      5  /
PORO
200*0.2 /
EQUALS
  PERMX 500 /
  PERMY 500 /
  PERMZ 50 /
/
ENDFIN

```

## 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.

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## ENDNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## ENDSCALE Use saturation table end-point scaling

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**, **SGCR**, **SGU**, **SOWCR**, **SOGCR** 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**, **SGCR**, **SGU**, **SOWCR**, **SOGCR** 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. ENDSALE 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  /
```

## 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## ENKRVC

## Relative permeabilities at saturation end point versus composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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 NTSFUN=1, reducing the oil relative permeability in the presence of the nominated component.

```

ENKRVC
0.0 5* 1.0 1.0
1.0 5* 0.5 0.5 /

```

## ENKRVD

## Relative permeability end point versus depth tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, ENKRVDX-. 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
--
--          water  gas   oil   water  gas   oil   oil
-- Depth   Krw   Krg   Kro   at crit at crit at crit at crit
--          oil to oil to  water  gas   water
--          water  gas
--
3000.0  0.50  1.0  1.0  0.3  1*   1*   0.34
6000.0  0.58  1.0  1.0  1*   1*   1*   0.36
9000.0  0.66  0.95 0.95 0.35 1*   1*   0.37/

```

### Example 2

Old dataset conversion:

```

ENKRVD
-- Extra columns
--

```

3000.0	0.9	1*	0.9	4*	
4000.0	0.95	1*	0.95	4*	/

## ENKRVT

## Relative permeabilities at saturation end point versus temperature

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- Temperature values.  
Should increase monotonically down the column.  
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
- Maximum water relative permeability.
- Maximum gas relative permeability.
- Maximum oil relative permeability.
- Water relative permeability at critical oil saturation
- gas relative permeability at critical oil saturation
- Oil relative permeability at an oil saturation = 1 - minimum water saturation - critical gas saturation.
- 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* /
```

## ENPCVC

## Capillary pressures at saturation end point versus composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data consists of tables of capillary pressures versus composition, 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. 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/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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. The number of rows should not exceed the maximum number of entries in a quantity versus composition table, [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 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 [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, modifying the water capillary pressure in the presence of the nominated component.

```
ENPCVC
0.0 1* 45
1.0 1* 500/
```

## ENPCVD

## Maximum capillary pressure versus depth tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

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 [ENDNUM](#) in the REGIONS section, and the keywords [ENPTVD](#), [ENSPCVD](#), [ENKRVD](#) and [IMPCVD](#) in the PROPS section.

### Example

```

---- 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  /

```

## ENPCVT

## Capillary pressures at saturation end point versus temperature

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ENPTVC

## Saturation end-point versus composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 RPTRST, RPTSOL and RPTSCHED keywords.

UNITS: kg-M/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (PVT-M)

2. Connate water saturation.
3. Critical water saturation.
4. Maximum water saturation.
5. Connate gas saturation.
6. Critical gas saturation.
7. Maximum gas saturation.
8. Critical oil-in-water saturation
9. Critical oil-in-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.

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.

```
ENPTVC  
0.0 0.2 0.2 6*  
1.0 0.0 0.0 6*/
```

## ENPTVD

## Saturation end-point versus depth tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

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 [ENDNUM](#) 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  /

```

## ENPTVT

## Saturation end-point versus temperature

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
4. Maximum water saturation.
5. Connate gas saturation.
6. Critical gas saturation.
7. Maximum gas saturation.
8. Critical oil-in-water saturation.
9. 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 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## EOS

## Specify which equation of state is to be used

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
EOS
PR /
```

### Example 2

With three reservoir EoS regions:

```
EOS
P /
S /
S /
```

## EOSNUM

## Equation of state region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 9<sup>th</sup> 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 /
```

## EOSS

# Specify surface equation of state

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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*.

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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
EPSCHECK
```

## EPSCOMP

## End-point scaling component

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## EPSDBGS

## Controls debug for end-point scaling option

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$1 \leq IX1 \leq IX2 \leq \text{NDIVIX}$$

$$1 \leq JY1 \leq JY2 \leq \text{NDIVIY}$$

$$1 \leq KZ1 \leq KZ2 \leq \text{NDIVIZ} \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)}$$

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 EPSTDBGS or EPSTDEBUG 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 EPSTDBGS keyword is not available with the ODD3P relative permeability and capillary pressure hysteresis model.

ECLIPSE 300

The EPSTDBGS 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 EPSTDBGS 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:

- 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  $S_w = 1 - S_{owcr} - S_{gco}$  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  $SW = 1 - SOWCR - SGL$ .
- SOWL corresponds to the lower limit of the scaled oil-water saturation extent and is calculated according to  $SOWL = 1 - SWU - SGL$ .
- 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  $S_{ow} = 1 - S_{wcr} - S_{gco}$  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  $SOW = 1 - SWCR - SGL$ .
- 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 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  $S_{wmax} = 1 - S_{owco} - S_{gco}$  and  $S_{gmax} = 1 - S_{ogco} - S_{wco}$ . 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
-----
Unscaled Swco          0.100000    Scaled SWL            0.200000
Unscaled Swco-Pcw      0.100000    Scaled SWLPC         0.200000
Unscaled Swcr          0.100000    Scaled SWCR          0.554638
Unscaled Sw(krwr)      0.700000    Scaled SW(KRWR)     0.851546
Unscaled Sw max        1.000000    Scaled SWU           1.000000

Unscaled krwr          0.500000    Scaled KRWR          0.500000
Unscaled krw max       1.000000    Scaled KRW           1.000000

Unscaled Sow min       0.000000    Scaled SOWL          0.000000
Unscaled Sowcr         0.300000    Scaled SOWCR         0.087762
Unscaled Sow(krorw)    0.900000    Scaled SOW(KRORW)   0.800000
Unscaled Sow max       0.900000    Scaled SOWU          0.800000

Unscaled krorw         1.000000    Scaled KRORW         1.000000
Unscaled kro max       1.000000    Scaled KRO           1.000000

Unscaled Pcw max       20.000000   Scaled PCW           20.000000
-----
SWAT          SOIL          KRW          KROW          PCOW
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     0.002347     0.291774     1.300000
0.650000     0.350000     0.017419     0.224688     0.968750
0.700000     0.300000     0.059324     0.162490     0.725000
0.750000     0.250000     0.145113     0.109839     0.543750
0.800000     0.200000     0.284766     0.064823     0.375000
0.850000     0.150000     0.492971     0.027963     0.237500

```

0.851546	0.148454	0.500000	0.027268	0.234022
0.900000	0.100000	0.663522	0.005498	0.125000
0.912238	0.087762	0.704595	0.000000	0.098733
0.950000	0.050000	0.831271	0.000000	0.056250
1.000000	0.000000	1.000000	0.000000	0.000000

-----

## EPSDEBUG Controls debug for end-point scaling option

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 7<sup>th</sup> 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:

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword [DIMENS](#))}$$

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 [EHYSTR](#).

---

**Note:** It is recommended to use either the [EPSDEBUG](#) or [EPSDBGS](#) 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 [EPSDBGS](#) 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

```

EPSDEBUG
-- IX1 IX2   IY1 IY2   IZ1 IZ2   TAB  LGR  HYST
   4   4     5   5     1   3     0   1*   0/

```

## 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  $S_w = 1 - S_{owcr} - S_{gco}$  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  $S_{ow} = 1 - S_{wcr} - S_{gco}$  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  $SOW = 1 - SWCR - SGL$ .
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.

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  $S_{wmax} = 1 - S_{owco} - S_{gco}$  and  $S_{gmax} = 1 - S_{ogco} - S_{wco}$ . 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
-----
Unscaled Swco          0.100000      Scaled SWL             0.200000
Unscaled Swco-Pcw     0.100000      Scaled SWLPC           0.200000
Unscaled Swcr          0.100000      Scaled SWCR            0.554638
Unscaled Sw(krwr)     0.700000      Scaled SW(KRWR)        0.851546
Unscaled Sw max       1.000000      Scaled SWU             1.000000

Unscaled krwr         0.500000      Scaled KRWR            0.500000
Unscaled krw max      1.000000      Scaled KRW             1.000000

Unscaled Sow min      0.000000      Scaled SOWL            0.000000
Unscaled Sowcr        0.300000      Scaled SOWCR           0.087762
Unscaled Sow(krorw)   0.900000      Scaled SOW(KRORW)     0.800000
Unscaled Sow max      0.900000      Scaled SOWU            0.800000

Unscaled krorw        1.000000      Scaled KRORW           1.000000
Unscaled kro max      1.000000      Scaled KRO              1.000000

Unscaled Pcw max      20.000000     Scaled PCW             20.000000
-----

SWAT      SOIL      KRW      KROW      PCOW
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  0.002347  0.291774  1.300000
0.650000  0.350000  0.017419  0.224688  0.968750
0.700000  0.300000  0.059324  0.162490  0.725000
0.750000  0.250000  0.145113  0.109839  0.543750
0.800000  0.200000  0.284766  0.064823  0.375000
0.850000  0.150000  0.492971  0.027963  0.237500
0.851546  0.148454  0.500000  0.027268  0.234022
0.900000  0.100000  0.663522  0.005498  0.125000
0.912238  0.087762  0.704595  0.000000  0.098733
0.950000  0.050000  0.831271  0.000000  0.056250
1.000000  0.000000  1.000000  0.000000  0.000000
-----

```

## EPSODD3P End-point and process scaling control for ODD3P option

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- Oil saturation end-point Land process scaling control.
  - Deselect oil saturation Land process end-point scaling.
  - Select oil saturation Land process end-point scaling.

DEFAULT: 0

For miscible runs, a value of 1 is recommended.

- Gas saturation end-point Land process scaling control.
  - Deselect gas saturation Land process end-point scaling.
  - Select gas saturation Land process end-point scaling.

DEFAULT: 0

For miscible runs, a value of 1 is recommended.

- Water saturation end-point Land process scaling control.
  - Deselect water saturation Land process end-point scaling.
  - Select water saturation Land process end-point scaling.

DEFAULT: 0

For miscible runs, a value of 1 is recommended.

- Capillary pressure scaling control.
  - Deselect capillary pressure scaling.
 

If a value of 0 is specified, no capillary pressure scaling is performed.

- 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.

- 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

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.

6. Gas-oil surface 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.

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.

8. Gas-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.

9. Gas-oil 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.

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.

11. Gas-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.

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 be 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 Lg 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*/
/
/
.....

```

## EQLDIMS

## Dimensions of equilibration tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

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](#).

DEFAULT: 20

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).

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 /
```

## EQLDKVCR Equilibrium deviation reaction k-values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$K(P, T) = (A + B / P + C \cdot P) \cdot e^{-D/(T-E)}$$

where

$T$  is the temperature (K (METRIC), °R (FIELD), K (LAB), K (PVT-M))

$P$  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:

```

EQLDKVCR
-- K1  K2
1.23E6  212
833.4E6  155.4E3
0  0
16000  4000

```

0 480  
/

## EQLDREAC Equilibrium deviation terms for reactions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

Equilibrium deviation terms can be used to model deviation from an equilibrium state. A reaction rate is multiplied by a term:

$$|d|^n, \quad d > 0$$

$$\text{or}$$

$$-|d|^n, \quad d < 0$$

where  $n$  is the order of the equilibrium deviation term and  $d$  is given by one of the following models:

$$d = \theta \cdot b_o \cdot S_o \cdot (x_i - y_j / K_k), \quad m = 1 \quad \text{Eq. 3.65}$$

$$d = \theta \cdot b_g \cdot S_g \cdot (K_k \cdot x_i - y_j), \quad m = 2 \quad \text{Eq. 3.66}$$

$$d = \theta \cdot b_w \cdot S_w \cdot (a_i - 1 / K_k), \quad m = 3 \quad \text{Eq. 3.67}$$

$$d = \theta \cdot (F_k(a_i) - C_a), \quad m = 4 \quad \text{Eq. 3.68}$$

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

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 ( $x_i$  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 1 2 1 BOTH /
  2 1 3 0 2 BOTH /
```

## EQLDTAB

## Equilibrium values for water-solid deviation reactions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 NPPVT (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
/

```

## EQLNUM

## Equilibration region numbers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [PVTNUM](#)). 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#)).

**IRREVER** 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.

```
EQLOPTS
THPRES  IRREVERS /
```

## EQLZCORN **Reset part of the corner point depth array**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$0 \leq \text{IX1} \leq \text{IX2} \leq \text{NDIVIX}$$

$$0 \leq \text{JY1} \leq \text{JY2} \leq \text{NDIVIY}$$

$$0 \leq \text{KZ1} \leq \text{KZ2} \leq \text{NDIVIZ} \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword } \text{DIMENS})$$

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](#):

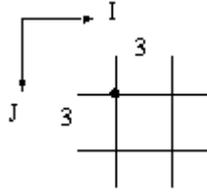
```

.....
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/
.....

```

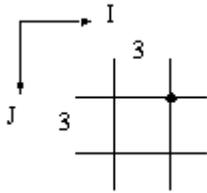
### Example 2

Set the top-left corner of cell (3,3,1) to 10890.0 ft



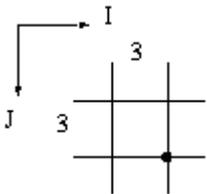
```
EQLZCORN
10890.0      3 0 3 0 1 1 /
```

Set the top-right corner of cell (3,3,1). Modifying the top corner depth leaving the adjacent cells unmodified.



```
10890.0      0 3 3 0 1 1 3 3 3 3 /
```

Set the lower-right corner of cell (3, 3, 5). Modifying the bottom corner depth leaving the adjacent cells unmodified.



```
10890.0      0 3 0 3 5 5 3 3 3 3 BOTTOM /
```

## EQUALREG Sets an array to a constant value in a region

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

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*

FLUXNUM            MULTNUM            MPFANUM

*ECLIPSE 300 only*

MIDS

**EDIT section arrays**

PORV                    DEPTH

TRANX (TRANR)    TRANY (TRANHT)    TRANZ

DIFFX (DIFFR)    DIFFY (DIFFHT)    DIFFZ

HEATX (HEATR)    HEATY (HEATHT)    HEATZ (HEATZ)

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

*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

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      PERMZ      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.
- 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:

Keyword	Description
<a href="#">DIFFMMF</a>	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.
<a href="#">MULSGGD</a> <a href="#">MULSGGDV</a>	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.
<a href="#">MULTSIG</a> <a href="#">MULTSIGV</a>	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.
<a href="#">MULTX</a> <a href="#">MULTY</a> <a href="#">MULTZ</a>	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.

Keyword	Description
PERMX PERMY PERMZ	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.
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 SIGMAV	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.
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 3.14: Additional notes for EQUALREG in the SCHEDULE section

## Examples

### Example 1

NX=6, NY=8, NZ=1:

```

FLUXNUM
1 1 2 2 2 2
1 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
4 4 4 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:

```

MULTNUM
1 1 2 2 2 2
1 1 1 2 2 2 /
EQUALREG
PERMX 0.1 1 /
PERMX 0.15 2 /
/

```

## EQUALS

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$1 \leq IX1 \leq IX2 \leq NDX$$

$$1 \leq JY1 \leq JY2 \leq NDY$$

$$1 \leq KZ1 \leq KZ2 \leq NDZ,$$

where NDX, NDY, NDZ are the limits in the current BOX.

See also keywords [BOX](#), [ENDBOX](#), [ADD](#), [MULTIPLY](#) and [COPY](#).

### 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  
 FLUXNUM                  MULTNUM                  MPFANUM

*ECLIPSE 100 only*

MULTPV

*ECLIPSE 300 only*

MIDS                      PERMMF

**EDIT section arrays**

PORV                      DEPTH  
 TRANX (TRANR)          TRANY (TRANHT)          TRANZ  
 DIFFX (DIFFR)          DIFFY (DIFFTHT)          DIFFZ

*ECLIPSE 100 only*

MULTPV

*ECLIPSE Thermal only*

HEATTX (HEATTR) HEATTY (HEATTTHT) HEATTZ (HEATTZ)

*ECLIPSE Thermal only*

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 SURFNUM

*ECLIPSE 300 only*

EOSNUM

**SOLUTION section arrays**

PRESSURE SWAT          SGAS  
 RV                      RS                      TBLK  
 GI                      OILAPI                  SALT  
 GASCONC                  SOLVCONC                  SOLVFRAC

*ECLIPSE 100 only*

*ECLIPSE 100 only*

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 [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 /
      PORV    1000     / defaults to last specified box
/
    
```

## EQUIL

## Equilibration data specification

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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  $R_S$  or  $R_V$  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)

DEFAULT: 0.0

*ECLIPSE 100 only*

Default not defined

*ECLIPSE 300 only*

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

Thermal only

---

**Note:** If the water-oil contact lies above the reservoir and if, based on the composition provided at the datum depth, oil is heavier than water, then the pressure in item 2 refers to the oil phase and the reservoir will be initialized with heavy oil. A non-zero value for `OPTIONS3` item 217 can be used to restore pre-2010.1 behavior for thermal heavy oil equilibration.

---

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

ECLIPSE 100 only

5. Depth of the gas-oil contact.

This value is ignored in single-phase, oil-water and gas-water problems.

---

**Note:** The contact should lie above the top of the reservoir if there is no initial free gas. The contact should lie below the bottom of the reservoir if there is only free gas (with vaporized oil) initially present. The values of the gas saturation above and below the transition region are set at the end-point saturations in the gas capillary pressure table.

In three-phase gas condensate problems in which there is no initial free oil (that is all the oil is initially contained in the gas phase), the gas-oil contact should be at the same depth as the water-oil contact. It then acts as a gas-water contact, and the initial solution is calculated in a manner analogous to a gas-water problem.

---

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

Black Oil cases only

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.

ECLIPSE 100

---

**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  $|P_{datum} - P_{sat}| < \epsilon$  where  $\epsilon$  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

$$S_{g,max} = 1 - S_{w,min} \quad \text{Eq. 3.69}$$

A residual oil saturation ( $S_{o,res}$ ) in the gas cap can be produced by setting

$$S_{g,max} = 1 - S_{w,min} - S_{o,res} \quad \text{Eq. 3.70}$$

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

$$P_{oil} = P - P_{cog} \times \left( \frac{S_g}{S_o + S_g} \right)$$

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 [NOPCSEPT](#). 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 NTEQUL=2:

```
EQUIL  
7100 4000 8000 0.0 6500 0.0 1 0 -3 /  
7000 4000 7800 0.0 7000 0.0 /
```

## ESPNODE

## Salt concentration nodes for polymer viscosity multiplier

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), or

UNITS: sm<sup>3</sup>/kg (METRIC), stb/lb (FIELD), scc/gm (LAB), or

UNITS: dimensionless.

## ESSNODE **Salt concentration nodes for water/oil surface tension**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), or

UNITS: dimensionless.

## EXCAVATE Remove grid blocks during a run

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

## EXCEL

## Requests run summary output to be in Excel format

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
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).
7. NFLOG Maximum number of global flows into or within the refinement.
8. NUMINT Maximum number of internal local flows in the refinement crossing boundaries between global cells.
9. NUMCON Maximum number of global-local connections.
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 [REFINE](#), [EXTFIN](#), [CARFIN](#), [RADFIN](#) or [RADFIN4](#) keyword is encountered.

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 /
    
```

*ECLIPSE 100 only*

*ECLIPSE 300*

## EXTHOST

## Specify parent cells for LGR cells in an external local grid refinement

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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 /
```

## EXTREPGL Specify replaced global cells for an external local grid refinement

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
EXTREPGL
-- NHOST=5
  2 5 9 4 15 /
```

## **F**

This section contains an alphabetic listing of keywords starting with the letter F.

## FACTLI

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$T_{\text{crit}}^{\text{mix}} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j} \quad \text{Eq. 3.71}$$

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  $T_{\text{crit}}^{\text{mix}}$ . 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 [COMPVD](#).

---

## 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 /
```

## FASTTRAC

**Enable shortcuts in tracer calculations**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
FAULTDIM
3 /
```

## FAULTS

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

*ECLIPSE 100*

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

```

FAULTS
--
'zigzag'  1  1  1  2  1  4  X /
'zigzag'  2  2  2  2  1  4  Y /
'zigzag'  2  2  3  4  1  4  X /
'zigzag'  3  3  4  4  1  4  Y /
'zigzag'  3  3  5  6  1  4  X /
'zigzag'  4  4  6  6  1  4  Y /
'zigzag'  4  4  7  8  1  4  X /
'zigzag'  5  5  8  8  1  4  Y /
'zigzag'  5  5  9 10  1  4  X /
'zigzag'  6  6 10 10  1  4  Y /
'block'   19 19 20 30  1  4  X /
'block'   30 30 20 30  1  4  X /
'block'   20 30 19 19  1  4  Y /
'block'   20 30 30 30  1  4  Y /
/

```

ECLIPSE 100

For the above example, a value of 14 has been specified in the `FAULTDIM` keyword in the `RUNSPEC` section.

## FCCRIT

## Flash convergence criteria

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)

2. Tortuosity of the porous media,  $\delta$ .

UNITS: dimensionless

DEFAULT:  $\text{SQRT}(2) = \sqrt{2} = 1.414$

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.  δ      n      τ0
  0.0   1.414  1.0    0.0
 10.0   1.414  0.6    0.5 /
```

## FIELD

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

## FIELDSEP Introduces a field separator

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 5<sup>th</sup> 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 8<sup>th</sup> 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*

DEFAULT: 0

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*.

9. Temperature for natural gas liquid (NGL) density evaluation.

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.

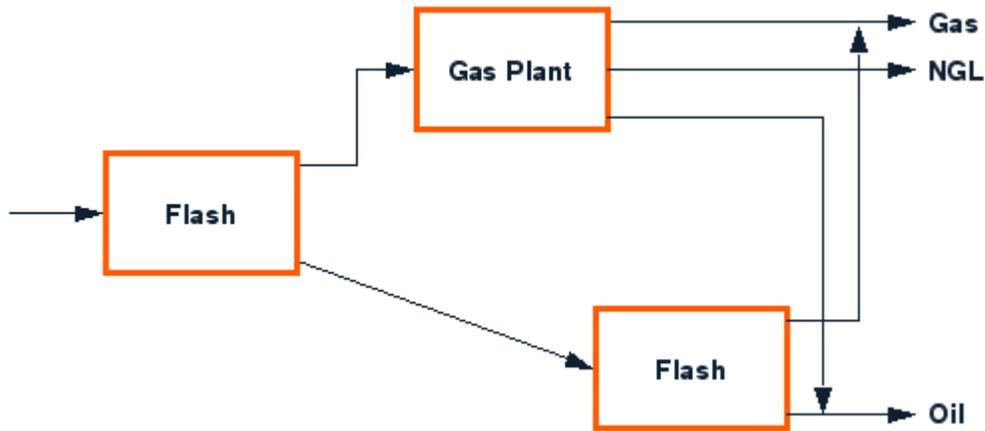


Figure 3.5. A complex field separator configuration

```
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
/
```

## FILEUNIT

## Confirm units of a data set

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 nnnnn 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

```

```
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4
1 1 2 2 3 3 4 4 /
-- 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
8*1 16*2 16*3 8*4 /
```

## FIPNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

---

ECLIPSE 100 only

See also the keyword [FIPOWG](#).

### Example

With NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword [DIMENS](#), and no input BOX set:

```
FIPNUM
40*2 8*1
40*3 8*1
40*4 8*1
40*5 8*1
40*6 8*1 /
```

## FIPOWG

## Sets up a family of FIP regions representing the original GAS, OIL and WATER zones

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,

```

ROIPOWG   for the oil-in-place in the liquid phase for the original oil zone.
2 /
ROIPOWG   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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 8<sup>th</sup> 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.

```
FIPSEP
1 1 80 815 /
1 2 60 14.7 /
2 1 60 14.7 /
3 1 90 730 /
3 2 80 345 /
3 3 60 14.7 /
/
```

## FLUXNUM **Identifies extent of each flux region**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
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 /
```

## FLUXOPTS Options for flux boundary runs

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## FLUXREG

**Identifies which flux regions are active**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## FLUXTYPE Specify type of flux boundary condition

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $R_s$  and  $R_v$  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' /
```

## FMTHMD

**Indicates that HMD file is to be formatted**

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## FMTOUT

### Indicates that output files are formatted

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#).

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## FOAM

### Enable FOAM option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## FOAMADS Foam adsorption functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

If item 1 of [FOAMOPTS](#) is set to 'WATER':

UNITS: kg/sm<sup>3</sup> (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 /

```

## FOAMDCYO Foam decay data as a function of oil saturation

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
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 /
```

## FOAMDCYW Foam decay data as a function of water saturation

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## 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 ([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:

$$F_c = \left( \frac{N_c^r}{N_c} \right)^e \tag{Eq. 3.72}$$

where

$N_c$  is the capillary number.

$N_c^r$  is the reference capillary number.

$e_c$  is an exponent which controls the steepness of the transition about the point where  $N_c = N_c^r$ .

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  $N_c = N_c^r$ . 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 [NTSFUN](#)=5

```
FOAMFCN
1.0E-07 /
2.0E-07 1.5 /
/
1.0E-07 3.0/
/
```

## FOAMFRM

## Gas mobility reduction dependence upon reference mobility reduction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

75 /  
/  
55 /  
60 /

## FOAMFSC

## Gas mobility reduction dependence upon foam surfactant concentration

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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(/):

- The reference foam surfactant concentration above which a strong foam can form.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)
- An exponent controlling the steepness in the change of mobility reduction due to surfactant concentration.  
DEFAULT: 1.0
- The minimum surfactant concentration for which gas mobility reduction due to the presence of foam will be calculated.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)  
DEFAULT: 10<sup>-20</sup>
- The minimum water saturation below which foam has no effect.  
DEFAULT: 10<sup>-6</sup>

ECLIPSE 300

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.

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. 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:

$$F_s = \left( \frac{C_s}{C_s^r} \right)^{e_s}$$

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.

$e_s$  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  $e_s < 1$ , 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 /
```

## FOAMFSO

## Gas mobility reduction dependence upon oil saturation

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## 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 ([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  /
```

## FOAMFST

## Gas-water surface tension versus foam surfactant concentration

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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      0.000057
```

```
3.5      0.000028
5.25     0.000011
7.0      0.0000057
8.75     0.0000006 /
```

### Example 3

With NTSFUN=1 and NSSFUN  $\geq$  5 and Lab units:

```
FOAMFST
--
-- Conc   ST
-- gm/scc dyne/cm
----
0.00     10.0
0.01     5.0
0.015    2.0
0.02     1.0
0.025    0.1 /
```

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$F_w = 0.5 + \frac{\text{atan}(f_w(S_w - S_w^l))}{\pi}$$

where

$S_w$  is the water saturation.

$S_w^l$  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  $S_w = S_w^l$ . 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^1   f_w
   0.3     1E3 /
   0.31    1E3 /
/
   0.32    2e4 /
   0.32    2e4 /
```

## FOAMMOB Gas mobility reduction data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

If item 1 of [FOAMOPTS](#) is set to 'WATER':

UNITS: kg/sm<sup>3</sup> (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:

```

FOAMMOB
--
-- Concentration      Mobility factor
--
      0.00              1.0
      0.01              0.2
      0.015             0.1
      0.02              0.09
      0.025             0.09 /

```

## FOAMMOBP Pressure dependence of foam mobility reduction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- The oil phase pressure.  
The values should increase monotonically down the column.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
- 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.

$$MP = (1 - M(C_{\text{foam}})) \cdot M_p(P) + M(C_{\text{foam}})$$

where

$MP$  is the mobility reduction factor with the pressure effect

$M(C_{\text{foam}})$  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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
FOAMOPTS
WATER FUNC /
```

## FOAMROCK Specifies the foam-rock properties

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/rm<sup>3</sup> (METRIC), lb/rb (FIELD), gm/rcc (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 adsorped 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

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $K_{rog}$  has to equal  $K_{row}$  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  $K_{rg}$  equal  $K_{ro}$  at connate water, which is not an unreasonable assumption. Alternatively, the HCSCAL option scales  $K_{rg}$  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 `NOPCSEPT` 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 `NOPCSEPT` 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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## FRICITION

### Enables the wellbore friction option

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
FRICITION
3 2 /
```

## FULLIMP

## The fully implicit solution option

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$F_K = \left( \frac{\sigma}{\sigma_0} \right)^N \quad \text{Eq. 3.73}$$

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:

$$K_r = F K_r^{\text{imm}} + (1-F) K_r^{\text{mis}} \quad \text{Eq. 3.74}$$

The two contributions are scaled so that they both have the same critical saturation, given by  $FS_{\text{cr}}^{\text{imm}}$ .

FVST avoids the use of both  $P$  and  $\sigma_0$  in relative permeability calculations.  $\sigma_0$  is still used in modifying capillary pressures:

$$P_{\text{cog}}^{\text{corr}} = \left( \frac{\sigma}{\sigma_0} \right) P_{\text{cog}} \quad \text{Eq. 3.75}$$

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.  
UNITS: dynes/cm (METRIC), dynes/cm (FIELD), dynes/cm (LAB), dynes/cm (PVT-M)
2. The miscibility factor  $F_K$ .  
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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## **G**

This section contains an alphabetic listing of keywords starting with the letter G.

## GADJUST

**Adjust diagonal permeability tensor**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## GADVANCE Assign advance import gas to a group

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

- 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.
- The name of the well stream (defined using the [WELLSTRE](#) keyword) which defines the composition of the advance gas.
- The maximum rate at which advance gas can be supplied.

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

By setting [OPTIONS3item 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 /
/
```

## GAS

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to 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 (\*).

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 NDIVIX=5, NDIVIY=4, NDIVIZ=10 and NCOMPS=2

```
GASADCO
--matrix fracture
100*86.0 100*0.0 --component 1
100*43.0 100*0.0 --component 2/
```

## GASADEC Initial equilibrium coal gas concentration

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to 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 (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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GASMONTH](#). 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:

Restriction	Keyword	Description
	BRANPROP	to change branch properties when using the extended network model
E100	DEBUG	to change debug switches.
E300	DEBUG3	to change debug switches.
Compulsory	GASEND	to mark the end of the annual scheduling file.
	GASMONTH	to indicate the time of the next set of operations.
	GCONPRI	to change group limits under prioritization group control.
	GCONPROD	to change group limits and group guide rates under guide rate group control.
E100	GCONSUMP	to change group gas consumption rates.
E300	GRUPFUEL	to change group gas fuel rates.
	GRUPNET	to change the VFP table or ALQ value on network branches, or to change the pressure of an existing fixed pressure node.
	GRUPTARG	to change individual group rate limits or guide rates.
	GRUPTREE	to change the group hierarchy
	GSATINJE	to change injection rates of satellite groups.
	GSATPROD	to change production rates of satellite groups.
E100	NCONSUMP	to change gas consumption at nodes in the extended network model.
	NEXTSTEP	to limit the length of the next timestep, for example to reduce it after some wells have been opened.
	NODEPROP	to change nodal properties in the extended network model.
E300	RPTPRINT	to change switches for printed reports.
E100	RPTSCHED	to change switches for printed reports. The following report switches will be ignored here: <ul style="list-style-type: none"> <li>'RESTART' - restart files are only written at the end of a contract period</li> <li>'CPU' - the timer switch setting must not be altered within the annual scheduling file.</li> </ul>
E300	TSCRIT	to modify timestepping criteria.
	TUNING	to modify tuning parameters, for example maximum step size.
	WEFAC	to change well efficiency factors.
	WELDEBUG	to change well debug switches.
	WELOPEN	to open or close wells or connections.
	WELPRI	to change well priorities for prioritization group control.
	WELTARG	to change well operating limits.

Restriction	Keyword	Description
	WGRUPCON	to change well guide rates for guide rate group control.

## Example

For a contract year beginning on 1<sup>st</sup> 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 1<sup>st</sup> Jan., and request a report

```
GASMONTH
JAN YES /
```

Advance the simulation to 1<sup>st</sup> June, without a report

```
GASMONTH
JUN NO /
```

From 1<sup>st</sup> 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 1<sup>st</sup> 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 /
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## GASCONC Initial coal gas concentration

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 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 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.

See the [GCVD](#) keyword.

ECLIPSE 100

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 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 [GASADCO](#).

### Example

With NDIVIX=8, NDIVIY=6, NDIVIZ=10 specified in RUNSPEC keyword [DIMENS](#):

```
GASCONC
48*89.1 48*87.3 48*85.1 48*85.0 48*85.0 /
```

## 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## GASFCOMP Automatic compressors for standard network option

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GMCPL](#). 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 (/).

- 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.
- 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
- 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](#).

DEFAULT: 0.0

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: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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     /
/
```

## GASFDECR Decrement for FIELD gas contract

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

The GASFDECR 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  $DCQ \times PROFILE - DECREMENT$ .

The GASFDECR can be used in conjunction with the GASFTARG keyword in which case the field is produced at the lesser of  $DCQ \times PROFILE - DECREMENT$  and the rate specified by the GASFTARG data items.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 GASFDECR keyword is followed by 12 numbers, terminating with a slash (/). The decrements for defaulted months are set to zero.

### Example

```
GASFDECR
-- JAN FEB MAR APR MAY JUN JUL AUG SEP OCT NOV DEC
  2*1000.0 3*          2*1000.0 3*          2*500.0 /
```

## GASFDEL C Delivery capacity definition for Gas Field Operations Model

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `GASFDEL C` 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

```
GASFDEL C
2 /
```

## GASFIELD Enables special options in the Gas Field Operations Model

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## GASFTARG Production rate for FIELD gas contract

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $DCQ \times PROFILE$  and the specified rate. For months in which GASFTARG is defaulted the field is produced at  $DCQ \times PROFILE$ . The GASFTARG keyword can also be used in conjunction with the [GASFDECR](#) keyword.

UNITS:  $sm^3/day$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $sm^3/day$  (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

```

GASFTARG
-- JAN  FEB  MAR  APR  MAY  JUN  JUL  AUG  SEP  OCT  NOV  DEC
   2*50000.0  3*                2*20000.0  3*                2*50000.0  /

```

## GASMONTH Sets month pointer in annual scheduling file

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 1<sup>st</sup> September:

```
GASBEGIN
GASMONTH
SEP Y /
```

(The GASMONTH keyword was not necessary here. The pointer is already at 1<sup>st</sup> 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 1<sup>st</sup> September.

```
Scheduling keywords...
```

Advance pointer to 1<sup>st</sup> December, the beginning of the winter quarter. No report is required.

```
GASMONTH
DEC N
```

Enter actions to be performed on 1<sup>st</sup> December.

```
Scheduling keywords...
```

Advance pointer to 1<sup>st</sup> March, the beginning of the spring quarter. Print a report for the timestep ending 1<sup>st</sup> March, the end of the winter quarter.

```
GASMONTH  
MAR Y /
```

Enter actions to be performed on 1<sup>st</sup> March.

```
Scheduling keywords...
```

Advance pointer to 1<sup>st</sup> June, the beginning of the summer quarter. No report is required.

```
GASMONTH  
JUN N /
```

Enter actions to be performed on 1<sup>st</sup> June.

```
Scheduling keywords...
```

End the annual scheduling file.

```
GASEND
```

---

**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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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  $DCQ \times \text{profile}$  to  $DCQ \times \text{swing}$  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  $\times$  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

(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 ( $DCQ \times FLIM$ ) 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  $DCQ_2$ . 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  $DCQ = DCQ_2 \times FANT$ .

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 GASPERIO.

## 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

```
GASPERIO
-- No of   Period   Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no   Conv
-- periods Length   DCQ      req     t/steps   reduc     reduc     DCQ its  tol
      10      3      100000   PER     1*        0.1      1*        4       0.05 /
```

### 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.

```
GASPERIO
-- No of   Period   Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no   Conv
-- periods Length   DCQ      req     t/steps   reduc     reduc     DCQ its  tol
      2      6      1*        PER     1*        0.1      1*        4       0.05 /
```

## GASSATC

## Initial saturated coal gas concentration

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

Grid blocks are ordered with the X axis index cycling fastest (from 1 to 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 NDIVIX=8, NDIVIY=6, NDIVIZ=10 specified in RUNSPEC keyword [DIMENS](#):

```
GASSATC
48*89.1 48*87.3 48*85.1 48*85.0 48*85.0 /
```

## GASSOL

## Specify number of gas components to dissolve in the aqueous phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## GASVISCF Gas viscosity function

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword can only be used with the [THERMAL](#) option.

For the  $i^{\text{th}}$  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 9<sup>th</sup> parameter of [TABDIMS](#).

### Example

```
GASVISCF
-- SGAS HEAVY
0.5E-4 1.0E-4
0.9 0.9
/
```

## GASVISCT Gas viscosity versus temperature table

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
GASVISCT
 75 0.0143 0.0180
100 0.0149 0.0188
150 0.0161 0.0203
200 0.0172 0.0218
250 0.0184 0.0233
300 0.0196 0.0247
350 0.0207 0.0262
500 0.0241 0.0305
15000 0.2946 0.3723
/
```

## GASWAT

### Use water-gas model

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 × profile to DCQ × 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 × DCQ).

JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC

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 ( $DCQ \times FLIM$ ) 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.

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*.

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.

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 year 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 will start the first pass of the next contract year with an initial  $DCQ = DCQ2 \times FANT$ .

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.

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 is pessimistic if there has been a significant reduction.

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.

ECLIPSE 100

If you are not using the accelerated iteration scheme this item is ignored.

RANGE: 0.0 to 1.0

DEFAULT: 0.1

End the data with a slash (/).

See also the [GASPERIO](#) 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.

```
GASYEAR
-- No of      Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no   Conv
-- years      DCQ         req     t/steps   reduc     reduc     DCQ its  tol
--           4      100000   YEAR    1*        0.1      1*       4      0.05 /
```

### Example 2

To continue the simulation for another two years with the same DCQ iteration parameters. The initial DCQ should be defaulted now.

```
GASYEAR
-- No of      Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no   Conv
-- years      DCQ         req     t/steps   reduc     reduc     DCQ its  tol
--           2      1*       YEAR    1*        0.1      1*       4      0.05 /
```

### Example 3

To advance the simulation through one year with the DCQ fixed at 50000, and timesteps limited just by rate changes.

```
GASYEAR
-- No of      Initial   Swing   Monthly   Lim DCQ   Ant DCQ   Max no   Conv
-- years      DCQ         req     t/steps   reduc     reduc     DCQ its  tol
--           1      50000   NO      NO        /
```

## GCALECON Energy and calorific value economic limit data for groups

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/sm<sup>3</sup> (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm<sup>3</sup> (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/  
/
```

## GCONCAL **Calorific value controls for groups**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm<sup>3</sup> (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 /
/
```

## GCONENG Energy production rate control for groups

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

UNITS: 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

```
GCONENG
FIELD 1.0E9 /
```

## GCONINJE Injection rate controls/limits for groups/field

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 300

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 100 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 group/field equals its production voidage rate times the voidage replacement fraction specified in item 7.

ECLIPSE 300 only

WGRA The group/field wet gas injection rate of the phase in item 2 is controlled to meet the target specified in item 13.

FLD 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: sm<sup>3</sup>/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm<sup>3</sup>/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: rm<sup>3</sup>/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm<sup>3</sup>/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 [GRUPFUEL](#)) 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 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 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

10. Definition of the guide rate in item 9.

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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

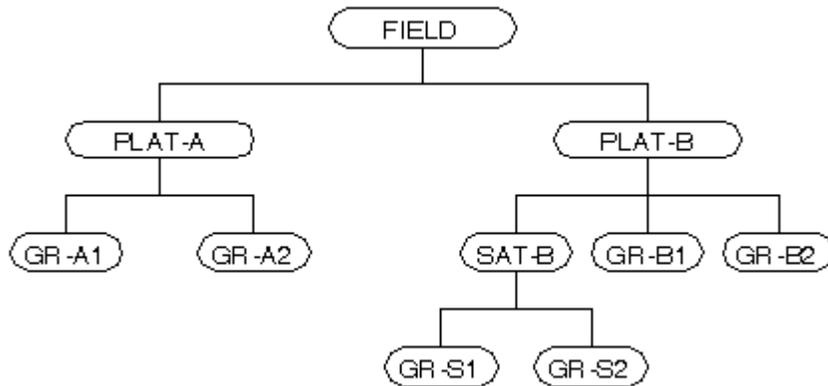


Figure 3.6. A five level hierarchy

#### 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 /
/
    
```

### 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

ECLIPSE 100

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 /
/
```

## GCONPRI

## Group/field production limits for prioritization

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- 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.
- Oil production rate upper limit. User defined arguments can be set with keywords `UDQ` and `UDADIMS`.

UNITS:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`PVT-M`)

DEFAULT: No limit

- 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`)

`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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{Mscf}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`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:  $\text{rm}^3/\text{day}$  (`METRIC`),  $\text{rb}/\text{day}$  (`FIELD`),  $\text{rcc}/\text{hr}$  (`LAB`),  $\text{rm}^3/\text{day}$  (`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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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
- Guide rate definitions VOID or NETV in keyword `GCONINJE` item 10
- Sales gas control (keyword `GCONSALE`), which uses reinjection.
- Advanced or Make-up gas (keywords `GADVANCE` or `GINJGAS`)
- Injection limited by availability (keyword `WAVAILIM`).

ECLIPSE 100 only

ECLIPSE 100 only

ECLIPSE 300 only

ECLIPSE 300 only

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 /
/
```

## GCONPROD Production rate controls/limits for groups or field

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

*ECLIPSE 300 only*

WGRA Group/field wet gas production rate is controlled to meet the target specified in item 16

*ECLIPSE 300 only*

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.

*ECLIPSE 300 only*

PBGS Group/field surface gas production rate is controlled to meet the target specified in item 18

*ECLIPSE 300 only*

PBWS Group/field surface water production rate is controlled to meet the target specified in item 19

FLD 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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 [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 [COMPLUMP](#) and [WORKLIM](#) for ECLIPSE 100.

ECLIPSE 300

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.

<i>ECLIPSE 300 only</i>	WGA The group's guide rate applies to the wet gas production.
<i>ECLIPSE 300 only</i>	CVAL The group's guide rate applies to the calorific value production.
<i>ECLIPSE 100 only</i>	<p>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.</p> <p>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 <i>ECLIPSE Technical Description</i>. 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.)</p> <p>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.)</p> <p>' ' 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.</p>

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:  $\text{rm}^3/\text{day}$  (METRIC),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{rm}^3/\text{day}$  (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: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), 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
- Guide rate definitions (VOID or NETV in keyword [GCONINJE](#)) item 10
- Sales gas control (keyword [GCONSALE](#)), which uses reinjection
- Advanced or Make-up gas (keywords [GADVANCE](#) or [GINJGAS](#))
- Injection limited by availability (keyword [WAVAILIM](#)), because these facilities require the producers to be solved before the injectors.

*ECLIPSE 100 only*

*ECLIPSE 100 only*

*ECLIPSE 300 only*

*ECLIPSE 300 only*

*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

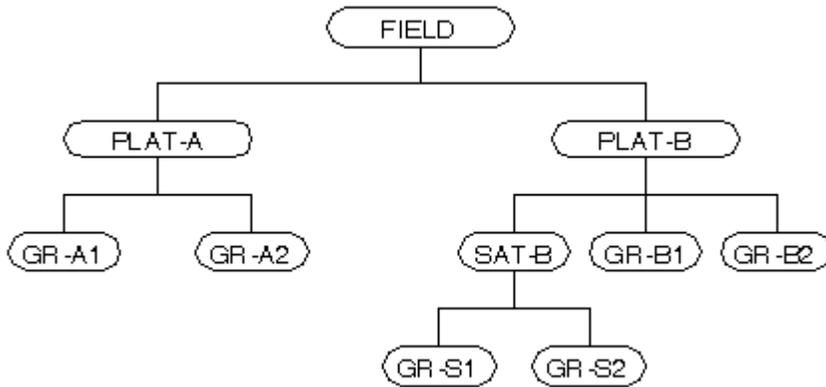


Figure 3.7. A five level hierarchy

### 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 /
GR-A1 NONE 5* NO /
/

```

### 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.

```

GCONPROD
FIELD ORAT 10000 /
'PLAT*' FLD 1* 6000 20000 1* CON 1* 1000 LIQ /
SAT-B NONE 1* 4000 10000 1* CON /
GR-A1 NONE 5* NO /
GR-B1 ORAT 2000 4* NO /
/

```

### Example 6

ECLIPSE 100

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.

```

GCONPROD
FIELD PRBL 12* 1.0 /
'PLAT*' FLD 7* INJV /
/

```

## GCONSALE Sales gas production controls for groups/field

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: Infinity (1.0E20)

4. Minimum permitted sales gas production rate. User defined arguments can be set with keywords [UDQ](#) and [UDADIMS](#).

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 WELSP ECS). 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 /
/
```

## GCONSUMP Gas consumption and import rates for groups

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GCONINJE](#)) and sales gas control (see keyword [GCONSALE](#)).

Under gas reinjection control, the gas injection rate is (gas injection rate of group) = (the group's reinjection fraction) \* [(the group's gas production rate) + (the gas import rate to the group and any subordinate groups) - (the gas consumption rate of the group and any subordinate groups)]

The sales gas production rate is calculated as (sales gas production rate) = (gas production rate) - (gas injection rate) + (gas import rate) - (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 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 [UDQ](#) and [UDADIMS](#).

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: Zero

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 [UDQ](#) and [UDADIMS](#).

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: Zero

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).

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

```
GCONSUM
  PLAT-A  20 50  /
  PLAT-B  15   /
/
```

## GCONTROL Tolerance for group control targets

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 GCONTROL 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 (/).

*ECLIPSE 100 only*

1. 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 GCONTROL or NUPCOL.

DEFAULT: Unchanged.

*ECLIPSE 300 only*

3. 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.

*ECLIPSE 300 only*

4. 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 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. (This can be set at any time in the simulation with the keyword [NUPCOL](#) or item 2 of keyword [GCONTOL](#).) Thus the group/field flow target is 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.

- 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 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 [GCONPRI](#) and [PRIORITY](#)). Also, the tolerance fraction currently does not apply to ECLIPSE 300.

- 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 [GINJGAS](#) and [WINJGAS](#)), or when the injection gas is made up of a mixture defined by [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.

- 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 [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.

- 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 [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:

```
GCONTOL  
0.02 3 /
```

To limit the number of injection calculation iterations in ECLIPSE 300 to 2:

```
GCONTOL  
3* 2 /
```

## GCUTBACK Group cutback limits

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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 [GCUTBACK](#) 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 item 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 [GCONPROD](#)), unless the group has been placed under prioritization with keyword [GCONPRI](#).

See also keyword [WLIMTOL](#).

## Example

```
GCUTBACK
G1 0.6 3* 0.9 LIQ /
G2 1* 3.0 2* 0.9 RESV /
/
```

## GCUTBACT Tracer cutback limits for groups

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 GCUTBACT 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*/
/
/

```

## GCVD

## Initial coal gas concentration versus depth tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (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  $\geq$  2

```
GCVD
 7000 74.0
10000 74.0 /
```

## GDCQ

## Initial DCQs for multiple contract groups

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

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  $\times$  Swing Factor; or DCQ  $\times$  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  $\times$  Profile Factor.

**FIX2** Same as for **FIX1**, but in addition a maximum gas production rate of DCQ  $\times$  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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 /
/
```

## GDFILE

## Imports a grid file

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## 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 /
```

## GDIMS

## Specify the dimensions for the instantaneous gradient option

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: Zero

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     /
/
```

## GECON

## Economic limit data for groups and the field

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC),  $\text{stb}/\text{stb}$  (FIELD),  $\text{scc}/\text{scc}$  (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC),  $\text{Mscf}/\text{stb}$  (FIELD),  $\text{scc}/\text{scc}$  (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC),  $\text{stb}/\text{Mscf}$  (FIELD),  $\text{scc}/\text{scc}$  (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

DEFAULT: No limit

(A value of 0.0 switches off this constraint)

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

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

```

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 /
/
/

```

### 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)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## 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 **WEFAC**. The treatment of well downtime is described in the **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' **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 '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.

---

**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 **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 **SUMMARY** section keyword **WEFFG**.

---

**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 /
/
```

## GEODIMS

## Geomechanics dimension data

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## GEOMECH Geomechanics calculation

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

INITH

NOINITH

DEFAULT: NOINITH

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

```
-- 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  
/
```

## GEOYCAP **Geomechanics cap model parameters**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 X0  
This must be a non-positive quantity.  
DEFAULT: 0.0

### Example

```

-- FIELD units
GEOYCAP
-- Co  PHI  UCS  K    R    W    D1
   1450 30  3600 1.0  3.2  0.13  9.79E-5  2*/

```

## GEOYLDF Geomechanics yield function parameters

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $C_{o_i}$   
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,  $C_o$ , with change in generalized plastic strain  $\bar{\epsilon}_p$ .

Generalized plastic strain is defined as  $\bar{\epsilon}_p = \sum_{i,j} \frac{\epsilon_{ij}^p \cdot \sigma_{ij}}{F}$

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  $Co = a\bar{\varepsilon}_p^b + Co_i$  while the hyperbolic law follows the relation

$$Co = \frac{\bar{\varepsilon}_p}{a + b\bar{\varepsilon}_p} + Co_i.$$

---

**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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

```
GETGLOB
```

## GETSOL

## Obtain initial solution arrays from restart file

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

## G

## Initial Gi values for enumeration

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{rm}^3$  (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 /
```

## GIALL

## Variation of saturated properties with pressure and Gi

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is an alternative method of specifying the variation of  $R_v$ ,  $R_s$ ,  $B_g$  and  $B_o$  with pressure and  $G_i$ . 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  $G_i$  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,  $G_i$ ) functions that multiply the saturated values of  $R_v$ ,  $R_s$ ,  $B_g$  and  $B_o$  at (P,  $G_i=0$ ) to give the true saturated value at (P,  $G_i$ ).

The first record of each elementary set of records contains the following items of data:

- The oil pressure value.  
The values should increase monotonically down the column.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB)
- The gas  $R_v$  multiplying factor at a  $G_i$  value specified as the first item in the [GINODE](#) table.
- The oil  $R_s$  multiplying factor at a  $G_i$  value specified as the first item in the [GINODE](#) table.
- The gas  $B_g$  multiplying factor at a  $G_i$  value specified as the first item in the [GINODE](#) table.
- The oil  $B_o$  multiplying factor at a  $G_i$  value specified as the first item in the [GINODE](#) table.

For record 2 and all subsequent records up to a maximum of NPPVT, there should be only 4 data item in the record. These items define the corresponding factors for the  $G_i$  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

```

--      RV      RS      BG      BO
GIALL
100.0   1.0     1.0     1.0     1.0
        0.97    0.93    0.98    0.93   /
200.0   1.0     1.0     1.0     1.0
        0.952  0.941  0.927  0.898  /
/

```

## GIMODEL

**Enable Gi pseudo-compositional option**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GCONTROL](#)).

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.

```
GINJGAS
'GI*' GV FIELD /
/
```

## GINODE

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Gi nodal values

The data comprises NTPVT (item 2 in keyword [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 [RSGI](#), [RVGI](#), [BGGI](#) and [BOGI](#) keywords.

See "[Gi Pseudo-compositional model](#)" in the *ECLIPSE Technical Description* for further information.

## Example

With NTPVT=2 and NPPVT  $\geq$  4

```
GINODE
  0.0
  10.0
  50.0
100.0 /
  0.0
100.0 /
```

## GLIFTLIM

## Maximum group capacity for artificial lift

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

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 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 /
/
```

## GLIFTOPT

## Group lift gas limits for gas lift optimization

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

- Group name or group name root, or FIELD 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.
- 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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)
- 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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

End the data record with a slash (/).

End the set of records with a blank record containing just a slash (/).

## Example

```

GLIFTOPT
-- group      max lift   max total
-- name       gas rate   gas rate
--           SM3/DAY
-- 'PL-*'     1E6 /
/

```

## GLTHEX

## Geomechanical linear thermal expansion coefficient

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 Lamé 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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)

DEFAULT: 0

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

```
-- set a normally rigid boundary in the x and y-directions
GMDISBC
-- side type displacement
  'X+' 1 /
  'X-' 1 /
  'Y+' 1 /
  'Y-' 1 /
/
```

## GMDISBCL Displacement boundary conditions for coupled rock and stress calculations in a Local Grid Refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GMPSTBCL](#) 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

```

-- set a normally rigid boundary in the x and y-
directions for the local grid refinement LGR1
GMDISBCL
-- side type velocity
  'X+' 1 'LGR1' /
  'X-' 1 'LGR1' /
  'Y+' 1 'LGR1' /
  'Y-' 1 'LGR1' /
/

```

## GMPSTBC

## Principal stress boundary conditions coupled stress and fluid flow

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GMTRABC](#) 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)

- 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
  9. The Y component of the third principal stress vector.  
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.  
  
GMPSTBC  
-- side   P1X   P1Y P1Z   P2X   P2Y   P2Z   P3X P3Y P3Z  
  'X-'   -2000  0  0     0   -2500  200   9* /  
  'X+'   -2000  0  0     0   -2500  200   9* /  
  'Y-'   -2000  0  0     0   -2500  200   9* /  
  'Y+'   -2000  0  0     0   -2500  200   9* /  
  'Z-'   -2000  0  0     0   -2500  200   9* /  
  'Z+'   -2000  0  0     0   -2500  200   9* /  
/  
/
```

## GMPSTBCL Principal stress boundary conditions for coupled stress and fluid flow in a Local Grid Refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.

GMPSTBCL
-- side      LGR      P1X      P1Y      P1Z      P2X      P2Y      P2Z      P3X P3Y P3Z
-- 'X-'      'LG1'    -2000    0      0      0      -2500    200     9* /
-- 'X+'      'LG1'    -2000    0      0      0      -2500    200     9* /
-- 'Y-'      'LG1'    -2000    0      0      0      -2500    200     9* /
-- 'Y+'      'LG1'    -2000    0      0      0      -2500    200     9* /
-- 'Z-'      'LG1'    -2000    0      0      0      -2500    200     9* /
-- 'Z+'      'LG1'    -2000    0      0      0      -2500    200     9* /
/
```

## GMTRABC Traction boundary conditions

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

*Finite Element*

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 /
/

```

## GMTRABCL Traction boundary conditions in a Local Grid Refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GMDISBCL](#) 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 /
/

```

## GNETDP

## Controls the dynamic changing of fixed pressures

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: sm<sup>3</sup>/day (METRIC), stb/day for oil, water or liquid (FIELD), Mscf/day for gas (FIELD),  
scc/hr (LAB), sm<sup>3</sup>/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: sm<sup>3</sup>/day (METRIC), stb/day for oil, water or liquid (FIELD), Mscf/day for gas (FIELD),  
scc/hr (LAB), sm<sup>3</sup>/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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

- 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.
- Phase of the injection network in which this group is a node  
  
WAT or GAS
- 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)
- 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  /
/
```

## GNETPUMP Automatic pump/compressor switching in a standard network

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `NETWORK` in the `RUNSPEC` section), you should define automatic compressors or pumps with keyword `NETCOMP` 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

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  for oil (FIELD),  $\text{Mscf}/\text{day}$  for gas (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: 0.0

A zero or negative value will disable this option.

3. Phase to which the production rate in item 2 refers

OIL or GAS

DEFAULT: OIL

4. New VFP table number

(When the pump/compressor is switched on this replaces the table number previously specified in keyword `GRUPNET`.)

A zero or negative number will leave the original VFP table number unchanged.

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 `GRUPNET`.)

The artificial lift quantity may be regarded as the pump or compressor power, according to the definition used in the VFP 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: 0.0

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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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 /
/

```

## GPMaint

## Data for pressure maintenance groups

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

---

ECLIPSE 100

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).

ECLIPSE 300

This **pressure maintenance** region should be identified by an integer between 1 and `NUMPMA` (set in item 14 of keyword `TABDIMS` in the `RUNSPEC` section). The default value for the pressure maintenance region is 1. Pressure maintenance regions are defined with the keyword `PMANUM` in the `REGIONS` section.

ECLIPSE 100 only

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), psia (FIELD), atma (LAB), atma (PVT-M)

ECLIPSE 300

---

**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:  $\text{rm}^3$  or  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{rb}$  or  $\text{stb}/\text{day}/\text{psi}$  (FIELD),  $\text{rcc}$  or  $\text{sc}/\text{hr}/\text{atm}$  (LAB),  $\text{rm}^3$  or  $\text{sm}^3/\text{day}/\text{atm}$  (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)

## Example

```

GPMAINT
-- Group      Inj      FIP      FIP      Targ  Prop  Inte
-- Name      Phase    Reg      Fam      Pres  Const Tcon
      PLAT-A    WINJ     2        1*      3900  30   70 /
      SP-C      WINJ     6        1*      3800  30   70 /
      SP-B      PROD     3        1*      3600  50   70 /
/

```

## 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

$$V = V_i + K \left[ e + \frac{\Sigma e \cdot \delta t}{T_i} \right] \quad \text{Eq. 3.76}$$

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  $P_{\text{targ}} - P_{\text{reg}}$  at the end of the previous timestep

$\Sigma(e \cdot \delta t)$  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 [GCONPROD](#) or [GCONINJE](#).

## GPMAINT3 Data for pressure maintenance groups

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WINJMIX](#) or [WINJORD](#) keywords. The name of the mixture must be specified in item 8. The control target specified in item 9 is adjusted.

- WV 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.
- GV 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:  $\text{rm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{rb}/\text{day}/\text{psi}$  (FIELD),  $\text{rcc}/\text{hr}/\text{atm}$  (LAB),  $\text{rm}^3/\text{day}/\text{atm}$  (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 WVG 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 /
/
```

## GPTABLE Tables of gas plant recovery factors

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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  $N_c$  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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 0.00 0.00031 0.05 0.10 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 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
/
```

## GPTABLE3 Gas plant table with oil, NGL and gas recoveries

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.

```

GPTABLE3
1 9 9
--Z9 CO2 N2 C1 C2 C3 C4-6 CZ7 CZ8 CZ9
0.0 0.00 0.00 0.00 0.00031 0.05 0.10 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
      0.95 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 0.00 0.00032 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
/

```

## GPTDIMS

## Gas plant table dimensions

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

- Maximum number of gas plant tables.  
This includes both tables specified with [GPTABLE](#), [GPTABLEN](#) and [GPTABLE3](#).  
DEFAULT: 1
- Maximum number of rows of gas plant table data.  
DEFAULT: 1
- 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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## GRADGRUP Specify field and group quantities to be output to HMD file

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

GOPR Surface group oil rate

GWPR Surface group water rate

GGPR Surface group gas rate

GWCT Group water cut

GGOR Group gas-oil ratio

GOPT Surface group oil production cumulative total

GWPT Surface group water production cumulative total

GGPT Surface group gas production cumulative total

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' /
/
```

## GRADRESV Specify derivatives of solution quantities to be output to the restart file

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

```
GRADRESV
 'PRES' /
 'SWAT' /
 'ACIP' /
 /
```

## Formats

A record 'GRADRESV' is written to the HMD and Restart files to list the mnemonics. For example, referring to the above example

```
'GRADRESV'          3 'CHAR'
'GPRES'  'GSWAT'  'GACIP'  /
/
```

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
  /
```

## GRADWELL Specify well quantities to be output to HMD file

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 100

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.

ECLIPSE 300

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' /
/
```

## GRAVCONS Specify user defined gravity constant

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>2</sup>bars/kg (METRIC), ft<sup>2</sup>psi/lb (FIELD), cm<sup>2</sup>atm/gm (LAB), m<sup>2</sup>atm/kg (PVT-M)

### Example

```
GRAVCONS
0.05/
```

## GRAVDR

## Use gravity drainage for dual porosity runs

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## GRAVD RB Use the vertical discretized gravity drainage model for dual porosity runs

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## GRAVDRM **Use the alternative gravity drainage model for dual porosity runs**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## GRAVITY

## Fluid gravities at surface conditions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

ECLIPSE 300

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.

ECLIPSE 100

---

**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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (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/
/
```

## GRES

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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<sup>3</sup> (METRIC), 62.3664 lb/ft<sup>3</sup> (FIELD), 0.999014 gm/cc (LAB), 999.014 kg/m<sup>3</sup> (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

```
GRES
0.805 0.778 0.454 0.548 0.610 0.659 0.719 0.838 0.907 /
```

## GREFS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

ECLIPSE 100 only

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 300

---

**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 100

---

**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](#).

---

## Example

```
GRIDOPTS  
YES 0 3 /
```

## GRIDUNIT **Specifies the grid data units**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 /
/
```

## GRUPGR

## Specifies group guide rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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)

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

```
GRUPGR
Z45 Y 100.0 INJ /
/
```

## GRUPINJE Specifies group injection targets

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

6. The target/limit water volume rate.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

7. The target/limit gas volume rate.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

8. The target/limit reservoir volume rate.

UNITS:  $\text{rm}^3/\text{day}$  (METRIC),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{rm}^3/\text{day}$  (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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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 **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 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 WV, G if item 2 is GV

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 WV or GV in item 2, and to the name of the group in item 1 for the other cases (WA, GA, ST or MIX 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.

```
GRUPINJE
G1 Water ' ' Water 1* 1000.0 /
/
```

### Example 3

The hydrocarbon defined by wellstream MixB is to be injected so as to replace the voidage produced by group PGC.

```
GRUPINJE
PLATA Stream MixB Reij 5* 1.0 Group PGC 2* R /
/
```

### 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.

```
GRUPINJE
--group fluid name ctrl rates reinj reinj reinj misc Reservoir/
-- mode frac type contr Surface
FIELD STREAM SOLV RE 5* 0.5 G 1* 2* S /
FIELD WAT 1* RE 5* 1.0 G 1* 2* R /
/
```

### 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.

```
GRUPINJE
-- Group Phase Name Mode Rates
FIELD GV Field GA 2* 1000 /
FIELD Water ' ' RV 3* 1000 /
/
WELLINJE
IG1 2* GR /
IG2 2* GR /
IW1 WA 1* GR /
IW2 WA 1* GR /
/
```

## GRUPLIM

# Specifies group economic limits

GRUPLIM is an alias for [GECON](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## GRUPMAST Identifies the master groups and their slaves

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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      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       /
/
```

## GRUPNET **Defines the standard production network structure**

x	ECLIPSE 100
x	ECLIPSE 300
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

- 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.
- 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)
- 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 Implying the group has no pipeline connection to its parent group.
- 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
- 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

NO 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.

ECLIPSE 100

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 (/).

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 /
/
```

## GRUPPROD Specifies group production target/limits

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`PVT-M`)

4. The target/limit water volume rate.

UNITS:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`PVT-M`)

5. The target/limit gas volume rate.

UNITS:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{Mscf}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`PVT-M`)

---

**Note:** If, in compositional mode, a group is controlled by gas volume rate, surface volumes are obtained by  $Q_g^m RT_{\text{ref}} / P_{\text{ref}}$ , 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: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
  7. The reservoir volume rate.  
UNITS: rm<sup>3</sup>/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm<sup>3</sup>/day (PVT-M)
  8. The target/limit wet gas rate. (Compositional Only)  
UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
  9. The target/limit linearly combined rate  
UNITS: determined by keyword `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 *ECLIPSE Technical Description* for further information.

## Example

```

GRUPPROD
GRPB RV 4* 1000 /
/

```

## GRUPRIG Assigns workover and drilling rigs to groups

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#)).

*ECLIPSE 100 only*

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.

ECLIPSE 100 only

**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.

```
GRUPRIG
-- group      workover  drilling  add or
-- name       rig       rig       remove
  FIELD        0         1 /
  FIELD        0         2 /
  PLAT-A       1         0 /
  PLAT-B       2         0 /
/
```

### Example 2

Drilling rig 2 is to be removed from the FIELD and just made available to group PLAT-B.

```
GRUPRIG
-- group      workover  drilling  add or
-- name       rig       rig       remove
  FIELD        0         2   REM /
  PLAT-B       0         2   ADD  /
/
```

## GRUPSALE Specifies group gas sales rate

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 /  
/
```

## GRUPLAV Identifies the slave groups in a slave reservoir

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 GRUPLAV 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 [GRUPLAV](#) 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 [RSCONST](#) or [RSCONSTT](#).
  - 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.

```

.....
GRUPLAV
-- slave  assoc  oil    wat/liq  gas    resv   oil    wat    gas
-- group  master  prod  prod    prod   prod   inj    inj    inj
-- name   group  limits limits limits limits limits limits limits limits
.....

```

PL-A	UP-A	/			
PL-B	UP-B		5*		BOTH /

## GRUPTARG Resets a group production rate target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [GCONPRI](#). These other quantities are left unchanged. The group control data should initially have been set using [GCONPROD](#) (for guide rate control) or [GCONPRI](#) (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 [GCONPRI](#).

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 [GCONPRI](#), 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

```
GRUPTARG
  PLAT-A  ORAT  15000 /
  PLAT-B  RESV  20000 /
/
```

## GRUPTREE Sets up tree structure for multi-level group control

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 100

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 WELSPECS.) Thus a group either contains wells (that is a well-group) or has other groups as children (that is a node-group).

ECLIPSE 300

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 WELSPECS 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:

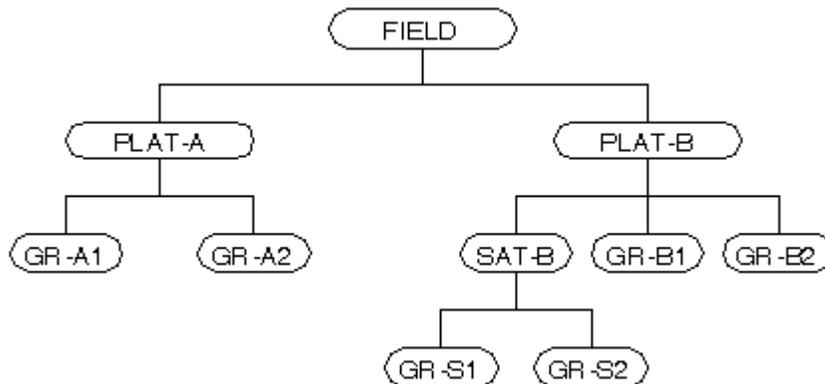


Figure 3.8. A five level hierarchy

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.

```
GRUPTREE
-- child  parent
GR-A1    PLAT-A  /
GR-A2    PLAT-A  /
GR-B1    PLAT-B  /
GR-B2    PLAT-B  /
SAT-B    PLAT-B  /
GR-S1    SAT-B   /
GR-S2    SAT-B   /
/
```

## GSATCOMP Specify a composition for a satellite group

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

### Example

```
GSATCOMP
SG2 IG STR1 /
/
```

## GSATINJE Injection rate data for satellite groups

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: sm<sup>3</sup>/day (METRIC), stb/day for oil, water or liquid (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm<sup>3</sup>/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: rm<sup>3</sup>/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm<sup>3</sup>/day (PVT-M)

DEFAULT: Zero

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<sup>3</sup> (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm<sup>3</sup> (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  /
/
```

## GSATPROD Production rate data for satellite groups

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: Zero

3. Water production rate from the group

The value must be greater than or equal to zero.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: Zero

4. Gas production rate from the group

The value must be greater than or equal to zero.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{rm}^3/\text{day}$  (METRIC),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{rm}^3/\text{day}$  (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.

ECLIPSE 100

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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: Zero

ECLIPSE 100 only

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:  $\text{kJ}/\text{sm}^3$  (METRIC),  $\text{Btu}/\text{Mscf}$  (FIELD),  $\text{J}/\text{scc}$  (LAB),  $\text{kJ}/\text{sm}^3$  (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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## GSEPREPT Assign separator conditions to a group for group flash summary vectors

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

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*

*Thermal only*

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

```
GSEPREPT
  PLAT-A      FIELDSEP      1      1 /
  PLAT-B      SEP1          2      1 /
/
```

## GSF

## Gas saturation functions (GASWAT/CO2STORE cases)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The gas saturation.  
Values should be between 0 and 1 and should increase monotonically down the column.
- 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.
- 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
/

```

## GSSCPTST Performs a sustainable capacity test

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

WGRA Wet gas rate control

DEFAULT: GRAT

3. Target production rate

UNITS:  $\text{sm}^3/\text{day}$  (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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

*ECLIPSE 300 only*

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: sm<sup>3</sup>/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm<sup>3</sup>/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, FGSRL, GGSTP, GGSPR, GGSRU and GGSRL all give information on the most recent sustainable capacity test (SCT) for gas. The [SUMMARY](#) section keywords FOSTP, FOSPR, FOSSP, FOSRU, FOSRL, GOSTP, GOSPR, GOSRU and GOSRL 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

ECLIPSE 300 only

- $R$  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.

---

## GSWINGF

## Swing and profile factors for multiple contract groups

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#).

## ECLIPSE 100

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).

## ECLIPSE 100

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 {profile factor × 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 × 365.25, which is the ACQ. A warning is printed if the profile factors are not normalized.

## Example

```

GSWINGF
-- GROUP  JAN  FEB  MAR  APR  MAY  JUN  JUL  AUG  SEP  OCT  NOV  DEC
  PLAT-A   2*1.5036  3*1.0024          3*0.5012          3*1.0024          1.5036
           2*1.5036  3*1.0024          3*0.5012          3*1.0024          1.5036 /
  PLAT-B   2*1.6551  3*1.0031          3*0.3511          3*1.0031          1.6551
           2*1.6551  3*1.0031          3*0.3511          3*1.0031          1.6551 /
/
    
```

## GTADD

## Adds to a group rate target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

WRAT [ 2 ] Water 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

WINR [ 11 ] Water reservoir volume 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/
/
    
```

## GTMULT

## Multiplies a group rate target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [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

WRAT [ 2 ] Water 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

WINR [ 11 ] Water reservoir volume 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. 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

```
GTMULT
PLAT-A  ORAT  1.5 /
PLAT-B  RESV  0.8 /
/
```

## GUIDECAL Scales guide rates according to calorific value

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $C_{gW}$

$$\beta_W = (1 + A C_{gW})^B \quad \text{Eq. 3.77}$$

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$GR_p = \frac{(POT_p)^A}{B + C(R_1)^D + E(R_2)^F} \quad \text{Eq. 3.78}$$

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  $GR_p$  is the oil phase guide rate.

The phase potential ratios are:

$R_1$  = water-oil ratio (dimensionless)

$R_2$  = gas-oil ratio

UNITS:  $sm^3/sm^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $sm^3/sm^3$  (PVT-M)

LIQ  $GR_p$  is the liquid phase guide rate (oil + water).

The phase potential ratios are:

$R_1$  = water cut (dimensionless)

$R_2$  = gas-liquid ratio

UNITS:  $sm^3/sm^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $sm^3/sm^3$  (PVT-M)

GAS  $GR_p$  is the gas phase guide rate.

The phase potential ratios are:

$R_1$  = water-gas ratio

$R_2$  = oil-gas ratio

UNITS:  $sm^3/sm^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $sm^3/sm^3$  (PVT-M)

RES  $GR_p$  is the reservoir fluid volume guide rate.

The phase potential ratios are:

$R_1$  = water-oil ratio (dimensionless)

$R_2$  = gas-oil ratio

UNITS:  $sm^3/sm^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $sm^3/sm^3$  (PVT-M)

COMB  $GR_p$  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:  $\text{sm}^3/\text{U}$  (METRIC),  $\text{stb}/\text{U}$  (FIELD),  $\text{scc}/\text{U}$  (LAB),  $\text{sm}^3/\text{U}$  (PVT-M) where U represents the units specified in LCUNIT.

$R_2$  = gas to linearly combined phase ratio

UNITS:  $\text{sm}^3/\text{U}$  (METRIC),  $\text{Mscf}/\text{U}$  (FIELD),  $\text{scc}/\text{U}$  (LAB),  $\text{sm}^3/\text{U}$  (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

$$GR_{used} = GRDAMP \cdot GR_{calculated} + (1-GRDAMP)GR_{previous} \quad \text{Eq. 3.79}$$

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<sub>2</sub>.

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<sub>min</sub>. If the calculated guide rate GR<sub>p</sub> is less than the minimum GR<sub>min</sub>, 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<sub>min</sub> > 10<sup>-19</sup> × flow

DEFAULT: 10<sup>-6</sup>

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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`.

```

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 /
/

```

## **H**

This section contains an alphabetic listing of keywords starting with the letter H.

## H2SSOL

## Allow H<sub>2</sub>S to dissolve in the aqueous phase

This keyword turns on the H<sub>2</sub>S solution option. The H<sub>2</sub>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<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## HALFTRAN Specifies half block transmissibilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## HAxxxxxx

# 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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## HDISP

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

```

.....
HDISPWT1
100 3 /
35 1/
HDISPWT2
80 0/
35 0/
.....

```

## HEATCR

## Rock heat capacity

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/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.

```
HEATCR
--
--In Field units
60*35 40*30 /
```

## HEATCRT

## Temperature dependence of the rock heat capacity

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/K<sup>2</sup> (METRIC), Btu/ft<sup>3</sup>/°R<sup>2</sup> (FIELD), J/cc/K<sup>2</sup> (LAB), kJ/m<sup>3</sup>/K<sup>2</sup> (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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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,  $H_{max}$

UNITS: kJ/day (METRIC), Btu/day (FIELD), J/hr (LAB), kJ/day (PVT-M)

DEFAULT: 0

6. The maximum temperature of the connection cell,  $T_{max}$

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)

7. 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 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:

$$H = H_{max}$$

where  $H_{max}$  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:

$$H = \min\left(H_{max}, V \cdot \frac{\Delta E}{\Delta t}\right)$$

where  $V$  is the grid-block volume,  $\Delta t$  is the timestep, and the estimated change in energy density

$$\Delta E \text{ is given by: } \Delta E = e_t(T_{max}) - e_{t-\Delta t}$$

where  $e_t(T_{max})$  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  $T_c$ .

$$H = \min[(T_{max} - T_c) \times R, H_{max}]$$

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  $H = \min[0, H_{max}]$ . 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

```
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* /
/
```

## HEATERL LGR heater data

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$\Delta E = e_t(T_{max}) - e_{t-\Delta t}$$

where  $e_t(T_{max})$  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 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  $T_{max}$  and the current grid-block temperature  $T_c$ .

$$H = \min[(T_{max} - T_c) \times R, H_{max}]$$

where  $H_{max}$ ,  $T_{max}$  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  $H = \min[0, H_{max}]$ . If  $H_{max}$  (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* /
/
```

## HEATTR

## Heat transmissibilities for radial run

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_r N_\theta N_z$  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 /
```

## HEATTTHT

**Heat transmissibilities for radial run**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword can only be used with the [THERMAL](#) option.

HEATTTHT 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_x N_y N_z$  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)

## Example

```
HEATTX
--
--In Field units
60*35 40*30 /
```

## HEATTY

**Heat transmissibility y-direction**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

# HEATVAP Heat of vaporization

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$\Delta H = A \cdot (1 - T / T_{crit})^B \quad \text{Eq. 3.80}$$

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

$$A = \frac{\Delta H_{nb}}{(1 - T_{nb} / T_{crit})^B} \quad \text{Eq. 3.81}$$

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/
```

## HEATVAPE Heat of vaporization exponent

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 9<sup>th</sup> 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

$$\Delta H = A \cdot (1 - T / T_{crit})^B \quad \text{Eq. 3.82}$$

where

*A* Constant part of  $\Delta H$  entered with keyword [HEATVAP](#)

*B* Exponent entered with this keyword (normally default)

*T* Temperature

*T<sub>crit</sub>* Critical temperature of the component, keyword [TCRIT](#).

### Example

For three components:

```
HEATVAPE
1.0 1.0 1* /
```

## HEATVAPS **Heat of vaporization at standard temperature**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/
```

## HMAQUCT

## Specify gradients for Carter-Tracy aquifers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [AQUANCONL](#) 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 $\geq$ 2 specified using [HMDIMS](#)

```

.....
AQUCT
--Aq. Datum Initial Perm Poro Ct Ro Thick- Angle Water AQU TAB 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 /
HMAQUCT
-- Aquifer Derivative w.r.t.      Derivative w.r.t.
-- number permeability multiplier opening angle multiplier
   1          YES                  YES /
/

```

## HMAQUFET Specify gradients for Fetkovich aquifers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [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](#).

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
  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 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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [AQUNUM](#).
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  $PORV = Length \times Area \times Poro$  using data entered in the [AQUNUM](#) keyword, where the porosity *Poro* is taken from the grid block value if defaulted. The transmissibilities of numerical aquifer connections are calculated from the permeability entered in the [AQUNUM](#) keyword. The regional parameter [HMPORVM](#) may include aquifer cells to obtain sensitivities, but any values of [HMMULTPV](#) 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 `HMMULRGT` 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`.

```

AQUNUM
--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 /
/
HMAQUNUM
-- Aquifer PoreV parameter Perm. parameter Trans. parameter
  1 YES YES NO /
/

```

## HMDIMS

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 HMxxxxxxx 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.

- ECLIPSE 100 only*
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

- ECLIPSE 100 only*
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

- ECLIPSE 100 only*
6. The maximum number of well parameters specified using keyword [HMWPIMLT](#) in the SCHEDULE section.  
DEFAULT: 0

7. Not used at present

- ECLIPSE 100 only*
8. The maximum number of parameters declared for gradients with respect to rock parameters (keywords [HMROCK](#) and [HMROCKT](#) in the PROPS section).  
DEFAULT: 0

- ECLIPSE 100 only*
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

*ECLIPSE 100 only*

---

**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  NRIFTB  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
  4        5       8 /

```

## HMFAULTS Specify gradients for faults

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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' /
/
```

## HMMLAQUN Multipliers for numerical aquifer properties

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword applies multipliers to properties determining the performance of a numerical aquifer defined in keyword [AQUNUM](#). It is intended to be used as a modifier on the base data specified in the [AQUNUM](#) 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 [AQUNUM](#) 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 [HMAQUNUM](#) keyword.

---

**Note:** The aquifer pore volume cannot be modified by editing [PORV](#) for the aquifer cells or by specifying [MULTPV](#). See the [AQUNUM](#) keyword.

---

### Example

```

AQUNUM
--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 /
/

```

## HMMLCTAQ Specify modifiers for Carter-Tracy aquifers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This keyword applies multipliers to properties determining the performance of the Carter-Tracy aquifer defined in keyword [AQUCT](#). It is intended to be used as a modifier on the base data specified in the [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 [AQUCT](#) item 1.
2. Multiplier on aquifer permeability.  
DEFAULT: 1.0
3. Multiplier on aquifer angle of influence.  
DEFAULT: 1.0
4. Multiplier on aquifer depth  
DEFAULT: 1.0

See also the [HMAQUCT](#) keyword.

### Example

```

AQUCT
--Aq. Datum Initial Perm Poro Ct Ro Thick- Angle Water AQU TAB Salt
--no. depth pressure
  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 /
/

```

## HMMLFTAQ Specify modifiers for Fetkovich aquifers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

```

AQUFET
-- Datum      Initial   Wi        Ct         J         Table      I1 I2  J1 J2  K1 K2  Face
-- depth      pressure
   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          /
/
    
```

## HMMLTWCN Modifies well connection transmissibility factor and skin factor

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

```
HMMLTWCN
-- 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*  /
/
```

## HMMLT<sub>xxx</sub>

# Cumulative multipliers for gradient parameters

See keyword [HMMULT<sub>xxx</sub>](#).

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## HMMMREGT Cumulative inter-region transmissibility modifiers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
HMMMREGT
 1 2 1.1 /
 3 4 1.5 /
 /
```

## HMMROCK Specify cumulative multipliers on rock compressibility

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$PV = PV_{ref}(1 + X + X^2)$$

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$PV = PV_{ref} M(p)$$

where  $p$  is the reservoir pressure. If the `OVERBURD` keyword has been entered,

$$P = P_{eff} = P_{fluid} - P_{overburden}$$

If the `STRESS` option is selected in item 1 of the `ROCKOPTS` keyword,

$$P = P_{eff} = P_{overburden} - P_{fluid}$$

ECLIPSE calculates gradients with respect to two parameters using the functional form

$$PV = PV_{ref} M(p) \left( \lambda_1 + \lambda_2 \frac{(P - P_{ref})}{P_{dim}} \right)$$

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 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 /
/
```

## HMMULRGT Calculate gradients for inter-region transmissibility multipliers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- From region number (I)
 

A non-positive or defaulted value acts as a wildcard on all regions.

DEFAULT: Negative
- To region number (J)
 

A non-positive or defaulted value acts as a wildcard on all regions.

DEFAULT: Negative
- Direction(s) in which to apply the multiplier:
 

Either X, Y, Z, XY, YZ, XZ or XYZ.

DEFAULT: XYZ
- 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
HMMULTXY	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:

Cartesian	Radial alias
MULTX / HMMULTX	MULTR / HMMULTR
MULTY / HMMULTY	MULTTHT / HMMULTTH
MULTZ / HMMULTZ	MULTZ / HMMULTZ
MULTPV / HMMULTPV	MULTPV / HMMULTPV

Table 3.15: Correspondence between MULTxx and HMMULTxx keywords

Additionally, the keyword HMMLTPXY (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 HMTRANXn 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 HMTRANXn 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/HMMULTYHMMULTZ 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/HMMULTYHMMULTZ 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 HMMULTxx 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 /  
/
```

## HMMULTFT **Modifies the transmissibility across the named fault**

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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      /
/

```

## HMMULTSG Dual porosity sigma factor modifiers

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [HMxxxxxxx](#) keywords pertaining to end-points. The modifier keywords [BOX](#), [EQUALS](#), [COPY](#), [MINVALUE](#), [MAXVALUE](#) and [ADD](#) may be used with the [HMxxxxxxx](#) 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 [REFINE](#) / [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 [HMxxxxxxx](#) 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.

[HASWLP](#) Specifies modifier on lower water saturation used for water capillary pressure.

[HASGLP](#) 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
  `HMKRWR' 1.2 1 10 1 10 4 5 /
  `HMKRW' 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

```

## HMROCK

## Calculate gradients for rock compressibility

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `HMROCKT` 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

$$PV = PV_{ref}(1 + X + X^2) \quad \text{Eq. 3.83}$$

with

$$X = C(p - p_{ref})$$

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

$$X = \lambda C(p - p_{ref}) \quad \text{Eq. 3.84}$$

## Example

```
HMROCK
1 1 /
2 1 /
/
```

## HMROCKT

## Calculate gradients for rock compaction tables

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$PV = PV_{ref}M(p) \quad \text{Eq. 3.85}$$

where  $p$  is the reservoir pressure. If the [OVERBURD](#) keyword has been entered,

$$p = P_{eff} = P_{fluid} - P_{overburden} \quad \text{Eq. 3.86}$$

If the [STRESS](#) option is selected in item 1 of the [ROCKOPTS](#) keyword,

$$p = P_{eff} = P_{overburden} - P_{fluid} \quad \text{Eq. 3.87}$$

ECLIPSE calculates gradients with respect to two parameters using the functional form

$$PV = PV_{ref}M\left(p\right)\left(\lambda_1 + \lambda_2 \frac{(p - P_{ref})}{P_{dim}}\right) \quad \text{Eq. 3.88}$$

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 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  $p_{ref}$  and  $p_{dim}$  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.  $p_{ref}$   
 UNITS: barsa (METRIC), psia (FIELD), atma (LAB)  
 DEFAULT: Error if not specified
2.  $p_{dim}$   
 UNITS: barsa (METRIC), psia (FIELD), atma (LAB)  
 DEFAULT: Error if not specified

Items  $p_{ref}$  and  $p_{dim}$  are described in the [HMMROCKT](#) keyword.

### Example

```

--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 /

```

## HMWELCON Specify gradients for well connection parameters

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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*' /
/
```

## HMxxxxxx

## Gradient regions

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, HMKRGR, HMKRORW, HMKRORG. 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 HMPERM 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 HMxxxxxxx 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 HMxxxxxxx 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
'HMTRANX' 1 1 24 1 8 2 2 / layer 3
'HMTRANZ' 1 1 24 1 8 2 2 / layer 2
'HMTRANX' 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/
```

## HSORW

## Hysteresis residual oil saturation in oil-water system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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/
```

## HSWRO

## Hysteresis residual water saturation in water-oil system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
HSWRO
60*0.1/
```

## HWELLS

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 Kx replacing Kz 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

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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
HWKRORG
 6*0.8
 6*0.8
 6*0.7 /

```

## HWKRORW Scaled (high salinity) water-wet end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## HWKRWR

## Scaled (high salinity) water-wet end point water relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## HWPCW

## Scaled (high salinity) water-wet maximum oil-water capillary pressure

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## HWSNUM

**(High salinity) water-wet saturation function region numbers**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ([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 [SURFNUM](#) 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 NTSFUN=2 specified in keyword [TABDIMS](#) and NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword [DIMENS](#):

```
HWSNUM
144*1 96*2 /
```

## HWSOGCR Scaled (high-salinity) water-wet critical oil-in-gas saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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 /

```

## HWSWCR

## Scaled (high salinity) water-wet critical water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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

```

## HWSWL

## Scaled (high salinity) water-wet connate water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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 /

```

## HWSWLPC

**Scaled (high salinity) water-wet connate water saturations, for Pc curves only**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The 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 HWSWLPC keyword and the maximum water saturation defined using the [HWSWU](#) keyword.

The HWSWLPC keyword is not used for scaling the oil relative permeability curves.

The 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 RUNSPEC keywords [ENDSCALE](#) and [SURFACT](#) (or [SURFACTW](#)) respectively.

The 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\*0.21). Note that spaces must not be inserted on either side of the asterisk.

If the keyword 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 [HWSWL](#) keyword.

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
HWSWLPC
7*0.22  7*0.20  7*0.18  /

```

## HWSWU

## Scaled high salinity (water-wet) saturation table maximum water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [HWSWLP](#) 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

```

----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 /

```

## HXFIN

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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* /
```

## HYDRHEAD Sets reference data for output of hydraulic heads

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises reference data that is used if either HYDH or HYDHFW 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 HYDHFW requested or another freshwater head.  
UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (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

$$HEAD = z_{ref} - z + (p_w - p_{atm}) / (\rho g)$$

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 /

```

## HYDRO

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Define hydrocarbon type

In a run with  $N_c$  components, this keyword defines the component type. This keyword can be used with the [SOLID](#) option for predicting asphaltene precipitation (see "[Solid phase](#)" in the *ECLIPSE Technical Description*).

The keyword should be followed by  $N_c$  values. Each value should be one of the following:

N Non-hydrocarbon

H Hydrocarbon

P Paraffin

C Cyclic hydrocarbon (naphthene)

A Aromatic

DEFAULT: 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 *ECLIPSE Technical Description*.

This keyword is used by the multiphase flash. For asphaltene precipitation, the heaviest hydrocarbon should be an aromatic.

### Example

For a four-component system with one EoS region:

```

.....
HYDRO
H H P A /
.....

```

## HYFIN

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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* /

```

## HYKR

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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](#).

## HYMOBGDR **Change the way secondary drainage curves are computed in hysteresis runs with soluble gas**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## HYPC

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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](#).

## HYST

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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](#), [HYPC](#) and [NOHYPC](#) 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 HYPC or NOHYPC 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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 EHISTR 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.

## HYSTK

### 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## HZFIN

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Local grid size ratios in z-direction

The keyword HZFIN 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](#).

## I

This section contains an alphabetic listing of keywords starting with the letter I.

## IHOST

## Groups LGRs together in the same process

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
/
```

## IKRG-ORW

## Imbibition end point relative permeabilities

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The keyword IKRG, IKRGR, IKRW, IKRWR, IKRO, IKRORG and IKRORW 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.

Imbibition Keyword	Drainage Keyword	End-point scaled
IKRG	<a href="#">KRG</a>	Maximum gas relative permeability.
IKRGR	<a href="#">KRGR</a>	Gas relative permeability at residual oil (or water in gas-water systems).
IKRW	<a href="#">KRW</a>	Maximum water relative permeability.
IKRWR	<a href="#">KRWR</a>	Water relative permeability at residual oil (or gas in gas-water systems).
IKRO	<a href="#">KRO</a>	Maximum oil relative permeability.
IKRORG	<a href="#">KRORG</a>	Oil relative permeability at the critical gas saturation.
IKRORW	<a href="#">KRORW</a>	Oil relative permeability at the critical water saturation.

Table 3.16: Imbibition relative permeability keywords

## IKU3P

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [SWF3](#) keyword (rather than [SWFN](#)). For the gas phase, both gas-oil and gas-water tables must be entered using the [SGF3](#) keyword (rather than [SGFN](#)).

The IKU method can be used with endpoint scaling. In this case the usual critical water and gas saturation values ([SWCR](#), [SGCR](#)) are interpreted as critical water and gas in oil. Critical water in gas values are entered with [SWGCR](#); critical gas in water values are entered with [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 [SOF3](#), [SWF3](#) and [SGF3](#) with [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 *ECLIPSE Technical Description*.

## IMBNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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**, **SO2**, **SO3**, **SO32D** 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 **IMBNUMX** 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 `IMBNUMMF`, `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 /
```

## IMBNUMMF Matrix-fracture imbibition region numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

---- RUNSPEC section
OIL
WATER
GAS
SATOPTS
  HYSTER /
ENDSCALE
--   NTENDP   NSENDP
   2*   1       3 /
---- PROPS section
IMKRVD
--
--           water   gas   oil   water   gas   oil   oil
--   Depth   Krw    Krg   Kro   at crit  at crit  at crit  at crit
--                               oil to  oil to   gas     water
--                               water   gas
--
--   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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

---- 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  /

```

## IMPES

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

## IMPFIL

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

```

-- Import from OLDCase.INIT
IMPFIL
  'OLDCase.INIT' /
  PORO PERMX PERMY PERMZ /

```

## IMPLICIT

### 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
```

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## IMPORT

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## Imports grid file data

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 /
```

## IMPSAT

### 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## IMPTVD

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [ENDNUM](#) in the [REGIONS](#) section, and keywords [ENPTVD](#) and [IMKRVD](#) in the [PROPS](#) section.

## Example

```

---- 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*/

```

## IMSPCVD

## Imbibition saturations for $P_c$ curves versus depth tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `ENDNUM` in the `REGIONS` section, and the keywords `ENSPCVD`, `SWLPC` and `SGLPC` in the `PROPS` section.

### Example

```

---- RUNSPEC section
OIL
WATER
GAS
ENDSCALE
-- NTENDP  NSENDP
 2*  1      2  /
---- PROPS section
IMSPCVD
3000.0  1*  0.21
9000.0  1*  0.30  /

```

## INCLUDE

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## 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' /
```

## INIT

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [FILLEPS](#), and RUNSPEC section keyword [INSPEC](#).

The INIT keyword has no associated data.

## INRAD

## Inner radius for radial geometry or radial local grid refinements

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## INSPEC

### 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## INTPC

## Invokes the Integrated Pc curve option for dual porosity

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (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 **Rock ion exchange constant**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$\frac{(C_{++})(C_+^r)^2}{(C_{++}^r)(C_+)^2} = \beta^r q^r \quad \text{Eq. 3.89}$$

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<sup>3</sup>)<sup>-1</sup> (METRIC), (lb-M/rb)<sup>-1</sup> (FIELD), (gm-M/rcc)<sup>-1</sup> (LAB)

DEFAULT: 0.0 (exchange equilibrium ignored)

### Example

For NTSFUN=2:

```

IONXROCK
-- beta
 0.25 /
 0.35 /

```

## IONXSURF Surfactant ion exchange constant

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$\frac{(C_{++})(C_+^s)^2}{(C_+^s)(C_+)^2} = \beta^s C^s \quad \text{Eq. 3.90}$$

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<sup>3</sup>)<sup>-1</sup> (METRIC), (lb-M/rb)<sup>-1</sup> (FIELD), (gm-M/rcc)<sup>-1</sup> (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

```

500	0.25	/
435	0.35	/

## IPCW,IPCG

## Imbibition table scaled capillary pressure end-points

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Imbibition keyword	Drainage keyword	End-point scaled
IPCW	<a href="#">PCW</a>	Maximum water capillary pressures
IPCG	<a href="#">PCG</a>	Maximum gas capillary pressures

Table 3.17: Imbibition capillary pressure keywords

## ISGAS

## Run is gas condensate

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

## ISTNUM

## Secondary hysteresis saturation function region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 PSTNUM 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 secondary 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 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 PSTNUM 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 tertiary hysteresis saturation region numbers PSTNUM and DSTNUM in the REGIONS section.

### Example

With NTSFUN=2 specified in keyword TABDIMS; NDIVIX=8, NDIVIY=6 and NDIVIZ=6 specified in RUNSPEC keyword DIMENS; and no input BOX set:

```
ISTNUM
144*1 96*2 /
```

## ISGL-OWCR Imbibition table end-points

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Imbibition keyword	Drainage keyword	End-point scaled
ISGL	<a href="#">SGL</a>	Connate gas saturation.
ISGLPC	<a href="#">SGLPC</a>	Connate gas, $P_c$ curves only. ECLIPSE 100 only
ISGCR	<a href="#">SGCR</a>	Critical gas saturation.
ISGU	<a href="#">SGU</a>	Maximum gas saturation.
ISWL	<a href="#">SWL</a>	Connate water saturation.
ISWLPC	<a href="#">SWLPC</a>	Connate water, $P_c$ curves only. ECLIPSE 100 only
ISWCR	<a href="#">SWCR</a>	Critical water saturation.
ISWU	<a href="#">SWU</a>	Maximum water saturation.
ISOGCR	<a href="#">SOGCR</a>	Critical oil-to-gas saturation
ISOWCR	<a href="#">SOWCR</a>	Critical oil-to-water saturation.

Table 3.18: Imbibition saturation table end-point keywords

## ISOLNUM

## Identifies extent of each independent reservoir region

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## **J**

This section contains an alphabetic listing of keywords starting with the letter J.

# JALS

## Linear solver for thermal problems

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$P_c = J(S) \cdot ST \cdot \left( \frac{\text{Porosity}}{\text{Permeability}} \right)^{1/2} \cdot U_{\text{const}} \quad \text{Eq. 3.91}$$

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

Porosity cell porosity

Permeability cell permeability

$U_{\text{const}}$  constant depending on 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 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:

$$\text{Scaling Factor} = ST \cdot \left( \frac{\text{Porosity}}{\text{Permeability}} \right)^{1/2} \cdot U_{\text{const}} \quad \text{Eq. 3.92}$$

---

**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:

$$P_c = J(S) \cdot ST \cdot \left( \frac{\text{Porosity}^\alpha}{\text{Permeability}^\beta} \right) \cdot U_{\text{const}}$$

ECLIPSE 100

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  $P_c$  is ignored. In this case:

$$P_c = J(S) \cdot ST \cdot \frac{ST(P)}{ST_{ref}} \cdot \left(\frac{Porosity}{Perm}\right)^{1/2} \cdot U_{const} \quad \text{Eq. 3.93}$$

where

ST(P) and  $ST_{ref}$  are input using the STOG or STOW keywords.

ECLIPSE 100

---

**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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

Y PERMY value

Z PERMZ value

DEFAULT: XY

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:

$$P_c = J(S) \cdot S_f \quad \text{Eq. 3.94}$$

where the scaling factor  $S_f$  is defined by the expression:

$$S_f = ST \cdot \left( \frac{\text{Porosity}^\alpha}{\text{Perm}^\beta} \right) \cdot U_{\text{const}} \quad \text{Eq. 3.95}$$

where:

$J(S)$  J-function values from the input saturation tables

ST Surface tension - data item 2 or 3

Porosity 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 JFUNC tables associated with the SATNUM array. The JFUNC 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 JFUNC tables associated with the IMBNUM array. The JFUNC tables associated with the directional imbibition arrays IMBNUMX, IMBNUMX-, IMBNUMY, IMBNUMY-, IMBNUMZ and IMBNUMZ- are ignored.

Using the JFUNC 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 JFUNC keyword.

The input values specified in the JFUNC keyword may be output by including the option JFUNC 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 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:

$$P_c = J(S) \cdot ST \cdot \frac{ST(P)}{ST_{ref}} \cdot \left( \frac{\text{Porosity}^\alpha}{\text{Perm}^\beta} \right) \cdot U_{const} \quad \text{Eq. 3.96}$$

where

ST(P) and ST<sub>ref</sub> are input using the STOG or STOW keywords.

---

**Note:**

ECLIPSE 100

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.

---

## Example

With `NTSFUN=5`:

```
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 /
/
NONE /
-- Table 4 copied from table 3
-- Capillary pressures taken directly from
-- region No 5 saturation function tables
```

## K

This section contains an alphabetic listing of keywords starting with the letter K.

## KRGDI

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [RPTRST](#), [RPTSOL](#), [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 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  $N = 1 + \text{number of hydrocarbon components} + \text{number of water components}$ :

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 <sup>3</sup> ) <sup>-1</sup> (METRIC), (lb-M/rb) <sup>-1</sup> (FIELD), (gm-M/rcc) <sup>-1</sup> (LAB)
Columns COMPS + 2 to COMPS + COMPW + 1	Damage factors for water components

UNITS: (kg-M/rm<sup>3</sup>)<sup>-1</sup> (METRIC), (lb-M/rb)<sup>-1</sup> (FIELD), (gm-M/rcc)<sup>-1</sup> (LAB)

---

**Note:** The damage factors must have values of zero or greater.

---

---

**Note:** The KRGD 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.

```
.....  
KRGD  
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  
/  
.....
```

## KRG-IKRGR Scaled end-point gas relative permeabilities

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [ENDSCALE](#) in the 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:

$$K_{rg} = K_{rg}(\text{Table}) \frac{\text{KRG}(\text{GridBlock})}{K_{rg\max}(\text{Table})} \quad \text{Eq. 3.97}$$

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 [SCALECRS](#), SR is given by

SR = 1 - SOGCR - SWL in gas/oil or gas/oil/water runs

SR = 1 - SWCR in gas/water runs

where SOGCR, SWL and SWCR are the scaled values. When the alternative scaling method is not employed, only the two nodes SGCR and SGU are used to scale the saturation endpoints, and SR is taken to be the transformed value of

$$\text{SR}(\text{Table}) = 1.0 - S_{ogcr}(\text{Table}) - S_{wco}(\text{Table})$$

or

$$\text{SR}(\text{Table}) = 1.0 - S_{wcr}(\text{Table}).$$

Hence the two cases are:

$$\text{SG} < \text{SR} \quad K_{rg} = K_{rg}(\text{Table}) \frac{\text{KRGR}(\text{GridBlock})}{(K_{rg}(\text{SR}))(\text{Table})} \quad \text{Eq. 3.98}$$

$$\text{SG} > \text{SR} \quad K_{rg} = \text{KRGR} + \frac{\text{KRG} - \text{KRGR}}{K_{rg\max} - K_{rg}(\text{SR})} (K_{rg}(\text{Table}) - K_{rg}(\text{SR})) \quad \text{Eq. 3.99}$$

Fourteen keywords are provided to allow the maximum gas relative permeability to be set for each grid cell independently. These keywords are:

KRG      Max gas  $K_r$  for the non-directional saturation function tables

KRGX     Max gas  $K_r$  for the +X face of each grid cell

KRGX-    Max gas  $K_r$  for the -X face of each grid cell

KRGY     Max gas  $K_r$  for the +Y face of each grid cell

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 KRGZ, KRGZ- 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, KRGZ 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.

ECLIPSE 300

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 /
```

## KRUNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Directional relative permeability table numbers

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: [SGFN](#), [SOF2](#), [SOF3](#), [SOF32D](#) and [SWFN](#), or: [SGOF](#), [SLGOF](#) and [SWOF](#) in the 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).

KRUNUMX Saturation table numbers for the +X face of each grid block

KRUNUMX- Saturation table numbers for the -X face of each grid block

KRUNUMY Saturation table numbers for the +Y face of each grid block

KRUNUMY- Saturation table numbers for the -Y face of each grid block

KRUNUMZ Saturation table numbers for the +Z face of each grid block

KRUNUMZ- 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:

KRUNUMR Saturation table numbers for the +R face of each grid block

KRUNUMR- Saturation table numbers for the -R face of each grid block

KRUNUMT Saturation table numbers for the +T face of each grid block

KRUNUMT- Saturation table numbers for the -T face of each grid block.

The +T direction is that of increasing azimuthal angle. The KRUNUMZ and KRUNUMZ- 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 [SATOPTS](#). KRUNUMX-, KRUNUMY- and KRUNUMZ- should be used only if irreversible directional relative permeabilities have been enabled by the keyword [SATOPTS](#). If reversible relative permeability tables are to be employed, the saturation tables specified using KRUNUMX are used for both the X+ and X- faces of each grid block. Similarly KRUNUMY is used for the Y+ and Y- faces, and KRUNUMZ 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 KRUNUMX.

If a table number is defaulted for any grid cell, the table number is filled in using the data entered under the [SATNUM](#) keyword.

---

**Note:** The potential difference between two cells uses the capillary pressures in the tables addressed by each of the KRUNUM keywords. If this is not required, then the capillary pressure data in these tables should be set equal to that in the 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

```

--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

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
/

```

## KRUNUMMF

## Matrix-fracture flow saturation tables numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  /
KRUNUMMF
 48*3
 48*1
/

```

## KRODI

## Oil relative permeability damage initiation

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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 /
```

## KRODM

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Oil relative permeability damage multipliers

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  $N = 1 + \text{number of hydrocarbon components} + \text{number of water components}$ :

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/rm <sup>3</sup> ) <sup>-1</sup> (METRIC), (lb-M/rb) <sup>-1</sup> (FIELD), (gm-M/rcc) <sup>-1</sup> (LAB)
Columns COMPS + 2 to COMPS + COMPW + 1	Damage factors for water components

UNITS: (kg-M/rm<sup>3</sup>)<sup>-1</sup> (METRIC), (lb-M/rb)<sup>-1</sup> (FIELD), (gm-M/rcc)<sup>-1</sup> (LAB)

---

**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 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
/
```

## KRO-ORG

## Scaled end point oil relative permeabilities

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$K_{ro} = K_{ro}(\text{Table}) \frac{KRO(\text{GridBlock})}{K_{romax}(\text{Table})} \quad \text{Eq. 3.100}$$

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 - SGL 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  $SR(\text{Table}) = 1.0 - S_{wcr}(\text{Table}) - S_{gco}(\text{Table})$  for the KRORW keyword, or  $SR(\text{Table}) = 1.0 - S_{gcr}(\text{Table}) - S_{wco}(\text{Table})$  for the KRORG keyword.

Hence the two cases are:

$$SO < SR \quad K_{ro} = K_{ro}(\text{Table}) \frac{KROR(\text{GridBlock})}{(K_{ro}(SR))(\text{Table})} \quad \text{Eq. 3.101}$$

$$SO > SR \quad K_{ro} = KROR + \frac{KRO - KROR}{K_{romax} - K_{ro}(SR)} (K_{ro}(\text{Table}) - K_{ro}(SR)) \quad \text{Eq. 3.102}$$

**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 KRORG	$K_r$ at critical for the non-directional tables
KRORWX KRORGX	$K_r$ at critical for the +X face of each grid cell
KRORWX- KRORGX-	$K_r$ at critical for the -X face of each grid cell
KRORWY KRORGY	$K_r$ at critical for the +Y face of each grid cell
KRORWY- KRORGY-	$K_r$ at critical for the -Y face of each grid cell
KRORWZ KRORGZ	$K_r$ at critical for the +Z face of each grid cell
KRORWZ- KRORGZ-	$K_r$ 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 KRORG) apply to the same point and the KRORW (KRORG) keyword is ignored.

If the Hysteresis model is active (see keyword [SATOPTS](#)), then the KRO, KRORW, KRORG data will scale the drainage curve. The imbibition curve can be scaled by using the equivalent IKRO (IKRORW, IKROGX, IKROX-, ...) 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

```
-----  
-- 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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-M/rm<sup>3</sup> (METRIC), lb-M/rb (FIELD), gm-M/rcc (LAB), kg-M/rm<sup>3</sup> (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 /
```

## KRWDM

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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  $N = 1 + \text{number of hydrocarbon components} + \text{number of water components}$ :

Column 1	Water 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 <sup>3</sup> ) <sup>-1</sup> (METRIC), (lb-M/rb) <sup>-1</sup> (FIELD), (gm-M/rcc) <sup>-1</sup> (LAB)
Columns COMPS + 2 to COMPS + COMPW + 1	Damage factors for water components UNITS: (kg-M/rm <sup>3</sup> ) <sup>-1</sup> (METRIC), (lb-M/rb) <sup>-1</sup> (FIELD), (gm-M/rcc) <sup>-1</sup> (LAB)

---

**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 $\geq$ 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
/
```

## KRW-IKRWR Scaled end-point water relative permeabilities

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The KRW, KRWR, IKRW and IKRWR keywords scale the relative permeability of water within each grid block. KRW scales the  $K_r$  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:

$$K_{rw} = K_{rw}(\text{Table}) \frac{\text{KRW}(\text{GridBlock})}{K_{rw\max}(\text{Table})} \quad \text{Eq. 3.103}$$

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  $K_r$  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

SR = 1 - SOWCR - SGL in gas/oil or gas/oil/water runs

SR = 1 - SGCR in gas/water runs

where SOWCR and SGCR 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  $SR(\text{Table}) = 1.0 - S_{owcr}(\text{Table}) - S_{gco}(\text{Table})$  or  $SR(\text{Table}) = 1.0 - S_{gcr}(\text{Table})$ .

Hence the two cases are:

$$SW < SR \quad K_{rw} = K_{rw}(\text{Table}) \frac{\text{KRWR}(\text{GridBlock})}{(K_{rw}(\text{SR}))(\text{Table})} \quad \text{Eq. 3.104}$$

$$SW > SR \quad K_{rw} = \text{KRWR} + \frac{\text{KRW} - \text{KRWR}}{K_{rw\max} - K_{rw}(\text{SR})} (K_{rw}(\text{Table}) - K_{rw}(\text{SR})) \quad \text{Eq. 3.105}$$

Fourteen keywords are provided to allow the maximum water relative permeability to be set for each grid cell independently. These keywords are:

KRW      Max water  $K_r$  for the non-directional saturation function tables

KRWX     Max water  $K_r$  for the +X face of each grid cell

KRWX-    Max water  $K_r$  for the -X face of each grid cell

KRWY     Max water  $K_r$  for the +Y face of each grid cell

KRWY-    Max water  $K_r$  for the -Y face of each grid cell

KRWZ     Max water  $K_r$  for the +Z face of each grid cell

KRWZ-    Max water  $K_r$  for the -Z 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

KRWR Y  $K_r$  at residual for the +Y face of each grid cell

KRWR Y-  $K_r$  at residual for the -Y face of each grid cell

KRWR Z  $K_r$  at residual for the +Z face of each grid cell

KRWR Z-  $K_r$  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.

ECLIPSE 300

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 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 /
```

## KVALUES

### 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## KVAN

## Oil component K-value correlation

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$K(P, T) = e^{A-B/(T+C)} / P$$

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)).

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), °R (FIELD), K (LAB), K (PVT-M)

3. C,  $N_c$  values

UNITS: K (METRIC), °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 [KVABCn](#) keywords and the corresponding interpolation points using the [KVMF](#) keyword.

**Example**

```
KVCOMP
 2 /
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## KVCR

## Oil component K-value correlation

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), atma (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     833.4E6
  0          0
 4000        16000
 480         0
/

```

## KVCRS

## Oil component K-value fitting function for surface conditions

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  
/
```

## KVGGEN

## Specify that K-values are to be generated internally

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 KVGGEN 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 KVTABT1 ... KVTABT9 ([KVTABTn](#)) 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.

```
KVGGEN
0.0247 0.2516 0.2407 0.0733 0.4097 10 2500 /
```

## KVMF

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## K-value mole fractions

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 24<sup>th</sup> 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 /
```

## KVTABCn

## Tables of K-values at composition n (n = 1 to 9)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 *KVMF* 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  $10^{-6}$ .

Column 1            The oil phase pressure.

Values should increase monotonically.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Columns 2 to  $N_c$     The corresponding K-values for each component.

+1

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 the vapor and liquid phases respectively.

## Example

```

KVTABC1
  5  .1197  .0192
 50  .0128  .0020
 95  .0071  .0011
140  .0051  .00081
/

```

## KVTABLE

## Tables of K-values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_c$     The corresponding K-values for each component.

+1

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

```
KVTABLE
 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
/
```

## KVTABTn

## Tables of K-values at T=Tn (n = 1 to 9)

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $K_c = y_c / x_c$ , 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

```

KVTABT1
  5  .1197  .0192
 50  .0128  .0020
 95  .0071  .0011
140  .0051  .00081
/

```

## KVTEMP

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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
/

```

## KVWI

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

$$K(P, T) = e^{5.372697 \cdot (1+A_c) \cdot (1-T_{crit}/T)} \cdot P_{crit} / P \quad \text{Eq. 3.106}$$

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
```

## KWTABTn Tables of Kw-values at T=Tn (n = 1 to 9)

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The keyword family 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 Tn, corresponding to the nth value specified in the KWTEMP keyword.

These keywords can only be used with the THERMAL option.

KWTABTn 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 pressure.

Values should increase monotonically.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

Columns 2 to  $N_c$     The corresponding K-values for each component.

+1

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 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:** In a thermal CO2STORE run, the CO<sub>2</sub>/H<sub>2</sub>O component can be defaulted.

---

See also keyword KWTEMP.

### Example

A two component system

```

KWTABT1
  5  .1197  .0192
 50  .0128  .0020
 95  .0071  .0011
140  .0051  .00081
/

```

## KWTEMP

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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
/

```

## **L**

This section contains an alphabetic listing of keywords starting with the letter L.

## LAB

### 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## LAGTRAC

**Activates Lagrangian tracer solver option**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: (Number of components + number of phases) times 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 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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' /
```

## LANGMEXT Extended Langmuir Isotherm

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $2 * NCOMPS$  entries:

Record 1  $P_1 V_1 P_2 V_2 \dots P_{ncomps} V_{ncomps}$

- $P_i$  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: barsa (METRIC), psia (FIELD), atma (LAB), atma (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:  $\text{sm}^3/\text{kg}$  (METRIC), Mscf/ton (FIELD), scc/gm (LAB),  $\text{sm}^3/\text{kg}$  (PVT-M)

DEFAULT: 0.0

Record 2  $P_1 V_1 P_2 V_2 \dots P_{ncomps} V_{ncomps}$

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:

```
LANGMEXT
-- psia      Mscf/ton      coal region 1
276.0       0.9938           --CO2
3951.0     0.482            -- N2
680.0      0.486 /         --CH4
-- psia      Mscf/ton      coal region 2
276.0       0.961           --CO2
3951.0     0.441            -- N2
680.0      0.442/         --CH4
/
```

## LANGMPL Langmuir pressure scaling

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), atma (PVT-M)

2 The corresponding surface gas concentration (the first component in ECLIPSE 300)

Values should be increase monotonically down the column.

UNITS:  $\text{sm}^3/\text{m}^3$  (METRIC), Mscf/ft<sup>3</sup> (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:  $\text{sm}^3/\text{m}^3$  (METRIC), Mscf/ft<sup>3</sup> (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](#)).

ECLIPSE 100

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](#)≥7

```
LANGMUIR
  0.0  0.0
1000.0 0.050
2000.0 0.065
3000.0 0.083
```

```

4000.0 0.100
6000.0 0.110
/
0.0 0.0
1000.0 0.052
2000.0 0.068
3000.0 0.087
4000.0 0.105
6000.0 0.116
7000.0 0.120
/

```

## ECLIPSE 300 Example

With NCOMPS=2, NTCREG=1 and NSSFUN≥20

```

LANGMUIR
--p      CO2      CH4
--bars   SM3/M3   SM3/M3   -- first entry should be zero
0        0        0
1        1.724007349 0.353592028
10       11.89523500 2.976488396
20       17.69503828 5.062945127
30       21.12902337 6.606654448
40       23.39953732 7.795018353
50       25.01221730 8.738067936
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 15.86887937
800      33.72952610 15.99438341
900      33.81682914 16.09337862
1000     33.88699764 16.17346150 /

```

## LANGMULC Langmuir multiplier for each component

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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\*0.9). 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](#).

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 [LANGMULT](#), [GASSATC](#) or [GASADDEC](#).

---

### Example

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword [DIMENS](#):

```
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.

## LANGMULT Langmuir multiplier

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [GASADDEC](#).

---

### 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (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
/
--
-- Solvent retained at zero pressure
--
  0.0  0.050
 1000.0  0.060
 2000.0  0.075
 3000.0  0.093
 4000.0  0.100
 6000.0  0.110
/

```

## LBCCOEF

**Set non-default LBC coefficients**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## LCUNIT

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

$$V_S = \alpha V_{oil} + \beta V_{wat} + \gamma V_{gas} \quad \text{Eq. 3.107}$$

and the flow rate of this quantity,  $Q_S$ , equal to

$$Q_S = \alpha Q_{oil} + \beta Q_{wat} + \gamma Q_{gas} \quad \text{Eq. 3.108}$$

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 (/).

- Up to 4 characters to define the units of  $V_S$ .

DEFAULT: UNIT

### Example

```
LCUNIT
SM3 /
```

## LGR

## Set options for LGR and coarsening

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
--                2      400    0        1        2 /      Interpolation
```

## LGRCOPY

## Allow LGRs to automatically inherit rock property data from host cells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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 LGRCOPY use. Say, the model contains only one LGR, so the LGRCOPY keyword could either be specified in the RUNSPEC section to apply to that LGR, or placed just after CARFIN definition (as in this example). It is possible to have:

```

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

```

This defines PORO and 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 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, LGRCOPY will invoke automatic inheritance of global properties within a REFINE / ENDFIN pair in the EDIT section.

---

**CAUTION:** The 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.

---

## Notes

- It is not possible to use the LGRCOPY keyword with keyword DPGRID.
- This keyword cannot be used with the EXTFIN / ENDFIN set for specifying unstructured local grids.
- The LGRCOPY keyword can be used with the keywords EQUALS, ADD, MULTIPLY, COPY, MAXVALUE, MINVALUE and COPYBOX.

### ECLIPSE 300 only

- For integer-valued keywords (MULTNUM, OPERNUM and FLUXNUM) in the GRID section only, if only a subset of the local cells have values set by the user inside an LGR, LGRCOPY is unable to copy global values to unset LGR cells.
- It is not possible to use the LGRCOPY keyword together with the DX, DY, DZ, TOPS, MIDS, MULTX, MULTY, MULTZ, MULTX-, MULTY- and 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 GRID section is reached.

### ECLIPSE 100 only

- Use of the AMALGAM keyword does not affect LGRCOPY. If used in the GRID section, LGRCOPY must still be specified for each individual LGR as required.

- 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
```

## LGRFREE

## Solves a local grid refinement using local timestepping

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

```
LGRFREE
SOUTH /
/
```

## LGRLOCK **Solves a local grid refinement in-place**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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*' /
/
```

## LGROFF

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## Switches a local grid refinement off

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](#).

---

**Note:** In parallel runs, LGRs cannot be switched on (using the [LGRON](#) keyword) after they have been switched off.

---



---

**Note:** In parallel runs, wells cannot be declared inside of LGRs (using the [WELSPECL](#) keyword) after the LGRs have been switched off.

---

## 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      /
```

## LGRON

## Switches a local grid refinement on

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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      /
```

## LICENSES

## Reserve ECLIPSE 100 option licenses

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

LICENSES option	ECLIPSE option
'eclipse4d'	The Petro-elastic Model
'friction'	The Wellbore Friction Model
'gasfield'	Gas Field Operations Model
'gaslift'	Gas Lift Optimization
'lgr' or ' <i>lgrs</i> '	Local Grid Refinement and Coarsening
'multiseg' or ' <i>multisegwells</i> '	Multisegment Wells
'network' or ' <i>networks</i> '	The Network Option
'unencodedhmd'	Allow unencoded sensitivities output from the Gradient option.

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.

- CO<sub>2</sub> 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

```
LICENSES  
  'network' /  
  'lgr' /  
  'friction' /  
 /
```

## LIFTOPT

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 [WLIPTOPT](#).

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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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.

ECLIPSE 100

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

```
LIFTOPT
-- increment    minimum    optimization    opt in lst
-- size         gradient   interval       NUPCOL its?
-- SM3/DAY      SM3/SM3    DAYS
4E4            5E-3      0.0            YES          /
```

## LILIM

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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  $T^{\text{crit}}$ , given by the Li correlation as:

$$T^{\text{crit}} = \frac{\sum_j T_{cj} \cdot V_{cj} \cdot z_j}{\sum_j V_{cj} z_j} \quad \text{Eq. 3.109}$$

Then define  $f$  as:

$$f = \frac{T^{\text{crit}}}{T^{\text{res}}}$$

$f$  will clearly be 1 if the reservoir temperature is the critical temperature, but become greater than 1 for an oil ( $T^{\text{crit}} > T^{\text{res}}$ ) 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

$$E = \frac{f - f_g}{f_o - f_g} \quad \text{Eq. 3.110}$$

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
.97 1.03 /
```

## LINCOM

## Coefficients for linear combination of phases

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$V_s = \alpha V_{oil} + \beta V_{wat} + \gamma V_{gas} \quad \text{Eq. 3.111}$$

and the flow rate of this quantity,  $Q_s$ , equal to

$$Q_s = \alpha Q_{oil} + \beta Q_{wat} + \gamma Q_{gas} \quad \text{Eq. 3.112}$$

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 [LCUNIT](#) (a default value UNIT is assumed). 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 (/).

- Oil coefficient,  $\alpha$ . This quantity can be a user defined argument (UDA) and can be set with keywords [UDQ](#) and [UDADIMS](#).  
 UNITS: UNIT/sm<sup>3</sup> (METRIC), UNIT/stb (FIELD), UNIT/scc (LAB), UNIT/sm<sup>3</sup> (PVT-M). Default value UNIT can be changed with [LCUNIT](#).  
 DEFAULT: 0.0
- Water coefficient,  $\beta$ . This quantity can be a user defined argument (UDA) and can be set with keywords [UDQ](#) and [UDADIMS](#).  
 UNITS: UNIT/sm<sup>3</sup> (METRIC), UNIT/stb (FIELD), UNIT/scc (LAB), UNIT/sm<sup>3</sup> (PVT-M). Default value UNIT can be changed with [LCUNIT](#).  
 DEFAULT: 0.0
- Gas coefficient,  $\gamma$ . This quantity can be a user defined argument (UDA) and can be set with keywords [UDQ](#) and [UDADIMS](#).  
 UNITS: UNIT/sm<sup>3</sup> (METRIC), UNIT/Mscf (FIELD), UNIT/scc (LAB), UNIT/sm<sup>3</sup> (PVT-M). Default value UNIT can be changed with [LCUNIT](#).  
 DEFAULT: 0.0

---

**Note:** This keyword must be specified before any linearly combined rate targets or limits are defined.

---

### Example

```
LINCOM
0.5 0.5 0.002 /
```

## LINKPERM

**Applies grid block permeabilities to cell faces**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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+ /
```

## LIVEOIL

### 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## LKRO

## Scaled low salinity (oil-wet) end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## LKRORW

## Scaled low salinity (oil-wet) end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## LKRW

## Scaled low salinity (oil-wet) end point water relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## LKRWR

## Scaled low salinity (oil-wet) end point water relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
5. A switch to request a SAVE file to be output (enclosed in quotes).  
YES or NO  
DEFAULT: NO

## ECLIPSE 100 only

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.

- ECLIPSE 300** Fast restarts are not supported for parallel runs.
- ECLIPSE 300** 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.
- ECLIPSE 100** 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.)
- ECLIPSE 100** 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.

```

--
-- 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.

```

--
-- 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.

```

--
-- 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
```



## LOWSALT

## Enables modeling of low salinity effects in the Brine model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [LWSALT](#) 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<sup>3</sup> ([METRIC](#)), lb/stb ([FIELD](#)), gm/scc ([LAB](#)), kg/sm<sup>3</sup> ([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 NTSFUN =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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

## LSCRITS

## Linear solution convergence criteria for individual sub-grids

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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
rad01 40 40 1.0E-7 / --set lgr rad01
/
```

## LSTWNUM Low-salt water-wet saturation function region numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (LSTWNUM region number) and the high salinity water-wet saturation functions ([SURFNUM](#) region number). The low-salinity weighting factors are taken from the LWSLNUM 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 LSTWNUM 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](#):

```
LSTWNUM
144*3 96*4 /
```

## LSNUM

## Low salinity (oil-wet) saturation function region numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [LWSLTNUM](#) 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 /
```

## LSOGCR

## Scaled low salinity (oil-wet) critical oil-in-gas saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

## LSWL

## Scaled low salinity (oil-wet) connate water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 LSWLPC 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 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 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

```

----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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [ENDSCALE](#) 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 /

```

## LTOSIGMA Calculate Sigma from Lx, Ly, Lz

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is optional in dual porosity runs with the viscous displacement option active (keyword VISCD 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:

$$\Sigma = \frac{f_x}{l_x^2} + \frac{f_y}{l_y^2} + \frac{f_z}{l_z^2} \quad \text{Eq. 3.113}$$

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 the using the expression:

$$\Sigma_{gd} = \frac{f_{gd}}{l_z^2} \quad \text{Eq. 3.114}$$

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

$$\text{TR} = \text{CDARCY} \cdot K \cdot V \cdot \sigma \quad \text{Eq. 3.115}$$

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:

$$\text{TR} = \text{CDARCY} \cdot V \cdot \left( \frac{f_x \cdot K_x}{l_x^2} + \frac{f_y \cdot K_y}{l_y^2} + \frac{f_z \cdot K_z}{l_z^2} \right) \quad \text{Eq. 3.116}$$

---

**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
```

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## LUMPDIMS

**Activates and dimensions the LUMPING option**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>th</sup> detailed component of item 1

---

**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.

---



---

**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:

```
LUMPING
C1 2 Z1 Z2/
C2 3 Z3 Z4 Z5/
/
```

## LWKRO

## Scaled low salinity water-wet end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## LWKRORG Scaled low salinity water-wet end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The LWKRORG 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 LWKRORG 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 LWKRORG 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 LWKRORG 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
LWKRORG
 6*0.8
 6*0.8
 6*0.7 /

```

## LWKRORW Scaled low salinity water-wet end point oil relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The LWKRORW 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 LWKRORW 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 LWKRORW 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 LWKRORW 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
LWKRORW
 6*0.5
 6*0.6
 6*0.6 /

```

## LWKRW

## Scaled low salinity water-wet end point water relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /

```

## LWKRWR

## Scaled low salinity water-wet end point water relative permeabilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## LWSLTNUM Low-salt oil-wet saturation function region numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [LSNUM](#) 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 NDIVIX = 8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword [DIMENS](#):

```
LWSNUM
144*1 96*2 /
```

## LWSOGCR Scaled low salinity water-wet critical oil-in-gas saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## LWSOWCR Scaled low salinity water-wet critical oil-in-water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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 /

```

## LWSWCR

## Scaled low salinity water-wet critical water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

## LWSWL

## Scaled low salinity water-wet connate water saturations

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [LWSWLP](#) 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

```

----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 /

```

## LX X direction matrix block sizes for the viscous displacement option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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 [LY](#), [LZ](#) and [LTOSIGMA](#).

### Example

```
----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX           3 16   3   8   1   1   /
LX
98*10.0 /
```

## LY Y direction matrix block sizes for the viscous displacement option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1 100   1  100   1   3   /
LY
10000*10.0
10000*20.0
10000*30.0 /

```

## LZ **Z direction matrix block sizes for the viscous displacement option**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX           1 5  1 5  1  1  /
LZ
5*40.0 5*1.5 15*40.0 /

```

## **M**

This section contains an alphabetic listing of keywords starting with the letter M.

## MAPAXES **Input of pre-processor map origin**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

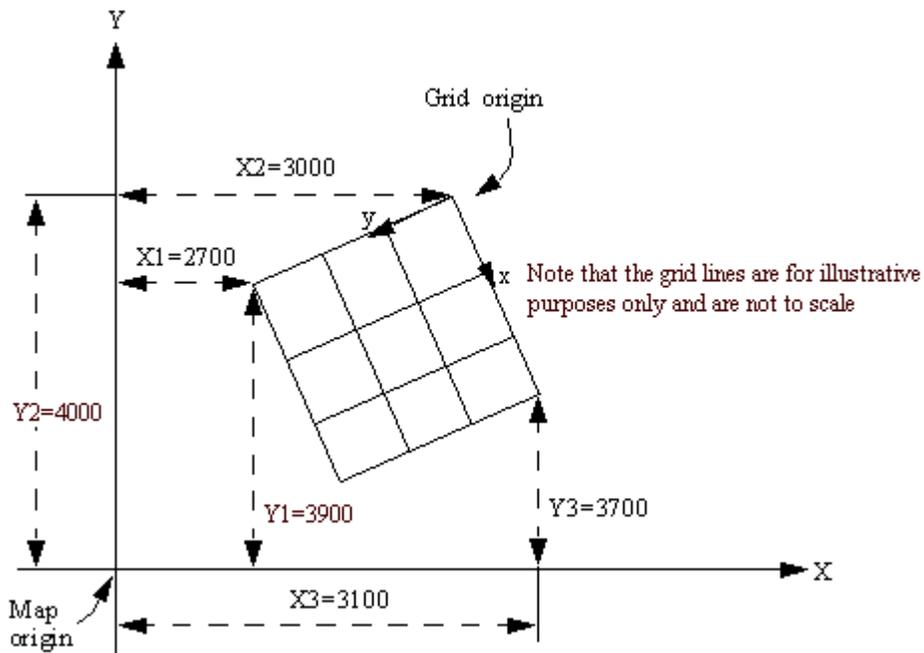


Figure 3.9. Meanings of the MAPAXES keyword entries

See also the [MAPUNITS](#) keyword.

### Example

```
MAPAXES
2700 3900 3000 4000 3100 3700 /
```

## MAPUNITS **Specifies units used for MAPAXES data**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## MASSFLOW Tables of river massflow data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  
 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.
2. Year input should be a 4 digit integer.
3. Time of observation in 24 hour clock format, hh:mm  
 Do not use 24:00; use 00:00 of the following day.
4. Massflow at this time  
 UNITS: m<sup>3</sup>/day (METRIC), ft<sup>3</sup>/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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

3. Material balance error.

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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)}$$

**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
/

```

## MEMORY

**Allocates the required memory at start of run**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 20 000 (twenty thousand) CHAR\*8 strings.

```
MEMORY
1* 20 /
```

## MEMSAVE Indicates use of Up-Front Memory option

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## MESSAGE Specify a message

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

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 [TIME](#), [TSTEP](#) or [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

DEFAULT: 100

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.
  - 3 Report forced timesteps as warning.
  - 4 Report forced timesteps as problem.

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.

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MIDS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$\text{TOPS}_i = \text{MIDS}_i - \frac{\text{DZ}_i}{2} \quad \text{Eq. 3.117}$$

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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.m<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB),  
cP.m<sup>3</sup>/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<sup>-6</sup> for transmissibility and 10<sup>-20</sup> 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), rb (FIELD), 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), rb (FIELD), 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The 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:  $\text{m}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{m}^3$  (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 /
```

## MINPVV

## Sets minimum pore volumes cells must have to be active

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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), rb (FIELD), 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 [PINCH](#) or [PINCHOUT](#), and non-neighbor connections must be enabled (they are enabled by default unless keyword [NONNC](#) is encountered in the [RUNSPEC](#) section).

Cells which have been set inactive via the [ACTNUM](#) keyword remain so even if their pore volume exceeds the threshold; using 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 [MINPV](#) or MINPVV, this is the default for all cells.

A list of all the cells that are made inactive can be obtained by the argument 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 [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

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1  8  2  2  2  3
MINPVV
1000.0 1000.0 1000.0 1000.0 1000.0 1000.0 1000.0 1000.0
 500.0 500.0 500.0 500.0 500.0 500.0 500.0 500.0 /

```

## MINROCKV **Minimum rock volume for active cell**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{rm}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{rm}^3$  (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

```
MINROCKV
0.001 /
```

## MINRV

# 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](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MINVALUE

## Apply a minimum limit to an array in the current box

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword DIMENS)}$$

---

**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
/

```

## MISC

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

```
MISC
0.0 0.0
0.1 0.5
0.3 1.0
1.0 1.0 /
```

## MISCEXP

### Miscibility exponent

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 /
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MISCIBLE

**Miscible treatment of hydrocarbons**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Note that the TWOPOINT upstreaming option is not available in Solvent runs.

**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 /
```

## 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  
/
```

## MISCNUM

## Miscibility region numbers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- 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 /
```

## MISCSTR

## Miscibility surface tension reference

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## MISCSTRP **Miscibility surface tension reference pressure**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

```
3 3.2 3.9 1* /  
1* 3.6 1* 1* /
```

## MIXACTG

## Specify gas activity functions for aqueous components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 CO<sub>2</sub> if the [CO2SOL](#) option is being used and H<sub>2</sub>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), atma (LAB), atma (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/K<sup>2</sup> (METRIC), 1/°R<sup>2</sup> (FIELD), 1/K<sup>2</sup> (LAB), 1/K<sup>2</sup> (PVT-M)

## Example

```
MIXACTG
 2/
--Pressure   bi1      bi2      bi3
 1.0000     0.0001   0.0001   0.0001
100.0000    0.0001   0.0001   0.0001
200.0000    0.0001   0.0001   0.0001
```

```
/
 4/
--Pressure  bi1      bi2      bi3
 1.0000    0.0001  0.0001  0.0001
100.0000   0.0001  0.0001  0.0001
200.0000   0.0001  0.0001  0.0001
/
/ null record terminates 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub> if the [CO2SOL](#) option is being used and H<sub>2</sub>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), atma (LAB), atma (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<sup>2</sup> (METRIC), 1/°R<sup>2</sup> (FIELD), 1/K<sup>2</sup> (LAB), 1/K<sup>2</sup> (PVT-M)

## Example

```
MIXACTO
 2/
--Pressure   ai1      ai2      ai3
 1.0000     0.0001   0.0001   0.0001
100.0000    0.0001   0.0001   0.0001
200.0000    0.0001   0.0001   0.0001
```

```
/
  4/
--Pressure   ai1      ai2      ai3
  1.0000     0.0001   0.0001   0.0001
 100.0000    0.0001   0.0001   0.0001
 200.0000    0.0001   0.0001   0.0001
/
/ null record terminates table
```

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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
MIXWPERM
```

## MLANG

## Maximum surface gas concentration

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

The keyword can be used in conjunction with the [EQUALS](#), [ADD](#) and [MULTIPLY](#) keywords.

See also keywords [LANGMUIR](#), [MLANGSLV](#) and [GASSATC](#).

## Example

```

----- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (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   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

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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
```

## MPFA

## Specify options for tensor permeabilities

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [NEWTRAN](#) calculation. In the absence of faults, it is equivalent to the standard [HALFTRAN](#) calculation when the vector  $K \underline{d}$  is parallel to the common cell face normal in the half of each connecting cell adjacent to the common face, where  $\underline{d}$  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

---

**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 *ECLIPSE Technical Description* for further information.

---

**Note:** The use of the MPFA option is not allowed with the [WARP](#) solver. Please use the pre-2002A solver (ECLSOL) instead by setting [item 66](#) in `OPTIONS3` to a non-zero value.

---

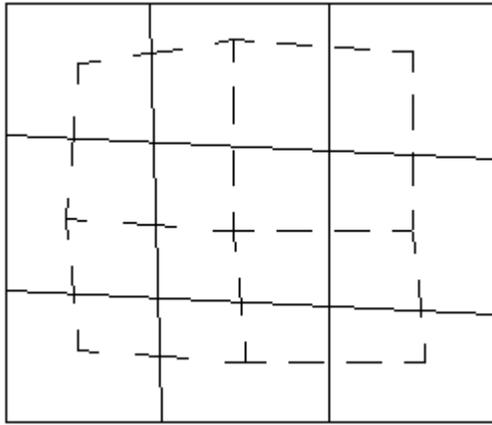


Figure 3.10. Stencil for nine point MPFA scheme in 2D

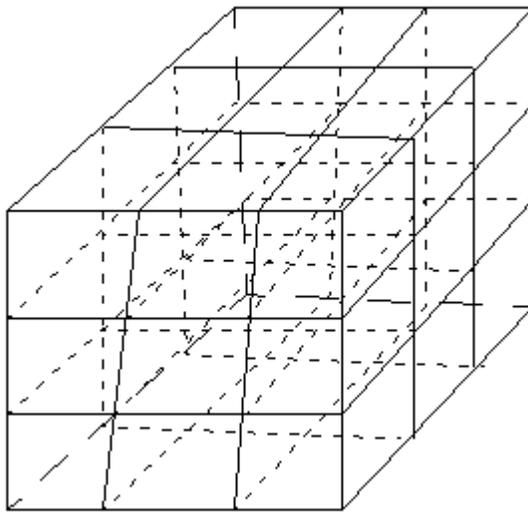


Figure 3.11. Stencil for 'O'

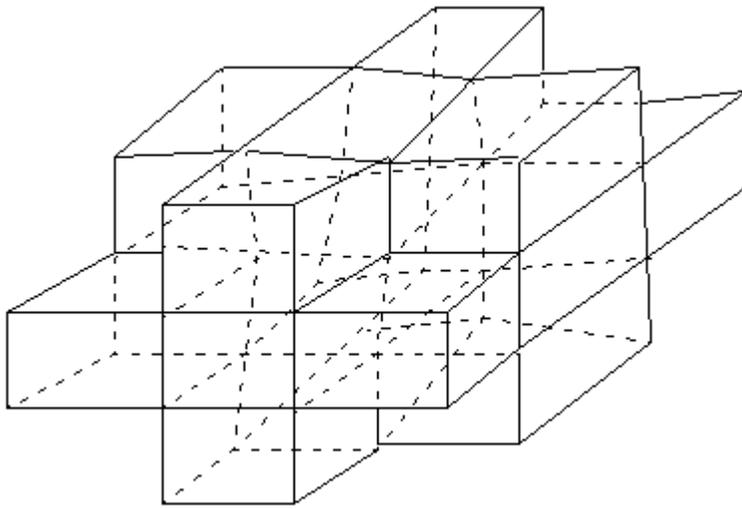


Figure 3.12. Stencil for 'U'

### Example

```
MPFA  
1 1 /
```

## MPFANUM

## Identifies regions where the multipoint flux discretization is used

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
 0 0 0 0
 16*0 /

```

## MPFNNC

## Explicit entry of multi-point flux non-neighbor connections

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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

```
MPFNNC
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 /
/
/
```

## MSFN

## Miscible saturation functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$k_{rsg} = M_{krsg} \left( \frac{S_g + S_s - S_{gc}}{S_n - S_{gc} - S_{or}} \right) k_m(S_n) \quad \text{Eq. 3.118}$$

$$k_{ro} = M_{kro} \left( \frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} \right) k_m(S_n) \quad \text{Eq. 3.119}$$

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**

$k_m(S_n)$  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

$$k_{rsg} = \frac{S_g + S_s - S_{gc}}{S_n - S_{gc} - S_{or}} k_m(S_n) \quad \text{Eq. 3.120}$$

$$k_{ro} = \frac{S_o - S_{or}}{S_n - S_{gc} - S_{or}} k_m(S_n) \quad \text{Eq. 3.121}$$

See also the [MISC](#), [PMISC](#), [SSFN](#), [SORWMIS](#) and [SGCWMIS](#) keywords.

## Example

With NTSFUN=2 and NSSFUN≥6

```

MSFN
-- Default table
-- Sfrac  MKrsg  MKro
0.0      0.0    1.0
1.0      1.0    0.0  /
-- Modified table
0.0      0.0    1.0
0.4      0.2    0.5
0.6      0.4    0.4
1.0      1.0    0.0  /

```

## MSGFILE

**Control output of the message file**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The keyword should be followed by an integer.

The possible values are:

- Control of the MSG file output
  - A MSG file is produced
  - 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**

```
MSGFILE
0 /
```

## MULSGGD **Dual porosity matrix-fracture coupling multiplier for oil-gas gravity drainage**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `MULSGGDV` 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

----- 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 0.5 0.5
  0.3 0.3 0.3 0.3 0.3 0.3 0.3 0.3 /

```

## MULTFLT

**Modifies the transmissibility across a named fault**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MULTIPHA Perform multi-phase flash

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## MULTIPLY **Multiply array by a constant in current box**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$1 \leq IX1 \leq IX2 \leq NDX$$

$$1 \leq JY1 \leq JY2 \leq NDY$$

$$1 \leq KZ1 \leq KZ2 \leq NDZ,$$

where NDX, NDY, NDZ are the limits in the current BOX.

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.

[TNUM](#) arrays can also be used with the MULTIPLY keyword.

ECLIPSE 100

In the REGIONS section only integer values should be used.

See also keywords BOX, ENDBOX, [EQUALS](#), [ADD](#) and [COPY](#).

## 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
FLUXNUM	MULTNUM	MPFANUM
MIDS		

*ECLIPSE 300 only*

*ECLIPSE 300 only*

### EDIT section arrays

PORV	DEPTH	
TRANX (TRANR)	TRANY (TRANHT)	TRANZ
DIFFX (DIFFR)	DIFFY (DIFFHT)	DIFFZ
HEATTX (HEATTR)	HEATTY (HEATHT)	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		
EOSNUM		

*ECLIPSE 300 only*

**SOLUTION section arrays**

	PRESSURE	SWAT	SGAS
	RV	RS	TBLK
<i>ECLIPSE 100 only</i>	GI	OILAPI	SALT
<i>ECLIPSE 100 only</i>	GASCONC	SOLVCONC	SOLVFRAC
<i>ECLIPSE 100 only</i>	SSOL	SURF	SFOAM
<i>ECLIPSE 100 only</i>	SPOLY		
<i>ECLIPSE 300 only</i>	SOIL	PBUB	PDEW
<i>ECLIPSE 300 only</i>	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
      TRANX      500          3 8 1 10 3 3 /
      TRANZ       0          / defaults to last specified box
      PORV      100.0        / defaults to last specified box
/

```

**Example 2**

In the GRID section:

```

----- ARRAY      CONSTANT ----- BOX -----
MULTIPLY

```

```
PERMX 0.1      3 8 1 10 3 3 /  
PERMZ 0        / defaults to last specified box  
PORO  0.2      / defaults to last specified box  
/
```

## MULTIREG **Multiplies an array by a constant in a given region**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

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*

FLUXNUM            MULTNUM            MPFANUM

*ECLIPSE 300 only*

MIDS

### EDIT section arrays

PORV                    DEPTH

TRANX (TRANR)    TRANY (TRANHT)    TRANZ

DIFFX (DIFFR)    DIFFY (DIFFHT)    DIFFZ

HEATX (HEATR)    HEATY (HEATHT)    HEATZ (HEATZ)

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

*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

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      PERMZ      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:

Keyword	Description
<a href="#">DIFFMMF</a>	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.
<a href="#">MULSGGD</a> <a href="#">MULSGGDV</a>	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.
<a href="#">MULTSIG</a> <a href="#">MULTSIGV</a>	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.
<a href="#">MULTX</a> <a href="#">MULTY</a> <a href="#">MULTZ</a>	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.

Keyword	Description
PERMX PERMY PERMZ	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.
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 SIGMAV	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.
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 3.20: Additional notes for MULTIREG in the SCHEDULE section

## Examples

### Example 1

Where  $NX = 4$ ,  $NY = 4$ ,  $NZ = 1$ :

```

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

$NX = 6$ ,  $NY = 8$ ,  $NZ = 1$ :

```

MULTNUM
1 1 2 2 2 2
1 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 4 1 3 3
4 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 [DUALPORO](#) 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.

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MULTNUM

## Defines regions for applying inter-region transmissibility multipliers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 2<sup>nd</sup> 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## MULTOUTS **Indicates that summary output files are multiple**

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## MULTPV

## Pore volume multipliers

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 MULTR has been aliased to MULTR. However, MULTR can still be used in the SCHEDULE section.

---

See also keyword [MULTR-](#).

### Example

```

----- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
      5   5   3   8   4   6   /
MULTR
18*0 /

```

## MULTR-

## Transmissibility multipliers in the negative radial direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
ENDBOX
```

## MULTREAL **Activates multi-realization licensing**

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
  'C38CCB5D-5D9A-4544-BCA4-42B13866A8B5' NO /
```

## MULTREGD **Multiplies the diffusivity between flux or MULTNUM regions**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- From region number (I)
 

A non-positive or defaulted value acts as a wildcard on all regions.

DEFAULT: Negative
- To region number (J)
 

A non-positive or defaulted value acts as a wildcard on all regions.

DEFAULT: Negative
- Diffusivity multiplier for all diffusivities connecting regions I and J
- Direction(s) in which to apply the multiplier:
 

Either X, Y, Z, XY, YZ, XZ or XYZ.

DEFAULT: XYZ
- 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 NOAQUNNC 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 **COPY** ing 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

```
MULTREGD
-- 'From' region      'To' region      Diffusivity      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.
/
```

## MULTREGH **Multiplies the rock thermal conductivity between flux or MULTNUM regions**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#). MULTREGH 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 (/).

- From region number (I)
  - A non-positive or defaulted value acts as a wildcard on all regions.
  - DEFAULT: Negative
- To region number (J)
  - A non-positive or defaulted value acts as a wildcard on all regions.
  - DEFAULT: Negative
- Rock thermal conductivity multiplier for all thermal conductivities connecting regions I and J
- Direction(s) in which to apply the multiplier:
  - Either X, Y, Z, XY, YZ, XZ or XYZ.
  - DEFAULT: XYZ
- 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.

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 NOAQUNNC 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 **COPY** ing 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

```
MULTREGH
-- 'From' region      'To' region      Conductivity
                    multiplier
      1                2                0.5 /
      1                3                0.0 /
      4                4                0.4
      5                1*              0.75 / Between region 5 and any region.
/
```

## MULTREGP **Multiplies the pore volumes for specific flux or MULTNUM region**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
   2      0.0
   4      0.4
   5      0.75
   O/
/
```

## MULTREGT **Multiplies the transmissibility between flux or MULTNUM regions**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $I \neq J$ , 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 NOAQUNNC 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 [COPY](#) ing 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

```
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.
/
```

## MULTSAVE Indicates that save files are multiple

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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**

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1   8   1   1   1   2   /
MULTSIGV
  0.5 0.5 0.5 0.5 0.5 0.5 0.5
  0.3 0.3 0.3 0.3 0.3 0.3 0.3 /

```

## MULTTHT

## Transmissibility multipliers in the theta direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

## MULTTHT- Transmissibility multipliers in the negative theta direction

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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
```

## MULTX

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 [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).

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 [MULTX-](#).

### Non-neighbor connections

Any non-neighbor connections generated due to faults have transmissibilities that reflect the MULTX values. 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 the reading of 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 a modifier 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).

Note that the effect of MULTX in the SCHEDULE section is cumulative. In other words, each time MULTX is encountered in the 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 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 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

```
----- 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 [TRANXY](#) 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 [TRANXY](#) 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [TRANXY](#) 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 [TRANXY](#) 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 [TRANXY](#) 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).

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## 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

```
----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1 11  1 19  2  2 /
MULTZ-
  209*1000 /
ENDBOX
```

## MW

## Molecular weights

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## MWS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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)

## MWW

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## MWWS

## Molecular weights for water components for surface EoS

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

## 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
```

## NCOMPS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 *i* 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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELSPEC](#)s mnemonic in the [RPTSCHE](#)D 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 /
/
    
```

## NEI

## Specify composition for non-equilibrium initialization

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

DEFAULT: 10

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 [GCONTROL](#).

If the value entered here is greater than the tolerance fraction entered in item 1 of keyword [GCONTROL](#), the value entered in [GCONTROL](#) 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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `VFP`PROD.

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: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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:

$$ALQ_i = ALQ_1 + (ALQ_N - ALQ_1) \left( \frac{i-1}{N-1} \right) \quad \text{Eq. 3.122}$$

$$CONS_i = CONS_N \cdot \frac{i}{N} \quad \text{Eq. 3.123}$$

where  $ALQ_N$  is defined in item 6,  $CONS_N$  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 `VFP`PROD.

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  rate  group  lev  lev 1  num
   N4     N3     PL-A    GAS    1*   50.0  1*   1*   TEMP  1    40.0  /
   N6     N5     PL-B    GAS    1*  100.0 1000  G-B3  MULT  3    40.0  /
/
    
```

## NETWORK Sets dimensions for extended network model

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

## 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
```

## NEWTRAN Specifies block corner transmissibilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

## NEXT

# Sets a maximum value for the next timestep

This is an alias for the keyword [NEXTSTEP](#).

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## NEXTSTEP Sets a maximum value for the next timestep

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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.

```
NGASREM
NODE-5    20000  /
/
```

### 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    1*    0.1  /
/
```

## NINENUM

## Identifies the extent of the nine-point region

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
0 0 0 0
1 1 0 0
1 1 0 0
0 0 0 0
0 0 0 0
16*0 /
```

## NINEPOIN

**Select the nine-point option**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## NMATOPTS Options for the discretized matrix (E100) or multi porosity (E300) models

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `NMATRIX` keyword in the `RUNSPEC` section.

For ECLIPSE 300 the keyword means that the model (see "Multi porosity" in the *ECLIPSE Technical Description*), activated with `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 `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 `LINEAR`, `RADIAL` or `SPHERICAL`

In ECLIPSE 300 there is an additional possibility - `UNIFORM`. This produces a uniform grid within the matrix cells.

A further alternative possibility is `VERTICAL`, which is used to model gravity drainage, whereas the previous options model transient effects in the flow from matrix to fracture. `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 *ECLIPSE Technical Description*. If this option is used then the keyword `GRAVDRE` must have been specified in the `RUNSPEC` section.

In ECLIPSE 100, for any discretized matrix option, the minimum number of global cells in the X direction (keyword `DIMENS`) is 2.

In all cases except `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 `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.

DEFAULT: `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 `UNIFORM` or `VERTICAL` is selected. `VERTICAL` assumes uniform distribution of the matrix volume in the same way as `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.

DEFAULT: 0.1

3. Method for controlling the partitioning (ECLIPSE 300 only).

Either `FPORV` or `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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## NNC

## Set non-neighbor connection values explicitly

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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.

*ECLIPSE 100 only* 10. Pressure table number to be associated with flow from the first to the second cell (IPT1)

*ECLIPSE 100 only* 11. Pressure table number to be associated with flow from the second to the first cell (IPT2)

*ECLIPSE 100 only* 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).

*ECLIPSE 100 only*

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).

UNITS: 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.

UNITS: 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).

UNITS: m<sup>-2</sup> (METRIC), ft<sup>-2</sup> (FIELD), cm<sup>-2</sup> (LAB), m<sup>-2</sup> (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).

UNITS: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (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).

UNITS: 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 DX<sub>i</sub> and DX<sub>j</sub> 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 `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 /
.
.
/

```

## NNCGEN

## Set non-neighbor connection values between cells in any two grids explicitly

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- Grid of first cell  
DEFAULT: GLOBAL
- I-index of first cell joined to the non-neighbor connection (IX)
- J-index of first cell joined to the non-neighbor connection (IY)
- K-index of first cell joined to the non-neighbor connection (IZ)
- Grid of second cell  
DEFAULT: GLOBAL
- I-index of second cell joined to the non-neighbor connection (JX)
- J-index of second cell joined to the non-neighbor connection (JY)
- K-index of second cell joined to the non-neighbor connection (JZ)
- Transmissibility value of the non-neighbor connection (TRAN)

UNITS: cP.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/day/atm (PVT-M)

DEFAULT: 0.0

- Diffusivity value of the non-neighbor connection  
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)

DEFAULT: 0.0

- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## NOCIRCLE Turn off completion of circle option for local radial refinements

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

### ECLIPSE 100

If the node in item 1 corresponds to a **well group**, this flag indicates whether the corresponding 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 (see "[Group production control in the network option](#)" in the *ECLIPSE Technical Description*). The uptree branch **can** be given a VFP table number other than 9999 in this case, in [ECLIPSE 100](#) only. Effectively, the choke's pressure drop is added to the pressure drop along the branch.

**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** 1). 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.

*ECLIPSE 300 only* 7. 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

```

NODEPROP
-- Node      Fixed      Automatic choke      Add      Choke control
-- name      pressure  un uptree branch  lift gas?      group
FIELD      200      /
GR-A1      1*      1*      YES      /
GR-A2      1*      1*      YES      /
GR-B1      1*      YES      YES      /
GR-B2      1*      YES      YES      /
/

```

## NODPCDT

### Disable DPCDT option

This keyword disables the [DPCDT](#) option.

This keyword has no associated data

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## NODPCO

**No dual porosity condensation**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## NODPPM

# No dual porosity permeability multiplier

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## NOFREEZE **Suppress use of the FREEZEPC option**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## NOGGF

### 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*.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## NOHMD

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

## 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' /
```

## NOHMO

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## NOHYST

### Disable 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## NOINSPEC

**Disables the initial index file output**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## NOMIX

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [ODD3P](#) relative permeability and capillary pressure hysteresis model is selected.

## NOMONITO **Disable output for run-time monitoring**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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
```

## NONNC

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## NOWARN

## Suppress ECLIPSE warning messages

If this keyword is specified, all ECLIPSE warning messages are suppressed after this keyword has been read.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

---

**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**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## NPROCX

## Set number of processors in the y-direction in a parallel run

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 NPROCX.

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [RPTRST](#) 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
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

```
NSTACK
50 /
```

## NTG

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## NUPCOL

## Number of iterations to update well targets

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword can be used in the extended network (constructed with keywords [BRANPROP](#) and [NODEPROP](#), 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: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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.

```
NWATREM
NODE-5    20000    /
/
```

## 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 I2-I1+1 values terminated with a slash (/), where I1 and I2 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

```

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 /

```

## NYFIN

## Number of local cells in each global cell of an LGR in Y direction

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#).

## O

This section contains an alphabetic listing of keywords starting with the letter O.

## ODD3P

## Request ODD3P three-phase relative permeability and capillary pressure model

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## OFM

## Requests output of SUMMARY data in OFM format

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## OILAPI

## Initial oil API values, for API tracking option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## OILCOMPR Oil compressibility

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>2</sup> (METRIC), 1/°R<sup>2</sup> (FIELD), 1/K<sup>2</sup> (LAB), 1/K<sup>2</sup> (PVT-M)  
 DEFAULT: 0.0

### Example

```
OILCOMPR
--Compressibility   therm.expansion   in Field units
--                  coefficients
   5.00E-06         3.80E-04   0.0   /
```

## OILMW

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## OILSPECH **Oil specific heat**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/°R (FIELD), J/gm/K (LAB), kJ/kg/K (PVT-M)

DEFAULT: 0.0

### Example

```
OILSPECH
-- In Field units
0.5 /
```

## OILVINDX

## Oil viscosity index

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$f_i(0) = 0 \text{ and } f_i(1) = 1$$

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 NPPVT see keyword [TABDIMS](#)).

The oil phase viscosity is calculated from the component viscosities:

$$\mu_o = \mu_T(T) = \prod_{i=1}^{N_c} \mu_{oi}^{f_i(x_i)} \quad \text{Eq. 3.124}$$

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:

$$\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})} \quad \text{Eq. 3.125}$$

where

$\mu_T$  is the viscosity defined in equation [3.124](#).

$\mu_p$  is the viscosity from the [PVCO](#) keyword.

$P_{ref}$  is the reference pressure defined by the `VISCREF` keyword.

See "Thermal option" in the *ECLIPSE Technical Description*.

## Example

```
OILVINDX
 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
/
```

## OILVISCC

## Oil viscosity-temperature correlation

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\text{ASTM: } \log_{10}(\mu_{oi} + A) = B \cdot T^{-C} \quad \text{Eq. 3.126}$$

$$\text{Andrade: } \log_{10}(\mu_{oi}) = A + B / T \quad \text{Eq. 3.127}$$

$$\text{Vogel: } \log_{10}(\mu_{oi}) = A + B / (T + C) \quad \text{Eq. 3.128}$$

$$\text{logarithmic: } \log_{10}(\mu_{oi}) = A + B \cdot \log_{10}(T) \quad \text{Eq. 3.129}$$

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:**

$$\mu_{oi}(T_1) + 1 > \mu_{oi}(\infty) \text{ and } \mu_{oi}(T_2) + 1 > \mu_{oi}(\infty)$$


---

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:

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 9<sup>th</sup> parameter of [TABDIMS](#).

The oil phase viscosity  $\mu_o$  is calculated from:

$$\mu_o = \mu_T(T) = \prod_{i=1}^{N_c} \mu_{oi}^{f_i(x_i)} \quad \text{Eq. 3.130}$$

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 [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:

$$\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{vref})} \quad \text{Eq. 3.131}$$

where

$\mu_T$  is the viscosity from this keyword.

$\mu_p$  is the viscosity from the [PVCO](#) keyword.

$P_{vref}$  is the reference pressure defined by the [VISCREF](#) keyword.

## Examples

### Example 1

In this example there is one reservoir EoS region.

```

OILVISCC
--
-- EoS region 1
--
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 /
/
```

## Example 2

In this example there are two reservoir EoS regions.

```
OILVISC
--
-- EoS region 1
--
ASTM CORRELATION
--TEMP  VISC1  VISC2  VISC3
100     1*    700.0  1200.0
200     1*    70.0   120.0
        1*    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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (METRIC), °R (FIELD), K (LAB), K (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:

$$\mu_o = \mu_T(T) = \prod_{i=1}^{N_c} \mu_{oi}^{f_i(x_i)} \quad \text{Eq. 3.132}$$

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 [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:

$$\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{\text{ref}})} \quad \text{Eq. 3.133}$$

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

```
OILVISCF  
-- LIGHT COMP LIGHTER COMP  
4.49E9 20.0E9  
-3.4 -3.4  
/
```

## OILVISCT

## Oil viscosity versus temperature data

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $R_s$  value given in the [VISCREF](#) keyword. The oil viscosity at the prevailing pressure and  $R_s$  is calculated as follows:

$$\mu_o = \mu_T(T) \frac{\mu_p(P, R_s)}{\mu_p(P_{\text{ref}}, R_{s \text{ ref}})} \quad \text{Eq. 3.134}$$

where

$\mu_T$  Viscosity from the OILVISCT keyword.

$\mu_p$  Viscosity from the [PVTO](#) (or [PVDO](#)) keyword.

$P_{\text{ref}}$  Reference pressure from the [VISCREF](#) keyword.

$R_{s \text{ ref}}$  Reference  $R_s$  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 `NTPVT = 1` and `NPPVT ≥ 5`

```
OILVISCT
 10 60
 100 10
 150 2
 200 0.8
 250 0.5
/
```

## 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 `NTPVT` 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 `NPPVT` (see keyword `TABDIMS`).

The oil phase viscosity is calculated from the component viscosities:

$$\mu_o = \mu_T(T) = \prod_{i=1}^{N_c} \mu_{oi}^{f_i(x_i)} \quad \text{Eq. 3.135}$$

where

$\mu_{oi}$  are the component viscosities

$x_i$  are the component mole fractions in the oil phase

$f_i(x_i)$  are the viscosity indices defined by the `OILVINDX` keyword.

For temperature values outside the interval specified, constant value extrapolation is performed.

If both the `PVCO` and `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:

$$\mu_o = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})} \quad \text{Eq. 3.136}$$

where

$\mu_T$  Viscosity from this keyword.

$\mu_p$  Viscosity from the `PVCO` keyword.

$P_{ref}$  The reference pressure defined by the `VISCREF` keyword.

See "Thermal option" in the *ECLIPSE Technical Description*.

**Note:** Oil viscosity in the `DEADOIL` option cannot be lower than 0.1 cP. It will be reset to 0.1 cP.

## Example

```
OILVISCT
 75 1* 5780
100 1* 1380
150 1* 187
200 1* 47
250 1* 17.4
300 1* 8.5
350 1* 5.2
500 1* 2.5
15000 1* 2.4999
/
```

## OILVTIM

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 EQUIL keyword.
2. The sign of the capillary pressure (4<sup>th</sup> item of the EQUIL keyword) at the water-oil contact is reversed. A positive value now implies  $P_{oil} > P_{water}$  at the contact.

## OLDTRAN **Specifies block center transmissibilities**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The use of OLDTRAN implies that transmissibilities are calculated from cell [DX](#) (or [DR](#)), [DY](#) (or [DTHETA](#)) and [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 [DX](#) (or [DR](#)), [DY](#) (or [DTHETA](#)) and [DZ](#) keywords.

(See also keywords [NEWTRAN](#) and [OLDTRANR](#).)

## OLDTRANR Specifies alternative block center transmissibilities

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The use of OLDTRANR implies that transmissibilities are calculated from cell [DX](#) (or [DR](#)), [DY](#) (or [DTHETA](#)) and [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.

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.

## OMEGAA

Overrides default  $\Omega_a$  values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: 0.45723529, RK, SRK and ZJ: 0.4274802

## Example

```
OMEGAA
4*0.457 3*0.461 0.462 /
```

## OMEGAIDE Overrides default $\Omega_a$ values for detailed composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 OMEGAIDE 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: 0.45723529, RK, SRK and ZJ: 0.4274802

### Example

```
OMEGAIDE
4*0.457 3*0.461 0.462 /
```

## OMEGAAS Overrides default surface $\Omega_a$ values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## OMEGAASD Overrides default surface $\Omega_a$ values for detailed compositions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [OMEGAIDE](#) 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 OMEGAIDE.

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 /
```

## OMEGAB

Overrides default  $\Omega_b$  values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
OMEGAB
0.0777 2*0.078 0.07677 /
```

## OMEGABDE Overrides default $\Omega_b$ values for detailed compositions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
OMEGABDE
0.0777 2*0.078 0.07677 /
```

## OMEGABS **Overrides default surface $\Omega_b$ values**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: 0.07779607, RK, SRK and ZJ: 0.08664035

### Example

```
OMEGABS
0.0777 2*0.078 0.07677 /
```

## OMEGABSD Overrides default surface $\Omega_b$ values for detailed compositions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

# OPERATE **Performs operations on arrays**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

Result	item 8	item 9	item 10	item 11
$R_i = \alpha X_i + \beta$	'MULTA'	X	$\alpha$	$\beta$
$R_i = R_i + \alpha X_i^\beta$	'POLY'	X	$\alpha$	$\beta$
$R_i = 10^{(\alpha + \beta X_i)}$	'SLOG'	X	$\alpha$	$\beta$
$R_i = \log_{10}(X_i)$	'LOG10'	X	n/a	n/a
$R_i = \ln(X_i)$	'LOGE'	X	n/a	n/a
$R_i = \frac{1}{X_i}$	'INV'	X	n/a	n/a

Result	item 8	item 9	item 10	item 11
$R_i = \alpha X_i$	'MULTX'	X	$\alpha$	n/a
$R_i = \alpha + X_i$	'ADDX'	X	$\alpha$	n/a
$R_i = X_i$	'COPY'	X	n/a	n/a
$R_i = \min(\alpha, X_i)$	'MAXLIM'	X	$\alpha$	n/a
$R_i = \max(\alpha, X_i)$	'MINLIM'	X	$\alpha$	n/a
$R_i = \alpha X_i^\beta$	'MULTP'	X	$\alpha$	$\beta$
$R_i = \text{abs}(X_i)$	'ABS'	X	n/a	n/a
$R_i = X_i R_i$	'MULTIPLY'	X	n/a	n/a

Table 3.21: Valid operation types

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
MIDS		

ECLIPSE 300 only

### EDIT section arrays

PORV	DEPTH	
TRANX (TRANR)	TRANY (TRANHT)	TRANZ
DIFFX (DIFFR)	DIFFY (DIFFHT)	DIFFZ
HEATTX	HEATY	HEATZ
ROCKV	BVRMX	BVRMY
BVRMZ	BULKV	DTPX1

ECLIPSE 300 only

ECLIPSE 300 only

ECLIPSE 300 only

ECLIPSE 300 only      DTPX2          DTPY1          DTPY2  
ECLIPSE 300 only      DTPZ1          DTPZ2

### PROPS section arrays

SWL SWCR SWU  
SGL SGCR SGU  
KRW KRO KRG  
PCG PCW

### REGIONS section

SATNUM PVTNUM EQLNUM  
IMBNUM FIPNUM ENDNUM  
ROCKNUM MISCNUM KRNUM  
WH2NUM WH3NUM

ECLIPSE 300 only      EOSNUM

### SOLUTION section arrays

PRESSURE SWAT SGAS  
RV          RS      TBLK  
SOIL          XMF      YMF  
TEMP          PBUB      PDEW

ECLIPSE 300 only

ECLIPSE 300 only

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  $SWL = F_n(PORO)$ . For use of the OPERATE 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 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 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).
- In the 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 EQUALS, ADD and MULTIPLY. These keywords will always be applied first, even if OPERATE 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.

## 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 /
```

```
PERMZ 6* 'COPY' PERMX / defaults to last specified box
/
```

In the REGIONS section:

```
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:

```
--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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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  $i^{\text{th}}$  element of the argument array, and  $R_i$  the  $i^{\text{th}}$  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.

Result	item 3	item4	item 5	item 6
$R_i = \alpha X_i + \beta$	'MULTA'	X	$\alpha$	$\beta$
$R_i = R_i + \alpha X_i^\beta$	'POLY'	X	$\alpha$	$\beta$
$R_i = 10^{(\alpha + \beta X_i)}$	'SLOG'	X	$\alpha$	$\beta$
$R_i = \log_{10}(X_i)$	'LOG10'	X	n/a	n/a
$R_i = \ln(X_i)$	'LOGE'	X	n/a	n/a
$R_i = \frac{1}{X_i}$	'INV'	X	n/a	n/a
$R_i = \alpha X_i$	'MULTX'	X	$\alpha$	n/a
$R_i = \alpha + X_i$	'ADDX'	X	$\alpha$	n/a
$R_i = X_i$	'COPY'	X	n/a	n/a
$R_i = \min(\alpha, X_i)$	'MAXLIM'	X	$\alpha$	n/a
$R_i = \max(\alpha, X_i)$	'MINLIM'	X	$\alpha$	n/a
$R_i = \alpha X_i^\beta$	'MULTP'	X	$\alpha$	$\beta$
$R_i = \text{abs}(X_i)$	'ABS'	X	n/a	n/a
$R_i = X_i R_i$	'MULTIPLY'	X	n/a	n/a

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

DX (DR)	DY (DTHETA)	DZ
PERMX (PERMR)	PERMY (PERMTHT)	PERMZ
MULTX (MULTR)	MULTY (MULTTHT)	MULTZ
PORO	NTG	DZNET
TOPS		
DIFFMX (DIFFMR)	DIFFMY (DIFFMTHT)	DIFFMZ
MIDS		

ECLIPSE 300 only

### EDIT section

PORV	DEPTH	
TRANX (TRANR)	TRANY (TRANHT)	TRANZ
DIFFX (DIFFR)	DIFFY (DIFFHT)	DIFFZ
HEATTX	HEATTY	HEATTZ
ROCKV	BVRMX	BVRMY
BVRMZ	BULKV	DTPX1
DTPX2	DTPY1	DTPY2
DTPZ1	DTPZ2	

ECLIPSE 300 only

### PROPS section

SWL	SWCR	SWU
SGL	SGCR	SGU

KRW KRO KRG

PCG PCW

### 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 MULTY

*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  $SWL = F_n(PORO)$ . 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 [TEMP I](#).

## 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:

Keyword	Description
<a href="#">DIFFMMF</a>	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 <a href="#">DIFFMMFS</a> .
<a href="#">MULSGGD</a> <a href="#">MULSGGDV</a>	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 <a href="#">MULSGGDS</a> .

Keyword	Description
MULTSIG MULTSIGV	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 .
MULTXZ MULTY MULTZ	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.
PERMX PERMY PERMZ	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 .
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 SIGMAV	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.
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, MULSGGDS, 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 /
/
```

### 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

ECLIPSE 100

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 PVTNUM) 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, ADDRREG, EQUALREG and COPYREG.

ECLIPSE 100

**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.

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 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.

ECLIPSE 300

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 /
```

## OPTCONS

**Specify optimization constraints**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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.  
DEFAULT: '<' or 'LE'
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

```
OPTCONS
-- Constrain gas production from well P1,
-- and water injection to well INJE2
'WGPR' P1 '<' 2000.0 /
'WWIR' INJE1 '>' 500.0 /
/
```

## OPTDIMS

## Specify optimization dimensions

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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'.

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/  
/
```

## OPTIONS

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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  $NEWTIT \leq n$ ).

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 4<sup>th</sup> 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 `WVFPEXP`.

If <0 Causes immediate shut-in if flow is negative in any well iteration.

10. If >0 chops on P, R<sub>s</sub> and R<sub>v</sub> 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 uncompactd pore volumes).

The default average field pressure is

$$FPR = \frac{\Sigma HCPV \cdot p}{\Sigma HCPV}$$

where  $HCPV = PV(1-S_w)$  is the hydrocarbon pore volume and  $p$  is the oil phase pressure. Setting `OPTIONS (31)>0` implies

$$FPR = \frac{\Sigma PV \cdot p}{\Sigma PV}$$

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  $R_s$  and  $R_v$  that are defined, for example, by

$$FRS = \frac{\Sigma PV \cdot R_s \cdot S_o}{\Sigma PV \cdot S_o}$$

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)

$$FPPO = \frac{\Sigma \Phi_o \cdot S_o \cdot PV}{\Sigma S_o \cdot PV}$$

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

$$FPPO = \frac{\Sigma\Phi_o \cdot PV}{\Sigma PV}$$

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 = 0 defaults to 1.
- If = 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 set = 0, this defaults to max (4, NETBAL/2) where NETBAL is the maximum number of network balancing iterations allowed (see the [NETBALAN](#) keyword in the Network option).
34. If set >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<sup>st</sup> pass, if they are not actually needed to meet the DCQ x profile target.
35. If set > 0 ECLIPSE will revert to the pre-98A method of calculating the explicit wellbore head (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 set >0 with the nine-point scheme active, matrix preconditioning is suppressed (for development testing).
37. If set >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 set >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.

---

**Note:** The use of this switch is not recommended in general.

---

39. If set >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 set >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](#)):

$$B_o \text{ (used)} = \frac{(B_o \text{ cell} + [ | \text{NWT} | - 1 ] B_o \text{ (previous)})}{| \text{NWT} |}$$

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 **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 **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 **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 **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 **MXWSIT** in the **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 **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 **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 **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:

$$H(\text{used}) = (H(\text{current}) + nH(\text{previous})) / (n + 1)$$

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 DUMPFLUX 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  $\cos \theta$ , instead of being multiplied by  $(\cos \theta)^{0.5}(1 + \sin \theta)^2$ .
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 `WEL SPECS`). 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  $RsSep = 0.01 * OPTIONS(100)$ . The minimum separation criterion is then calculated as  $((Rs2-Rs1)/Rs1) \geq RsSep$ , 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 [GCONSALE](#) 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  $-OPTIONS(103)$  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  $\text{THETA}=4$ .
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, WTIC and CTIC 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 adsorbed 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 [DPKRMOD](#) 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 [EPSDBGS](#) 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 [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, 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 `PVTNUM` 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 [WCONPROD](#) and [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 [WCONPROD](#) or [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 [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 [MESSAGES](#) keyword or [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 [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.

212. If set to 1, connate gas will be included in the end-point scaling calculation of the saturation at which [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 [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 [GRAVDR](#) model when used with either the [SURFACT](#) or [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 [WELSPECL](#) on an LGR that is switched off is downgraded to a warning.
217. If set to 1, the pre-2014.1 behavior for [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 [VDFLOW](#) or [VDFLOWR](#) keyword. Prior to 2014.1 this was **only** applied to the phase mobility calculations for the well connections if the [FHERCHBL](#) keyword had also been specified.
- For the 2014.1 version, by default, the non-Darcy flow component due to the [VDFLOW](#) or [VDFLOWR](#) keyword is not applied to the well connections in any circumstance. If the [VDFLOW](#) or [VDFLOWR](#) flow component is required to be applied to the well connections, this may be enabled by setting this [OPTIONS](#) item to 2. 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) will supersede those due to the [VDFLOW](#) or [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 [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 [MINNNCT](#) 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 [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 **LWSWL**
  - 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 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.

```
.....
OPTIONS
20* 2 12* 1 /
.....
```

or

```
.....
OPTIONS
20* 2 /
OPTIONS
33* 1 /
.....
```

## OPTIONS3 **Activates special program options**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

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 equilibration option 1 (10<sup>th</sup> 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

- =1 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.
- =2 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.
- =3 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.
- =4 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.
- =5 the simulator reverts to the pre-2000A treatment of a well's bottom hole pressure reference depth that is defaulted in [WELLSPEC](#), [WELSPECS](#) or [WELSPECL](#). 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.
- =6 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 [WELLSPEC](#) keyword) being to shut the well.
- =7 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.
- =8 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.
- =9 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 [GCONTROL](#)).
- =10 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 end-points 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  $\geq 1$ , then in black oil mode, the oil formation volume factor and the viscosities are interpolated in  $1/b_o$  and  $1/(b_o.V_o)$  as in ECLIPSE.
- If =2, and using `PVDG`, the gas formation volume factor and viscosities are also interpolated in  $1/b_g$  and  $1/(b_g.V_g)$ . This option has no effect on `PVTG`, where the formation volume factor is interpolated in  $1/b_g$  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  $\sqrt{\text{target}/\text{tte}}$  rather than  $\text{target}/\text{tte}$ .
  - <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  $n^{\text{th}}$  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 (0.01psia).
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\_1 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

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 [PCG](#) 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 behavior of the [ACTION](#) ([ACTIONG](#), [ACTIONR](#), [ACTIONW](#), [ACTIONS](#), [ACTIONX](#)) keywords is restored. Specifically, the [ACTION](#) ([ACTIONR](#), [ACTIONX](#)) conditions are tested before the region summary quantities such as [ROIIP](#) are updated, and before the summary quantities are output. As a result summary data from the previous timestep can be overwritten when an [ACTION](#) ([ACTIONG](#), [ACTIONR](#), [ACTIONW](#), [ACTIONS](#), [ACTIONX](#)) is triggered. In addition the maximum timestep length after a well modification is not imposed, see item 11 of [TSCRIT](#) keyword, and item 10, record 1 of the [TUNING](#) keyword. If the user defined quantity feature is active then these quantities will also be calculated before the region summary quantities are updated.
  - 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, AI1, BI2). If the 'REGONLY' option is used then the tracer AI1 refers to region A and injected component 1, BI2 to region B and injected component 2, and so on. For example, mnemonic FTTLBI2 can be used in the summary section to monitor FTTL for the injected tracer BI2.

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 H<sub>2</sub>O-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 "H<sub>2</sub>O" 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 [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 [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 [RPTRST](#), [RPTSCHED](#) and [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 [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 [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 [WINJORD](#) or [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 [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 [WINJORD](#) or [WINJMIX](#).
  - If non-zero and divisible by 11 the pre-2008.1 behavior of [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 [WINJORD](#) or [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, FPRH and FPPO, 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 WAVAILIM.

- 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 GEFAC 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 [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](#).
  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.
  - If divisible by 2, streamline update and retrace times are calculated as they were for 2009.1.
  - If divisible by 3, streamline tracing uses the 2009.1 cell porosity calculation.
  - If divisible by 5, streamline tracing ignores rock compressibility.
  - 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  $(x_i * m_i)$  where  $x_i$  is the liquid fraction and  $m_i$  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, the pre-2010.1 default for the WELSEGS top segment effective wellbore volume is used.
235. 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.
236. 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.
237. 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.
238. If set to a non-zero value, the pre-2009.2 method for interpreting group injection inheritance rules will be restored. From 2009.2 onwards, the group injection inheritance linkage is automatically reevaluated when the injection fluid changes to ensure that the correct fluid properties are used.
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 [FLOCn](#), [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 [VELDEP](#) 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, KRORG 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, KRORG or KROWG end-point scaling arrays in the INIT file when [ENDSCALE](#) and [FILLEPS](#) 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](#), [KRORW](#) and [KRORG](#) 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](#), [KRORW](#) and [KRORG](#) (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`, `HYPC`, `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.
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**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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:

Extension	Unified	Formatted
FOPT	Yes	Yes
OPT	Yes	No
K000#	No	Yes
J000#	No	No

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 /
```

## OPTODD3P Specify ODD3P relative permeability and capillary pressure hysteresis model options

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## OPTOPTS

## Specify optimization options

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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  $\underline{\Psi}$ .

In the **alternative** formulation, as described in [Introducing Additional Constraints](#) in the *ECLIPSE Technical Description* is used to eliminate  $\underline{\Phi}$ .

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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 [WSEGVAVL](#) keyword). The segment cross sectional area parameter is currently only available for producing multisegment wells, not injectors.

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 `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

```

--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

```

-- 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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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 /
```

## ORTHERRO **Set a minimum orthogonality error for calculation of nine-point transmissibilities**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
ORTHERRO
0.001 /
```

## ORTHERRX **Minimum orthogonality error for X-interface nine-point transmissibility calculation**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## OUTRAD

## Outer radius for radial geometry

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $i^{\text{th}}$  and  $i-1^{\text{th}}$  grid blocks) will be related by

$$\frac{R_i}{R_{i-1}} = \left( \frac{\text{OUTRAD}}{R_{i_L-1}} \right)^{1/(\text{NR}-(i_L-1))} \quad \text{Eq. 3.137}$$

so that neighboring pairs of radii have the same ratio for all  $i > i_L-1$ .  $DR$  for the  $i^{\text{th}}$  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 /

```

## 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.

---

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

## OVERBDI

## Initial overburden boundary conditions

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

```
--Set a 3000 psi overburden
OVERBDI
-3000 /
/
```

## OVERBDIL Initial overburden boundary conditions in an LGR

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- For the local grid refinement LGR1 set a 3000 psi overburden
OVERBDIL
'LGR1' -3000 /
/

```

## OVERBURD **Rock overburden pressure tables**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The depth.  
Values should increase monotonically down the column.  
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
- 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 ( $P_{\text{eff fl}} = P_{\text{fluid}} - \text{Overburden}$ ). 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,

$$P_{\text{eff ov}} = \text{Overburden} - P_{\text{fluid}}, \text{ which is } - P_{\text{eff fl}}.$$

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
/
```

## OVERTR

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  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.

---

## OVERTX

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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:

```

-- 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.

## OVERTY

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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: cP.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 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.

---

## OVERTZ

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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:

```

-- 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.

## **P**

This section contains an alphabetic listing of keywords starting with the letter P.

## PARACHOR **Component parachors**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)<sup>1/4</sup>cc/gm-M (**METRIC**), (dynes/cm)<sup>1/4</sup>cc/gm-M (**FIELD**), (dynes/cm)<sup>1/4</sup>cc/gm-M (**LAB**), (dynes/cm)<sup>1/4</sup>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 /
```

## PARALLEL **Select a parallel run**

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

DEFAULT: 0

9. Reserved.

## Example

```
PARAOPTS  
2* 10 3 /
```

## PARTTRAC Dimensions for the partitioned tracer option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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/modell1.dat' /
/

```

## PBUB

## Initial bubble point pressure

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## PBVD

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 ( $P_{\text{bub}}$ ).

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  $\geq$ 3:

```
PBVD
7000 4000
8000 4020
9000 4045 /
```

## PCG, IPCG Scaled maximum gas capillary pressures

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$P_c = P_{ct} \left( \frac{PCG}{P_{cm}} \right) \quad \text{Eq. 3.138}$$

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.

ECLIPSE 300

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):

```
PCG  
6*10.0  
6*12.0  
6*15.0 /
```

# PCG32D

## 2D table of gas/oil capillary pressure

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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-----
--SWAT  0.01  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$H(S_g) = \frac{\delta_g S_g^{\alpha_g}}{S_g^{\alpha_g} + (1 - S_g)^{\beta_g}}$$

and

$$F(S_w) = \frac{\delta_w S_w^{\alpha_w}}{S_w^{\alpha_w} + (1 - S_w)^{\beta_w}}$$

The *representative* capillary pressures, that is, the values obtained from table look-up using the normalized saturations are corrected according to the following equations:

$$P_{cow} = \hat{P}_{cow} + HR$$

$$P_{cgo} = \hat{P}_{cgo} + FR$$

$$R = \hat{P}_{cgo} - \hat{P}_{cgo} - \hat{P}_{cow}$$

where the capillary pressure terms  $\hat{P}_{cij}$  denotes the representative capillary pressures obtained from table look-up as a function of normalized hysteresis saturation  $\tilde{S}_i$  or  $\tilde{S}_j$ . The capillary pressures may be made functions of one of two normalized saturations as shown below:

Saturation Dependence	Oil-Water System	Gas-Oil System	Gas-Water System
1	$\hat{P}_{cow}(\tilde{S}_o)$	$\hat{P}_{cgo}(\tilde{S}_g)$	$\hat{P}_{cgo}(\tilde{S}_g)$
2	$\hat{P}_{cow}(\tilde{S}_w)$	$\hat{P}_{cgo}(\tilde{S}_o)$	$\hat{P}_{cgo}(\tilde{S}_w)$

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:

Relationship type	Calculation of oil-water and gas-oil capillary pressure	
0	$P_{cow} = 0.0$	$P_{cgo} = 0.0$
1 and 2	$P_{cow} = \hat{P}_{cow}(\tilde{S}_o) + HR$	$P_{cgo} = \hat{P}_{cgo}(\tilde{S}_g) + FR$
3 and 4	$P_{cow} = \hat{P}_{cow}(\tilde{S}_o) + HR$	$P_{cgo} = \hat{P}_{cgo}(\tilde{S}_o) + FR$
5 and 6	$P_{cow} = \hat{P}_{cow}(\tilde{S}_w) + HR$	$P_{cgo} = \hat{P}_{cgo}(\tilde{S}_g) + FR$
7 and 8	$P_{cow} = \hat{P}_{cow}(\tilde{S}_w) + HR$	$P_{cgo} = \hat{P}_{cgo}(\tilde{S}_o) + FR$

Relationship type	Calculation of oil-water and gas-oil capillary pressure
9	$P_{cow} = \hat{P}_{cow}(\tilde{S}_o)$ <span style="float: right;"><math>P_{cgo} = \hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cow}(\tilde{S}_o)</math></span>
10	$P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}_{cgo}(\tilde{S}_w) + \frac{S_o}{S_g + S_o} \hat{P}_{cow}(\tilde{S}_w)$ $P_{cgo} = \frac{S_w}{S_o + S_w} \hat{P}_{cgo}(\tilde{S}_g) + \frac{S_o}{S_o + S_w} \hat{P}_{cgo}(\tilde{S}_g)$
11	$P_{cow} = \frac{S_g}{S_g + S_o} \hat{P}_{cgo}(\tilde{S}_w) + \frac{S_o}{S_g + S_o} \hat{P}_{cow}(\tilde{S}_w) \quad P_{cgo} = \hat{P}_{cgo}(\tilde{S}_g)$
12	$P_{cow} = \frac{S_g(S_o[\hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_g)] + S_w[\hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_g)]) + S_o(S_o + S_w)\hat{P}_{cow}(\tilde{S}_w)}{S_o(S_g + S_o + S_w)}$ $P_{cgo} = \left[ \frac{S_o}{(S_o + S_w)} + \frac{S_g S_w}{(S_o + S_w)(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgo}(\tilde{S}_g) +$ $\left[ \frac{S_w}{(S_o + S_w)} + \frac{S_g S_w^2}{S_o(S_o + S_w)(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgo}(\tilde{S}_g) -$ $\left[ \frac{S_g S_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cgo}(\tilde{S}_w) - \left[ \frac{S_w}{(S_g + S_o + S_w)} \right] \cdot \hat{P}_{cow}(\tilde{S}_w)$

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:

Relationship type	Calculation of value R
1	$R = \hat{P}(\tilde{S}_g) - \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}_{cow}(\tilde{S}_o)$
2	$R = \hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}_{cow}(\tilde{S}_o)$
3	$R = \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}_{cgo}(\tilde{S}_o) - \hat{P}_{cow}(\tilde{S}_o)$
4	$R = \hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_o) - \hat{P}_{cow}(\tilde{S}_o)$
5	$R = \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}(\tilde{S}_g) - \hat{P}_{cow}(\tilde{S}_w)$
6	$R = \hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}_{cow}(\tilde{S}_w)$
7	$R = \hat{P}_{cgo}(\tilde{S}_g) - \hat{P}_{cgo}(\tilde{S}_o) - \hat{P}_{cow}(\tilde{S}_w)$
8	$R = \hat{P}_{cgo}(\tilde{S}_w) - \hat{P}_{cgo}(\tilde{S}_o) - \hat{P}_{cow}(\tilde{S}_w)$

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`

```
PCODD3P
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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## PCRIT

## 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), atma (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 /
```

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## PCREDIT

**Critical pressures for detailed components**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [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 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:

```
PCREDIT
1050 1300 1500 1803 /
```

## PCRITS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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*.

## PCRITSDE

## Critical pressures for surface EoS for detailed components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$P_c = P_{ct} \left( \frac{PCW}{P_{cm}} \right) \quad \text{Eq. 3.139}$$

where

$P_{ct}$  Capillary pressure from the table

$P_{cm}$  Maximum  $P_c$  in the table - at  $S_w = S_{wco}$ , connate water

PCW Maximum  $P_c$  from the PCW data.

The PCW 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 PCW 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 PCW data scales the drainage curve only. The imbibition curve can be scaled by using the equivalent IPCW keyword.

---

**Note:** The PCW 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 PCW 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.

---

ECLIPSE 300

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 PCW 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**:

```

PCW32D
-----SOIL-----
--SGAS
0.01  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
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 /
/
    
```

## PDEW

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## PDIVX

## Select a division of the x-direction for parallel runs

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_{px}$  integer items (where  $N_{px}$  is set using the [NPROCX](#) keyword). Each integer specifies the last x-index for the particular processor. The last item must have the value  $N_x$ .

See also "[Parallel option](#)" in the *ECLIPSE Technical Description*.

### Example

```

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 /

```

## PDIVY

## Select a division of the Y-direction for parallel runs

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_{py}$  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 /

```

## PDVD

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 ( $P_{\text{dew}}$ ).

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≥3:

```
PDVD
7000 3000
8000 3020
9000 3045 /
```

## PEBI

## Specifies a Pebi/Petra grid is to be used

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 Pebi grid that includes [DUALPORO](#) or [DUALPERM](#).

## Example

```
PEBI
NO YES /
```

## PECOEFS

## Coefficients for petro-elastic model

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (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:

$$K_{minerals} = C_1 \phi_{eff} + C_0 \quad \text{Eq. 3.140}$$

UNITS: barsa (METRIC), psia (FIELD), atma (LAB)

5. The zero<sup>th</sup> 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.

$$K_{frame}^{\alpha} = C_K K_{minerals}^{\alpha} + \sum_{i=0}^{\alpha} C_{K,i}(P) \phi_{eff}^i \quad \text{Eq. 3.141}$$

where  $\alpha$  is the **item 9** and the  $C_{K,i}(P)$  tables are specified using the keyword **PEKTABx**.

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.

$$G_{frame}^{\alpha} = C_G G_{minerals}^{\alpha} + \sum_{i=0}^7 C_{G,i}(P) \phi_{eff}^i \quad \text{Eq. 3.142}$$

where  $\alpha$  is defined in the previous item,  $G_{minerals}$  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,  $G_{minerals}$ , in the region.

UNITS: barsa (METRIC), psia (FIELD), atma (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  $e$  term in equation 3.143.

$$K_{fluid} = (K_{liquid} - K_g)(1 - S_g)^e + K_g \quad \text{Eq. 3.143}$$

where

$K_{fluid}$  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

$K_{liquid}$  is given by equation 3.144.

Note that this item will be ignored if item 11 is 0.

$$\frac{S_o + S_w}{K_{liquid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} \quad \text{Eq. 3.144}$$

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,

$$\frac{1}{K_{fluid}} = \frac{S_o}{K_o} + \frac{S_w}{K_w} + \frac{S_g}{K_g} \quad \text{Eq. 3.145}$$

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
PEDERSEN
```

## PEDIMS

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$M_{w,mix} = c_1 \cdot 1.304 \times 10^{-4} (\bar{M}_w^{c_2 \cdot 2.303} - \bar{M}_n^{c_2 \cdot 2.303}) + \bar{M}_n$$

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:

$$\alpha_{mix} = 1 + c_5 \cdot (\rho_{r,mix})^{c_3} \cdot (M_{w,mix})^{c_4}$$

$$\alpha_o = 1 + c_6 \cdot (\rho_{r,mix})^{c_3}$$

where

$\rho_{r,mix}$  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**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$M_{w,mix} = c_1 \cdot 1.304 \times 10^{-4} (\bar{M}_w^{c_2 \cdot 2.303} - \bar{M}_n^{c_2 \cdot 2.303}) + \bar{M}_n$$

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 by this keyword.

For the Pedersen viscosity correlation, the rotational coupling factors are given by:

$$\alpha_{mix} = 1 + c_5 \cdot (\rho_{r,mix})^{c_3} \cdot (M_{w,mix})^{c_4}$$

$$\alpha_o = 1 + c_6 \cdot (\rho_{r,mix})^{c_3}$$

where

$\rho_{r,mix}$  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 [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 /
```

## PEGTABx

## Pressure table for shear modulus of frame

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>a</sup> (METRIC), psia<sup>a</sup> (FIELD), atma<sup>a</sup> (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.

## PEKTABx

## Pressure table for bulk modulus of frame

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

These keywords specify the tables which contain the coefficients of a polynomial in effective porosity  $\varphi_{eff}$  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,  $K_{frame}$ , is

$$K_{frame}^{\alpha} = C_K K_{mineral}^{\alpha} + \sum_{i=0}^7 C_{K,i}(P) \varphi_{eff}^i \quad \text{Eq. 3.146}$$

where

$K_{mineral}$  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,

$\varphi_{eff}$  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<sup>a</sup> (METRIC), psia<sup>a</sup> (FIELD), atma<sup>a</sup> (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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

```
PERMAVE
1* 0.0 1.0 /
```

## PERMMF

**Permeability for matrix-fracture coupling**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

## PERMR

## Specifies radial permeability values

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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             $\sigma_{1,1}$     $\sigma_{1,2}$    ...    $\sigma_{1,NGMPTC}$

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             $\sigma_{3,1}$     $PM_{1,1}$    ...    $PM_{1,NGMPTC}$

Record 3             $\sigma_{3,2}$     $PM_{2,1}$    ...    $PM_{2,NGMPTC}$

... ..

... ..

Record NGMPTR    $\sigma_{3,NGMPTR}$     $PM_{NGMPTR,1}$    ...    $PM_{NGMPTR,NGMPTC}$   
+1

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:

```
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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

ECLIPSE 100

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The PERMXY, PERMYZ, PERMZ, PERMXX, PERMY 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  $K_{31} = K_{13} = \text{PERMZ}$ .

The diagonal terms are aliased: PERMXX to [PERMX](#), PERMY 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

ECLIPSE 100

## 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.

Every PERMZ 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 /
PERMZ
72*0 /

```

## PETGRID

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

```
PETGRID  
MODEL1.GSG /
```

## PETOPTS

## Options for Petrel runs

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

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

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

$$\beta_{\text{used}} = \text{PPDAMP}\beta_{\text{calculated}} + (1-\text{PPDAMP})\beta_{\text{previous}}$$

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.

$$p_L = (\text{PPBFAC}) \times p_{\text{completion cell}}$$

ECLIPSE 100

ECLIPSE 100

ECLIPSE 300

ECLIPSE 300 only

ECLIPSE 300 only

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, 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, 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, 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, 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 /
```

## PIMTDIMS

## PI scaling table dimension data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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  $\geq 5$ :

```
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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- Pinchout threshold thickness.  
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)  
DEFAULT: 0.001 (all unit sets)
- 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).

---

- 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)

- 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

- 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 MULTZ values for the active cell at the top of the pinchout and all the inactive cells in the pinched-out column. This replaces OPTIONS keyword switch 17. In ECLIPSE 300, this option is only available for Cartesian corner-point geometry.

DEFAULT: TOP

---

**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 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 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 PINCH has been specified and if keyword MINPV or 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 MINPV or 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 NOGAP 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 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 (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 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 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 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 PINCHXY.

## 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; there is no connection across active cells.
- The default method (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 OLDTRAN (see

"Transmissibility Calculations" in the *ECLIPSE Technical Description*) the pinchout transmissibility between top and bottom cells  $t$  and  $b$  is

$$Tr = \frac{1}{\frac{1}{Tr_t} + \frac{1}{Tr_b}} MULTZ_t$$

where

$$Tr_t = CDARCY \frac{2PERMZ_t DX_t DY_t}{DZ_t}$$

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 `MULTZ` between all cells in the pinched-out column (that is  $\min(MULTZ_i)$  for  $i = t, \dots, b-1$ ).

If the flag `ALL` is specified in `item 4`, then the pinchout transmissibility is calculated as a harmonic average

$$Tr = \frac{n}{\sum \frac{1}{TRANZ_i}}$$

where

the sum is over all cells  $i = t, \dots, i = b-1$  in the column between  $t$  and  $b$ ,

$n$  is the number of cells being summed over, and

each transmissibility  $TRANZ_i$  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' /
```

## PINCHNUM Identifies pinchout regions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, `NRPIN` 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 `NRPIN`=4:

```
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
/
```

## PINCHOUT

**Generates connections across pinched-out layers**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

- Pinchout threshold thickness.
 

UNITS: m ([METRIC](#)), ft ([FIELD](#)), cm ([LAB](#)), m ([PVT-M](#))

DEFAULT: 0.001 (all unit sets)
- 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**
- 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)
- 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**
- 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.

ALL 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

```
PINCHREG
--Threshold  Gap or   Empty
--thickness  No Gap   Gap
             0.1    1*    10.0  /
             0.2    1*    20.0  /
             1.0    'NOGAP' /
```

## PINCHXY

## Generates horizontal pinchout connections

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
- Only horizontal pinchouts in the global grid will be generated. Currently, no local grid PINCHXY connections will be calculated.

*ECLIPSE 300 only*

### Example

```
PINCHXY
1* 0.1 /
```

## PLMIXNUM Polymer mixing regions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## PLMIXPAR Polymer Todd-Longstaff mixing parameter

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```

-- RUNSPEC section:
REGDIMS
 9* 5 /
--PROPS section:
PLMIXPAR
0.35 /
0.57 /
0.49 /
0.8 /
1.0 /

```

## PLYADS

## Polymer adsorption functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (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:

```

PLYADS
.0000 .00000
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0010 .00010 /

```

## PLYADSS

## Polymer adsorption functions varying with salinity

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> ([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:

```
PLYADSS
.0000 0.0
0.0
```

```
0.0
0.0 /
.0100 0.0002
0.0003
0.0004
0.0004/
.0200 0.0005
0.0006
0.0007
0.0008/
.0300 0.0008
0.0007
0.00065
0.0006/
/
```

## PLYATEMP Polymer adsorption table temperatures

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

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 .0010 /

```

## PLYCAMAX Specifies maximum polymer adsorption values for the Polymer Flood Model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## PLYDHFLF Polymer thermal degradation half life

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- The local temperature in the solution.  
The values should increase monotonically down the column.  
UNITS: °C (METRIC), °F (FIELD), °C (LAB)
- 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 /
```

## PLYESAL

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

$$C_{SEP} = C_- + (\alpha^P - 1)C_{++} \quad \text{Eq. 3.147}$$

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 (/).

UNITS: dimensionless

DEFAULT: 0.0

### Example

For NTSFUN=2:

```
PLYESAL
-- alpha
  10 /
  15 /
```

## PLYKRRF

## Specifies residual resistance factors for the Polymer Flood Model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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 /
```

## PLYOPTS

## 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

```
PLYOPTS
2.0e-6 /
```

	ECLIPSE 100
X	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## PLYRMDEN Specifies mass densities for the Polymer Flood Model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/rm<sup>3</sup> (METRIC), lb/rb (FIELD), gm/rcc (LAB), kg/rm<sup>3</sup> (PVT-M)

See also keywords [PLYKRRF](#) and [PLYCAMAX](#).

### Example

```

----- IX1-IX2 JY1-JY2 KZ1-KZ2
BOX
      1   4   1   4   1   1   /
PLYRMDEN
8*1880 8*2010 /

```

## PLYROCK Specifies the polymer-rock properties

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- 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.
- 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.
- 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<sup>3</sup> (METRIC), lb/rb (FIELD), gm/rcc (LAB), kg/rm<sup>3</sup> (PVT-M)

- 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
- 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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).

- 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](#).
- 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
- 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
- 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](#).
- 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 /
```

## PLYSHEAR Polymer shear thinning/thickening data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$\mu_{w,sh} = \mu_{w,eff} Z_w^*, \quad \mu_{p,sh} = \mu_{p,eff} Z_p^*$$

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,  $\frac{d(\log(M))}{d(\log(v))} < -1$ . 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
.....

```

1.0	0.8
3.0	0.75
6.0	0.7
10.0	0.68 /

## PLYSHLOG Polymer shear thinning/thickening logarithmic model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- The reference polymer concentration that is associated with the shear data provided.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)  
Note that this item cannot be defaulted and must take a positive value.
- The reference salinity, if appropriate, that is associated with the shear data provided.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)  
Note that this item must be specified if either of the [PLYVISCS](#) or [PLYVSCST](#) keywords are used, and should otherwise remain defaulted.
- 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.

- 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).
- 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, PLYVISCs, 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

$$\frac{d(\ln M^{ref})}{d \ln X} = \frac{\ln M_{i+1}^{ref} - \ln M_i^{ref}}{\ln X_{i+1} - \ln X_i} > -1$$

## 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≥6 specified in keyword [TABDIMS](#) and keyword PLYVSCST present:

```

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
/

```

## PLYTRRF

## Polymer residual resistance factor temperature dependence

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## PLYVISC

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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:

- The polymer concentration in the solution.  
The values should increase monotonically down the column.  
The first entry should be 0.0  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (PVT-M)
- 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.

*ECLIPSE 100 only*

*ECLIPSE 100 only*

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 PLYVISC should be used as Multi-Component Brine model cannot currently be used with salt-sensitive Polymer model.

*ECLIPSE 100 only*

---

**Note:** If the surfactant option is active, the viscosity multiplier applies to the water viscosity supplied in the [SURFVISC](#) 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:

```
PLYVISC
.0000  1.0
.0003 10.0
.0005 25.0
.0008 50.0
.0010 80.0 /
```

## PLYVISCS

## Polymer/salt solution viscosity function

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/sm<sup>3</sup> ([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 is 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 /

```

```
.0300 50.0  
      30.0  
      20.0  
      12.0 /  
/
```

## PLYVISCT

## Polymer/temperature viscosity function

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/sm<sup>3</sup> ([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
.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 /
.0300  50.0
        30.0
        20.0
        12.0 /
/

```

# PLYVSCST Polymer/salt solution/temperature viscosity function

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

Column 1	Column 2
Polymer concentration value 1	multiplier at polymer concentration 1, brine concentration 1, and temperature 1
multiplier at polymer concentration 1, brine concentration 1, and temperature 2	
multiplier at polymer concentration 1, brine concentration 1, and temperature 3 /	<i>end of first record set (brine concentration 1) for polymer concentration 1</i>
multiplier at polymer concentration 1, brine concentration 2, and temperature 1	
multiplier at polymer concentration 1, brine concentration 2, and temperature 2	
multiplier at polymer concentration 1, brine concentration 2, and temperature 3 /	<i>end of 2nd record set (brine concentration 2) for polymer concentration 1</i>
/	<i>end of records for polymer concentration 1</i>
Polymer concentration value 2	multiplier at polymer concentration 2, brine concentration 1, and temperature 1
<i>Record sets repeat for remaining polymer concentration values</i>	
<i>/ one additional slash to terminate this table</i>	
<b>remaining tables (if any) have the same format</b>	

Table 3.28: PLYCSCST 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/sm<sup>3</sup> (`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
        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

```

## PMANUM

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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  $P_{\text{pub}}$  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](#).

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  $R_s$  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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
PMISC
1000.0 0.0
4000.0 0.0
4100.0 1.0
8000.0 1.0 /
```

## POISSONR Poisson's ratio for rock stress balance

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

# PORO

## Specifies the grid block porosity values

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

ECLIPSE 300 only

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
0.18 0.15 0.12 0.1 0.08 0.09 0.1 0.11 0.12 0.13 0.15 0.15
0.18 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{rm}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{rm}^3$  (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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
PPCWMAX
 15.0      YES    /
 17.0      YES    /
```

## PRCORR

## Request modified Peng-Robinson EoS

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## PREF

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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, `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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.148}$$

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 this keyword,

$T_{ref_c}$  is the reference temperature, specified with keyword [TREF](#), and

$c_{T1_c}$  is the thermal expansion coefficient, specified with keyword [THERMEX1](#).

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 [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

```
PREF
1527.4
1527.4 /
```

## PREFS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## PREFW

## Reference pressures for water components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\rho_w = \rho_{ref_w} \cdot (1 + X + X^2/2)$$

$$X = c_{P_w} \cdot (P - P_{ref_w})$$

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:

$$\mu_w = \mu_{ref_w} \cdot (1 + Y + Y^2/2)$$

$$Y = -c_{V_w} \cdot (P - P_{ref_w})$$

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 /
```

## PREFWS

## Reference pressures of water components for surface EoS

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), atma (LAB), atma (PVT-M)

## PRESSURE Initial pressures

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## PRIORITY

## Sets coefficients for well prioritization option

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

ECLIPSE 100

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 [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 [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 [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 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 NUPCOL is set by keyword [NUPCOL](#).

While a well has its flow rate cut back to meet a group's limit, its minimum rate limits set in keywords [WECON](#) and [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 *ECLIPSE Technical Description*) at regular intervals, using the general formula

$$\text{Priority} = \frac{A + BQ_o + CQ_w + DQ_g}{E + FQ_o + GQ_w + HQ_g} \quad \text{Eq. 3.149}$$

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 [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 `GCONPRI`, not `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 `GCONPRI`, while groups using the guide rate method have their rate limits set with keyword `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 `GCONPROD` group targets. If a prioritization group is subordinate to one with a `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 `GCONPROD` group's target. Wells from the drilling queue (keyword `QDRILL` or `WDRILPRI`) that are not subordinate to a prioritization group will be opened when necessary to meet the `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 `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 (`METRIC`), day (`FIELD`), hr (`LAB`), day (`PVT-M`)

Items 2 - The coefficients A - H in the first priority formula.

9

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

*ECLIPSE 100 only*

Items The coefficients A - H in the second priority formula.

10-17

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 0.0 1.0 /
```

ECLIPSE 100

## PROPFLOW **Proppant flowback in a fracture**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 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 `WLIFT` a default, zero or negative value.

- The THP action has no effect within groups that are part of a network (see "[Network option](#)" in the *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 *ECLIPSE Technical Description*), by default all the requested production rule actions are attempted **before** gas lift is allocated. But if `LIFT` is entered in Record 1, any production rule actions entered after `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 `LIFTOPT` keyword, and is not affected by the presence or absence of the `LIFT` mnemonic in `PRORDER`.
- If the `PRORDER` keyword is not present then, for back compatibility, only the `DRILL` action will be applied (if a drilling queue has been set up), followed by the `COMP` action (in an Extended Network Model containing automatic compressors).
- If compressors have been defined in the extended network model with the `NETCOMPA` keyword (see "[The extended network model](#)" in the *ECLIPSE Technical Description*), they are used regardless of whether the `PRORDER` keyword is entered or if the `COMP` option is missing from the `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 `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 `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 `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 iterations have converged, by using the `WAITBAL` keyword (see "[Network option](#)" in the *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

```
PRORDER
DRILL THP REPERF /
NO /
```

causes the following actions to take place whenever a group cannot make its `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.

## PRVD

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## PSEUPRES

**Activate the generalized pseudo-pressure option in ECLIPSE 300**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 SCHEDULE section before any well operations keywords or timestepping keywords are specified. Alternatively this option can be activated for individual wells by entering GPP in item 8 of [WELSPECS](#).

For details of the GPP model, see "[Special inflow equations](#)" in the *ECLIPSE Technical Description*.

See also keyword [PICOND](#).

## PSGRO

## Primary residual gas saturation in gas-oil system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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`, `NDIVIY=4` and `NDIVIZ=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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

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
/
--

```

## PSPLITY

## Specifies divisions of the y-direction for parallel runs

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $N_{py}$  integer items (where  $N_{py}$  is the number of processors in 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  $N_{py}$ .

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
/
--

```

## PSSTA

### Saturation pressure state assignment

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

$$\text{MAXABS} \quad \frac{\text{Max}}{i} \frac{|Q_i^{n+1} - Q_i^n|}{S \cdot \Delta t}$$

$$\text{L2} \quad \frac{\sqrt{\sum_i (PV(Q_i^{n+1} - Q_i^n))^2}}{\Delta t \sum_i PVQ_i^{n+1}}$$

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  $1.0 \times 10^{-3}$  for the saturations and brine concentration and  $1.0 \times 10^{-2}$  for pressure. If the norm is L2 then the defaults are all  $1.0 \times 10^{-6}$ .

## 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 [AQUCHWAT](#), [AQUCHGAS](#), [AQUFETP](#) and [AQUCT](#), 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 /
```

## PSTNUM

## Primary saturation function region numbers

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## PSTRBC

## Principal stress boundary conditions coupled stress and fluid flow

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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
9. The Y component of the third principal stress vector.  
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:
-- P1=(-2000,0,0), P2=(0,-2500,200), P3=(0,0,0)
-- are defined on all boundary faces of the entire grid.

PSTRBC
-- side   P1X   P1Y P1Z   P2X   P2Y   P2Z   P3X P3Y P3Z
-- 'I-'   -2000  0  0     0   -2500  200   9* /
-- 'I+'   -2000  0  0     0   -2500  200   9* /
-- 'J-'   -2000  0  0     0   -2500  200   9* /
-- 'J+'   -2000  0  0     0   -2500  200   9* /
-- 'K-'   -2000  0  0     0   -2500  200   9* /
-- 'K+'   -2000  0  0     0   -2500  200   9* /
/
```

## PSTRBCL

## Principal stress boundary conditions coupled stress and fluid flow in a Local Grid Refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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)

- 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
  9. The Y component of the third principal stress vector.  
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:
-- P1=(-2000,0,0), P2=(0,-2500,200), P3=(0,0,0)
-- are defined on all boundary faces of the local grid refinement LG1.

PSTRBCL
-- side      LGR      P1X      P1Y      P1Z      P2X      P2Y      P2Z      P3X P3Y P3Z
`I-`        `LG1`    -2000    0    0    0    -2500    200    9* /
`I+`        `LG1`    -2000    0    0    0    -2500    200    9* /
`J-`        `LG1`    -2000    0    0    0    -2500    200    9* /
`J+`        `LG1`    -2000    0    0    0    -2500    200    9* /
`K-`        `LG1`    -2000    0    0    0    -2500    200    9* /
`K+`        `LG1`    -2000    0    0    0    -2500    200    9* /
/
```

## PSWRG

## Primary residual water saturation in water-gas system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/
```

## PSWRO

## Primary residual water saturation in water-oil system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The reference pressure ( $P_{\text{ref}}$ ) for items 2 and 4.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
- The oil formation volume factor at the reference pressure,  $B_o(P_{\text{ref}})$   
UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)
- The oil compressibility

$$C = -\left(\frac{dB_o}{dP}\right)\bigg|_{B_o}$$

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

- The oil viscosity at the reference pressure  $\mu_o(P_{\text{ref}})$ .  
UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)
- The oil “viscosibility”

$$C_v = \left(\frac{d\mu_o}{dP}\right)\bigg|_{\mu_o}$$

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

ECLIPSE calculates  $B_o$  using:

$$B_o(P) = B_o(P_{\text{ref}})e^{-X} \quad \text{Eq. 3.150}$$

where

$$X = C(P - P_{\text{ref}})$$

and the product  $B_o\mu_o$  using:

$$B_o\mu_o P = B_o P_{\text{ref}}\mu_o P_{\text{ref}}e^{-Y} \quad \text{Eq. 3.151}$$

where

$$Y = (C - C_v)(P - P_{\text{ref}})$$

See also keyword [PVDO](#).

### Example

With NTPVT=1:

```
PVCDO  
3600 1.12 1.6e-5 0.88 0.0 /
```

## PVCO

## PVT properties of live oil in compressibility form (with dissolved gas)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [PVCO](#) 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 ( $P_{\text{bub}}$ ) 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: barsa ([METRIC](#)), psia ([FIELD](#)), atma ([LAB](#)), atma ([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:  $\text{sm}^3/\text{sm}^3$  ([METRIC](#)), Mscf/stb ([FIELD](#)), scc/scc ([LAB](#)),  $\text{sm}^3/\text{sm}^3$  ([PVT-M](#))

3. The formation volume factor of saturated oil at  $P_{\text{bub}}$ .

UNITS:  $\text{rm}^3/\text{sm}^3$  ([METRIC](#)), rb/stb ([FIELD](#)), rcc/scc ([LAB](#)),  $\text{rm}^3/\text{sm}^3$  ([PVT-M](#))

4. The viscosity of saturated oil at  $P_{\text{bub}}$ .

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](#))

ECLIPSE 300

**Note:** When the [PVCO](#) 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*

ECLIPSE 300

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 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.

ECLIPSE 100

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](#).

ECLIPSE 100

---

**Note:** The undersaturated parts of the oil PVT tables are extrapolated over NRPVT 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](#).

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 [PVCO](#) data will be interpolated in the same way as ECLIPSE 100.

---

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 NTPVT=2. NRPVT must have been set  $\geq 14$ :

```

PVC0
400  0.165  1.012  1.17  1.0E-5  1*
800  0.335  1*      1.14  1*      1*
1200 0.500  1.038  1*      1*      1*
1600 0.665  1.051  1.08  1*      1*
2000 0.828  1*      1.06  1*      1*
2400 0.985  1.075  1*      1*      1*
2800 1.130  1.087  1.00  1*      1*
3200 1.270  1*      0.98  1*      1*
3600 1.390  1.11   1*      1*      1*
4000 1.500  1.12   0.94  1*      1*
4400 1.600  1*      0.92  1*      1*

```

4800	1.676	1.14	1*	1*	1*
5200	1.750	1.148	0.9	1*	1*
5600	1.810	1.155	0.89	1*	1* /
400	0.18	1.02	1.12	1.0E-5	1*
1200	0.520	1.042	1.07	1*	1*
2000	0.84	1.070	1.01	1*	1*
2800	1.200	1.091	0.94	1*	1*
3600	1.50	1.124	0.90	1*	1*
4400	1.720	1.146	0.88	1*	1*
5200	1.870	1.161	0.87	1*	1*
5600	1.990	1.172	0.86	2.0E-5	1* /

## PVDG

## PVT properties of dry gas (no vaporized oil)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises NTPVT (item 2 in keyword `TABDIMS` in the `RUNSPEC` section) tables of dry gas PVT functions, each terminated by a slash (/).

Each table consists of 3 columns of data:

- The gas phase pressure.  
Values should increase monotonically down the column.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
- The corresponding gas formation volume factor.  
Values should decrease down the column.  
UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/Mscf (FIELD), rcc/scf (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)
- 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`).

See also keyword `PVZG`.

ECLIPSE 100

ECLIPSE 100

---

**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.

---

ECLIPSE 300

---

**Note:** ECLIPSE 300 linearly interpolates both  $B_g$  and  $\mu_g$  directly. If item 56 of the `OPTIONS3` keyword is set to 2, then PVDG data (as well as oil property data) will be interpolated in the same way as ECLIPSE 100.

---

## Example

With one pressure table:

```
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
```

## PVDO

## PVT properties of dead oil (no dissolved gas)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), atma (LAB), atma (PVT-M)

2. The corresponding oil formation volume factor.

Values should decrease down the column.

UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (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 [PVCDO](#).

## 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:

```
PVDO
 400 1.012 1.16
 1200 1.0040 1.164
 2000 0.9960 1.167
 2800 0.9880 1.172
 3600 0.9802 1.177
 4400 0.9724 1.181
 5200 0.9646 1.185
 5600 0.9607 1.19 /
 800 1.0255 1.14
 1600 1.0172 1.14
 2400 1.0091 1.14
 3200 1.0011 1.14
 4000 0.9931 1.14
 4800 0.9852 1.14
 5600 0.9774 1.14 /
```

## PVDS

## PVT properties of the solvent

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

2. The corresponding solvent formation volume factor.

Values should decrease down the column.

UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC),  $\text{rb}/\text{Mscf}$  (FIELD),  $\text{rcc}/\text{scc}$  (LAB),  $\text{rm}^3/\text{sm}^3$  (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)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- 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), atma (PVT-M)
- The vaporized oil-gas ratio for saturated gas at pressure  $P_g$ .  
UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)
- The gas formation volume factor for saturated gas at  $P_g$ .  
UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/Mscf (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)
- 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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

Column 3 The gas formation volume factor ( $B_g$ ) corresponding to  $R_v$  and  $P_g$ .

UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/Mscf (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (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 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
- ECLIPSE 300 linearly interpolates the **reciprocals** of  $B_g$  but  $\mu_g$  is interpolated directly.

## Example

With NTPVT=2. NRPVT must have been set  $\geq 2$  and NPPVT $\geq 7$ :

```

PVTG
 30 0.00014 0.0523 0.0234
    0      0.0521 0.0238 /
 90 0.00012 0.0132 0.0252
    0      0.0131 0.0253 /
150 0.00015 0.00877 0.0281
    0      0.00861 0.0275 /
210 0.00019 0.00554 0.0318
    0      0.00555 0.0302 /
270 0.00029 0.00417 0.0355
    0      0.00421 0.0330 /
330 0.00049 0.00357 0.0392
    0      0.00361 0.0358 /
530 0.00060 0.00356 0.0393
    0      0.00360 0.0359 /
/ null record to terminate table 1
 60 0.00014 0.0523 0.0234 /
120 0.00012 0.0132 0.0252 /
180 0.00015 0.00877 0.0281 /
240 0.00019 0.00554 0.0318 /
300 0.00029 0.00417 0.0355 /
360 0.00049 0.00357 0.0392 /
560 0.00060 0.00356 0.0393
    0      0.00360 0.0359 / undersaturated data for Pg=560
/ null record to terminate table 2

```

## PVT-M

## PVT-metric units are to be used

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## PVTO

## PVT properties of live oil (with dissolved gas)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

2. The bubble point pressure ( $P_{\text{bub}}$ ) 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  $P_{\text{bub}}$ .

UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)

4. The oil viscosity for saturated oil at  $P_{\text{bub}}$ .

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

However some records (optionally all) contain additional data which defines the properties of under-saturated 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  $P_{\text{bub}}$ ).

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:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)

Column 4 The oil viscosity ( $\mu_o$ ) corresponding to  $R_s$  and  $P_o$ . Values of  $\mu_o$  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’,

$$\left( \frac{d\mu_o}{dP_o} \right) / \mu_o \quad \text{Eq. 3.152}$$

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  $B_o$  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 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  $B_o$  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 $\geq 6$ :

```

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

```

## PVTW

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

- The reference pressure ( $P_{\text{ref}}$ ) for items 2 and 4.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  
DEFAULT: ERROR (ECLIPSE 100), 1 atma = 14.7psia (ECLIPSE 300).
- The water formation volume factor at the reference pressure,  $B_w(P_{\text{ref}})$   
UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC),  $\text{rb}/\text{stb}$  (FIELD),  $\text{rcc}/\text{sc}$  (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)  
DEFAULT: 1.0 (in all units)
- The water compressibility

$$C = -\left(\frac{dB_w}{dP}\right) / B_w \quad \text{Eq. 3.153}$$

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)

- 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).
- The water “viscosibility”

$$C_v = \left(\frac{d\mu_w}{dP}\right) / \mu_w \quad \text{Eq. 3.154}$$

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:

$$B_w(P) = \frac{B_w(P_{\text{ref}})}{1 + X + (X^2/2)} \quad \text{Eq. 3.155}$$

where

$$X = C(P - P_{\text{ref}})$$

ECLIPSE 100 calculates the product  $B_w\mu_w$  using:

$$B_w\mu_w(P) = \frac{B_w(P_{\text{ref}})\mu_w(P_{\text{ref}})}{1 + Y + (Y^2/2)} \quad \text{Eq. 3.156}$$

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

$$Y = (-C_v)(P - P_{\text{ref}})$$

## 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:

```
PVTW
3600.0000  1.00341  3.00E-06  0.52341  0.00E-01  /
3900      1      2.67E-06  0.56341  1.20E-07  /
```

## PVTWSALT Water PVT functions with salt present

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The reference pressure ( $P_{ref}$ ) for this table.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
- The reference salt concentration for stock tank water,  $c_{s\ ref}$ .  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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).

---

- The salt concentration,  $c_s$ .  
Values should increase monotonically down the column.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (PVT-M)
- The water formation volume factor at the reference pressure as a function of salt concentration,  $B_w(P_{ref}, c_s)$ .  
Values should be level or decreasing down the column.  
UNITS: rm<sup>3</sup>/sm<sup>3</sup> (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

3. The water compressibility as a function of salt concentration,

$$C(c_s) = -\left(\frac{dB_w}{dP}\right) \Big| B_w \quad \text{Eq. 3.158}$$

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,  $\mu_w(P_{ref}, c_s)$ .

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The water “viscosity” as a function of salt concentration,

$$C_v(c_s) = \left(\frac{d\mu_w}{dP}\right) \Big| \mu_w \quad \text{Eq. 3.159}$$

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:

$$B_w(P, c_s) = \frac{B_w(P_{ref}, c_s)}{1 + X + (X^2/2)} \quad \text{Eq. 3.160}$$

where

$$X = C(P - P_{ref})$$

$$B_w(P, c_s)\mu_w(P, c_s) = \frac{B_w(P_{ref}, c_s)\mu_w(P_{ref}, c_s)}{1 + Y + (Y^2/2)} \quad \text{Eq. 3.161}$$

where

$$Y = (C - C_v)(P - P_{ref})$$

## ECLIPSE 300

ECLIPSE 300 calculates  $B_w$  using equation 3.160.  $\mu_w$  is calculated similarly:

$$\mu_w(P, c_s) = \frac{\mu_w(P_{ref}, c_s)}{1 + Y + (Y^2/2)}$$

where

$$Y = -(C_v)(P - P_{ref})$$

## Example

With NTPVT=2, NPPVT≥4:

```

PVTWSALT
--
-- Reference Pressure      Reference concentration

```

```

--          5000          0.0          /
--
-- Salt conc      Bw      Cw          Vw      Cv
--
--   0.0          1.01    3.00E-06    0.52    0.0
--   2.0          1.00    3.00E-06    0.52    0.0
--   5.0          0.98    3.00E-06    0.52    0.0
--  20.0          0.95    3.00E-06    0.52    0.0    /
--
-- Reference Pressure
--
-- 4000 /
--
-- Salt conc      Bw      Cw          Vw      Cv
--
--   0.0          1.00    3.00E-06    0.50    0.0
--  20.0          0.95    4.00E-06    0.60    0.0    /

```

## PVZG

## PVT properties of dry gas (using Z-factors)

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$B_g = Z \times \left( \frac{T_{ref} + T_{base}}{T_s + T_{base}} \right) \times \frac{P_s}{P} \quad \text{Eq. 3.162}$$

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)

$T_s = 15.56 \times C$  (METRIC),  $60 \times F$  (FIELD), 15.56 °C (LAB)

and  $T_{base}$  is the absolute temperature at base measured temperature given by

$T_{base} = 273.105$  K (METRIC),  $459.598 \times R$  (FIELD), 273.105 K (LAB).

See also keyword [PVDG](#).

## Example

With NTPVT=1 and NPPVT≥8:

```
PVZG
-- Temperature
 150 /
```

```
-- Press      Z          Visc
   400.00    1.22      .01300
  1200.00    1.30      .01400
  2000.00    1.34      .01500
  2800.00    1.50      .01600
  3600.00    1.55      .01700
  4000.00    1.70      .01750
  4800.00    1.82      .01850
  5200.00    1.91      .01900
/
```

## **Q**

This section contains an alphabetic listing of keywords starting with the letter Q.

## QDRILL

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/day (METRIC), ft<sup>3</sup>/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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 1 2 4 4 4 /
QMOBIL
NO /
ENDFIN

```

## **R**

This section contains an alphabetic listing of keywords starting with the letter R.

## RADFIN4

## Specifies a four-column radial local grid refinement

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $I1 - I2, J1 - J2, K1 - K2$ . 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.  $I1$  lower I-coordinate of the column in the global grid.
3.  $I2$  upper I-coordinate of the box in the global grid.

This must be  $I2 = I1 + 1$

4.  $J1$  lower J-coordinate of the box in the global grid.
5.  $J2$  upper J-coordinate of the box in the global grid.

This must be  $J2 = J1 + 1$

6.  $K1$  minimum K-coordinate of the box in the global grid.
7.  $K2$  maximum 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 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, K1, K2. The dimensions of the refined grid within this column are specified as NR, NTHETA, 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.
3. J J-coordinate of the column in the global grid.
4. K1 Minimum K-coordinate of the column in the global grid.
5. K2 Maximum K-coordinate of the column in the global grid.
6. NR Number of radial divisions in the refined grid.
7. NTHETA 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 NTHETA = 4 is allowed.

*ECLIPSE 100 only*

8. NZ Number of vertical divisions in the refined grid.
9. NWMAX 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.)

*ECLIPSE 300 only*

10. Inner radius of refinement  
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)  
DEFAULT: 0.5 ft = 0.1524 m

*ECLIPSE 300 only*

11. 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

*ECLIPSE 300 only*

12. Minimum  $\Delta R$  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

```

## 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 `NTHETA = 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

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

**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 [AQUDIMS](#) 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:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 3. February aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of February.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 4. March aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of March.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 5. April aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of April.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 6. May aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of May.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 7. June aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of June.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

### 8. July aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of July.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

## 9. August aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of August.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

## 10. September aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of September.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

## 11. October aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of October.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

## 12. November aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of November.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

## 13. December aquifer flux

This is the water inflow rate per unit area of connected cell face for the month of December.

UNITS:  $\text{sm}^3/\text{day}/\text{m}^2$  (METRIC),  $\text{stb}/\text{day}/\text{ft}^2$  (FIELD),  $\text{scc}/\text{hr}/\text{cm}^2$  (LAB)

The water flow rate into a grid block from a constant flux aquifer is given by:

$$Q_w = F_w \cdot A \cdot M \quad \text{Eq. 3.163}$$

$F_w$  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
```

0.12  
0.15  
0.2 /

## RBEDCONT Tables of river bed/grid-block contact area data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
RCMASTS
10.0 /
```

## REACT

**Activation molar energy in chemical reaction rates**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The rate  $R_r$  of a chemical reaction depends on a rate constant ([REACRATE](#)), porosity ([REACPORD](#)), component concentrations ([REACCORD](#)), component phases ([REACPHA](#)), and temperature  $T$ :

$$R_r \propto e^{\left(-\frac{E_r}{RT}\right)}$$

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 [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 ([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 with activation energies 19500 kJ/kg-M and 13700 kJ/kg-M respectively:

```
REACT
19500 13700 /
```

## REACCORD Order of component terms in chemical reaction rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The rate  $R_r$  of a chemical reaction depends on a rate constant ([REACRATE](#)), the temperature ([REACTACT](#)), porosity ([REACPORD](#)), component phases ([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 [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.

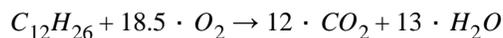
For reactants the order must be non-negative. non-zero orders can be entered for components that are not reactants - in which case, when the component concentration  $m_c > 1$ , the component acts as a catalyst (or inhibitor if the order is negative).

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 [EQLDREAC](#) is also used.

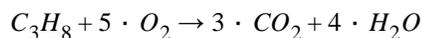
See "[Chemical reactions](#)" in the *ECLIPSE Technical Description*.

### Example

For a run with four components  $C_{12}H_{26}$ ,  $C_3H_8$ ,  $O_2$ ,  $CO_2$ , and water, and two reactions



and



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  /

```

## REACCRIT Reaction rate interpolation criteria

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$T = \theta \cdot T_n + (1-\theta) \cdot T_{n-1}, 0 < \theta \leq 1$$

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 multibranching) 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* /

```

## REACLIMS Pressure and temperature limits for calculating chemical reaction rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- Minimum pressure  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  
DEFAULT: No minimum
- Maximum pressure  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)  
DEFAULT: No maximum
- Minimum temperature  
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)  
DEFAULT: No minimum
- 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 NLRESMAX 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 /
```

## REACPHA Phase of component terms in chemical reaction rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The rate  $R_r$  of a chemical reaction depends on a rate constant ([REACRATE](#)), the temperature ([REACTACT](#)), 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 (/).

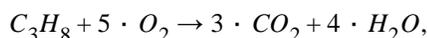
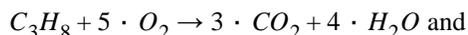
The default value is ALL. Phases are ignored for components with zero order ([REACCORD](#)).

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:



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:

```
REACPHA
--HEAVY LIGHT O2 CO2 H2O
 1* OIL GAS 1* 1* /
 1* GAS GAS 1* 1* /
```

**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**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The rate  $R_r$  of a chemical reaction depends on a rate constant ([REACRATE](#)), the temperature ([REACTACT](#)), 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The rate  $R_r$  of a chemical reaction depends on the temperature ([REACT](#)), porosity ([REACPOD](#)), 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 /
```

## REACSMAX Maximum solid saturation in chemical reaction rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

In chemical reactions involving solid components, the rate  $R_r$  of a chemical reaction depends on the solid saturation  $\hat{S}_s$ :

$$R_r \propto \left(1 - \frac{\hat{S}_s}{\hat{S}_{smax}}\right)^{n_s}$$

where

$\hat{S}_{smax}$  is the limiting solid saturation, defined by 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

In chemical reactions involving solid components the rate  $R_r$  of a chemical reaction depends on the solid saturation  $\hat{S}_s$ :

$$R_r \propto \left(1 - \frac{\hat{S}_s}{\hat{S}_{s \max}}\right)^{n_s}$$

where  $\hat{S}_{s \max}$  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:

```
REACSORD
--1 2 3 4
 1* 1* 1* 2 /
```

## REACTION **Number of chemical reactions**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $p$

$c_c = m_c$  if the tracer phase is ALL

$c_c = b_o \cdot S_o \cdot x_c$  if the tracer phase is OIL

$c_c = b_g \cdot S_g \cdot y_c$  if the tracer phase is GAS

$c_c = b_w \cdot S_w$  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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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
/

```

## REFINE

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 [ENDFIN](#) or another REFINE keyword is entered.

Local grid refinement is described in "[Local grid refinement and coarsening](#)" in the *ECLIPSE Technical Description*.

The REFINE keyword is available in the GRID, EDIT, PROPS, REGIONS, SOLUTION and SCHEDULE sections.

ECLIPSE 100 imposes the following restrictions on the keywords that can be used with this keyword.

### GRID section

The REFINE keyword can be used in the GRID section to return to a local grid that has previously been introduced with [CARFIN](#) or [RADFIN](#), to make further modifications to its grid data or to supply additional data.

In addition, the [BOX](#), [COPY](#), [EQUALS](#), [MULTIPLY](#) and [COPYBOX](#) keywords can be used to set or modify GRID section keywords in regions of the local grid.

### EDIT section

The following grid arrays may be edited for a local grid:

```
PORV  TRANX  TRANR  DIFFX  DIFFR
DEPTH  TRANY  TRANTH  DIFFY  DIFFTH
      TRANZ          DIFFZ
```

In addition, the [BOX](#), [COPY](#) 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.

### PROPS section

In end point scaling runs (keyword [ENDSCALE](#) in the RUNSPEC section) the following keywords and their directional equivalents are available:

```
SWL ,      SWCR ,      SWU
SG ,      SGCR ,      SGU
SOWCR ,   SOGCR
KRO ,     KRORW ,     KRORG
IKRO ,    IKRORW ,    IKRORG
KRW ,     KRWR ,     IKRW ,     IKRWR
```

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

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 [PVTNUM](#) 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:

PVTNUM

SATNUM

KRUNUMX    KRUNUMY    KRUNUMZ    (Directional  $K_r$  cases)

KRUNUMX-    KRUNUMY-    KRUNUMZ-

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

```
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

```
REFINE
  'LGR1' /
SWL
  300*0.25 /
EQUALS
  'SWL' 0.20 1 10 1 10 2 2 /
/
COPY
  'SWL' 'SWCR' /
/
ENDFIN
```

### REGIONS section example

```
REFINE
  'EAST' /
SATNUM
  20*1
  20*2
  20*1 /
EQUALS
  'SATNUM' 1 2 4 2 5 2 2 /
/
ENDFIN
```

### SOLUTION section example

```
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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

---

**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, NTCREG, as used by [COALNUM](#) in the GRID section.

DEFAULT: 1

7. The maximum number of user-defined regions used by the [OPERNUM](#) keyword in the GRID and REGIONS sections.

DEFAULT: 0

8. The maximum number of double-precision work arrays for use with the [OPERATE](#) and [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 [OPERATE](#) and [OPERATER](#) keywords. An integer value between 1 and 999 should be specified.

DEFAULT: 0

*ECLIPSE 100 only*

10. The number of polymer flooding mixing regions, NPLMIX, as used by [PLMIXNUM](#) in the GRID section.

Default: 1

## Example

```
REGDIMS
5 5 0 1 /
```

## RESIDNUM **VE** residual flow region numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## 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*.

---

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
	OPTIMIZE

## RESORB

## Coal Bed Methane re-adsorption factor

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.  $r_1 r_2 r_3 \dots r_{ncomps}$

Values should be in [0,1]

DEFAULT: 1.0

Re-adsorption factors for coal region number 1.

2.  $r_1 r_2 r_3 \dots r_{ncomps}$

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
/
```

## RESTART

## Read solution from a restart or a save file

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 `RPTRST` 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.

---

*ECLIPSE 100 only*

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.

*ECLIPSE 100 only*

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`, `RPTRST`, `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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## RIVDEBUG Controls on debug output for individual rivers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>st</sup> figure controls debug from river flow and property calculations.

The 2<sup>nd</sup> figure controls debug from connection flow calculations.

The 3<sup>rd</sup> figure is currently unused.

The 4<sup>th</sup> 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 /
/
```

## RIVERSYS **Defines the branch structure and boundary conditions of a river**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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 MXBRR1. 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 MXBRR1. 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  $dh/dl > 0$  and m is the table number to use when  $dh/dl < 0$ . 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 /
/
```

## RIVRDIMS River dimension data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

- The maximum number of rivers in the model.  
DEFAULT: 0
- 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
- The maximum number of branches in a river (including the main channel).  
DEFAULT: 1
- The maximum number of grid blocks to be associated with any one river. (See [COMPRIV](#))  
DEFAULT: 1
- Number of tables describing channel profile, MXTBPR. (See [RIVRXSEC](#))  
DEFAULT: 1
- Maximum number of depth entries in a channel profile table, or bed contact area table, MXDPTB. (See [RIVRXSEC](#) or [RBEDCONT](#))  
DEFAULT: 2
- Number of tables describing contact areas between grid blocks and the river bed, MXTBGR. (See [RBEDCONT](#))  
DEFAULT: 1
- Number of depth tables used as boundary conditions, NMDEPT. (See [DEPTHHTAB](#))  
DEFAULT: 0
- The maximum number of entries in a downstream boundary condition depth table, MXDEPT.  
DEFAULT: 2
- The number of massflow (boundary condition) tables, NMMAST. (See [MASSFLOW](#))  
DEFAULT: 0
- The maximum number of entries in a massflow table, MXMAST.  
DEFAULT: 2
- Number of rating tables, NRATTA. (See [QHRATING](#))  
DEFAULT: 0
- The maximum number of entries in a rating table, MXRATE.

DEFAULT: 2

## Example

```
RIVRDIMS  
1 40 2 40 10 20 1 1 50 1 50 /
```

## RIVRPROP

**Modifies properties of river reaches**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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**

```
RIVRPROP
-- Name      Reaches  Manning Coef
  RIVER1     15  17    0.03 /
  RIVER2     12  12    0.02 /
/
```

## RIVRXSEC Tables of river profile data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword is used to define the cross-section of a river as a function of the depth.

The keyword is followed by MXTBPR 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 3 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. Wetted perimeter at this depth

The values should increase monotonically down the table

UNITS: m (METRIC), ft (FIELD), cm (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<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (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 two tables for a simple box section channel 5 metres wide; the second being identical to the first:

```
RIVRXSEC
0.0 5.0 0.0
10.0 25.0 50.0/
/
```

## RIVSALT

## Sets salt concentrations in the upstream flow of rivers branches

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

The set of records must end with a blank record, containing only a slash (/).

1. River name
2. The concentration of salt in the injection stream.

UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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/
/
```

## RIVTRACE Sets tracer injection concentrations for rivers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$T_c = \text{MIN}(CI \cdot T_{\text{cum}}, T_{\text{conc}}) \quad \text{Eq. 3.164}$$

where

CI Total cumulative water flow at the previous timestep

$T_{\text{cum}}$  Cumulative tracer factor

$T_{\text{conc}}$  Tracer concentration entered in item 3.

UNITS: 1/sm<sup>3</sup> (METRIC), 1/stb (FIELD), 1/scc(LAB), 1/sm<sup>3</sup> (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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises NTROCC (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 NTROCC 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 NTROCC =1

```
RKPVMNMX
0.3 0.8 /
```

## RKTRMDIR

## Allow ROCKTAB tables to contain directional transmissibility multipliers

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## ROCK

## Rock compressibility

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ( $P_{ref}$ ).

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 1.0132 barsa (METRIC), 14.7 psia (FIELD), 1 atma (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)

DEFAULT: 0.0

ECLIPSE 100

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

$$V(P) = V(P_{ref}) \left( 1 + X + \frac{X^2}{2} \right) \quad \text{Eq. 3.165}$$

where  $X = C(P - P_{ref})$

and  $V(P_{ref})$  is the pore volume at the reference pressure.

The NTPVT records correspond to the PVT region numbers, which are set using keyword [PVTNUM](#).

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  $C_{bc} = \varphi_0 \cdot C_{pp} + (1 + \varphi_0 \cdot )C_R$  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  $C_R = \frac{C_{bc}}{1 + \frac{3(1-\nu_R)\varphi_0}{2(1-2\nu_R)(1-\varphi_0)}}$  if the reference Poisson ratio

$\nu_R$  is set in item 6 or  $C_R = \frac{C_{bc}}{1 + \frac{3\varphi_0}{2(1-\varphi_0)}}$  if it is not. Note that if  $C_{pp}$  is unset, then a default value of

0.00005 bars<sup>-1</sup> will be assumed. Also note that if the reference porosity  $\varphi_0$  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  $C_r = -\left(\frac{dB_r}{dP}\right) / B_r$ .

---

## Example

With NTPVT=3:

```
ROCK
 3600.00 .40E-05 /
```

```
3600.00 .40E-05 /  
3000 0 /
```

## ROCK2D

## 2D rock compaction tables versus P and Sw

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```

ROCK2D
  0.0   0.88
        0.88
        0.88
        0.88 /
 1000.0 0.92
        0.92
        0.90
        0.90 /
 9000.0 1.0
        1.0

```

0.94  
0.94 /  
/

## ROCK2DTR 2D transmissibility modifier tables for water induced rock compaction

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ( $P_{\text{eff fl}} = P_{\text{fluid}} - \text{Overburden}$ ) 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](#), [ROCK2D](#), [ROCKWNOD](#), [ROCKOPTS](#) and [OVERBURD](#), and "[Rock compaction](#)" in the *ECLIPSE Technical Description*.

### Example

With NTROCC=1 and NSSFUN ≥4:

```

ROCK2DTR
  0.0  0.81
      0.80
      0.74
      0.73 /
1000.0 0.93
      0.92

```

```
          0.87  
          0.86 /  
9000.0   1.0  
          1.0  
          0.94  
          0.94 /  
/
```

## ROCKCOMP Enables the rock compaction option

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 <a href="#">ROCKTABH</a> 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 <a href="#">ROCKTABH</a> in the PROPS section. |
| <i>ECLIPSE 100 only</i> | 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 <a href="#">COAL</a> ). The Palmer-Mansoori model ( <a href="#">[Ref. 78]</a> in the <i>ECLIPSE Technical Description</i> ) is used to calculate the pore volume and transmissibility multipliers. The parameters for the Palmer-Mansoori model should be entered using keyword <a href="#">ROCKPAMA</a> in the PROPS section.   |
| <i>ECLIPSE 300 only</i> | TPLM-MAN This option is only available with the THERMAL option (see <a href="#">THERMAL</a> ). The Thermal Palmer-Mansoori model ( <a href="#">[Ref. 78]</a> in the <i>ECLIPSE Technical Description</i> ) is used to calculate the pore volume and transmissibility multipliers. The parameters for the Palmer-Mansoori model should be entered using keyword <a href="#">ROCKPMAT</a> 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:

$$\frac{\kappa}{\kappa_0} = \frac{\varphi}{\varphi_0} \tau \left[ \left( \frac{1-\varphi_0}{1-\varphi} \right) \right]$$

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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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+' /
/
```

## ROCKCONL Specifies local grid refinement connection data for cap and base rocks

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
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. 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 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 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+' /
/
```

## ROCKDEN **Rock density**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> ([METRIC](#)), lb/ft<sup>3</sup> ([FIELD](#)), gm/cc ([LAB](#)), kg/m<sup>3</sup> ([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 /
```

## ROCKDIMS **Rock over and underburden dimensions**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

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.

```
ROCKFRAC
500*0.8 500*0.1 /
```

*ECLIPSE 100 only*

## ROCKNUM **Rock compaction table region numbers**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [ROCKTSIG](#) 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.

See also keywords [ROCKWNOD](#), [ROCKOPTS](#) and [OVERBURD](#), and "[Rock compaction](#)" in the *ECLIPSE Technical Description*.

ECLIPSE 100

ECLIPSE 300

---

**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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $P_{\text{effov}} = \text{Overburden} - P_{\text{fluid}}$ , 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.

*ECLIPSE 300*

If the **STORE** option is used with the **ROCK** keyword for the elastic rock compressibility model, the input pore volume for a cell is similarly defined as the reservoir pore volume at initial conditions rather than the pore volume at the reference pressure.

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*.

ECLIPSE 300

---

**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

```
ROCKOPTS
STRESS  NOSTORE/
```

## ROCKPAMA Palmer-Mansoori rock model data

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $\varepsilon_l$   
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,  $\varepsilon_s$

UNITS: dimensionless

DEFAULT: Same as item 5,  $\varepsilon_l$

The Palmer-Mansoori model adjusts the pore volumes,  $V(P)$ , at the current pressure,  $P$ , using

$$V(P) = V(P_0) \left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) (\text{eps} - \text{eps}_0) \right\}$$

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  $\left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) (\text{eps} - \text{eps}_0) \right\}^n$

The calculation of the volumetric strain term due to gas adsorption or desorption (eps) depends on the setting of [CBMOPTS](#) item 5, where eps0 is the initial strain term.

PMSTD:

This is the standard form assuming instant adsorption

$$\text{eps}(P) = \frac{\varepsilon_l \beta P}{1 + \beta P}$$

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 (eps) 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:

ECLIPSE 100 only

$$\text{eps}(G_C, S_C) = \frac{\varepsilon_l \beta P_g + \varepsilon_s \beta_s P_s}{1 + \beta P_g + \beta_s P_s}$$

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 [RPTRST](#) using [COALGAS](#) or [RPTSCHED](#) using [GASCONC](#). Using the [LANGMUIR](#) and [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,  $G_C = \text{LANGMUIR}(P_g)$  and  $S_C = \text{LANGSOLV}(P_s)$ .

Note that using the different sorption models [COALADS](#) or [COALPP](#) also affects the computations of the eps term, since the adsorption models are different.

Alternatively the eps term can be looked up directly from the [TPAMEPS](#) and [TPAMEPSS](#) tables (ECLIPSE 100 and ECLIPSE 300)

$$\text{eps}(G_C, S_C) = T_{\text{PAMEPS}}(G_C) + T_{\text{PAMEPSS}}(S_C)$$

which enables the user to specify the sorption-induced strain term directly as a function of the adsorbed quantities. If [TPAMEPS](#) is used and the case includes [SOLVENT](#), [TPAMEPSS](#) should also be present, otherwise the solvent concentration is not accounted for ( $\epsilon_s$  is not used).

For ECLIPSE 300, [TPAMEPS](#) tabulates the eps term for each [NCOMPS](#) component, where the total volumetric strain is the sum over each component:

$$\text{eps}(Gc_1, \dots, Gc_{\text{NCOMPS}}) = \left( \sum_{i=1}^{\text{NCOMPS}} T_{\text{PAMEPS}}(Gc_i) \right)$$

The bulk coal gas concentration can be output by using [RPTRST](#) or [RPTSCHED](#) using [CGAS](#).

Using the [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 [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.

[PMADTT](#) This method is doing exactly the same as [PMADTN](#) but use the gas concentration and solvent concentrations at the previous timestep. For ECLIPSE 300 this method requires [TPAMEPS](#).

---

**Note:** The alternative methods will only equal the standard Palmer-Mansoori method ([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  $\varepsilon_l$ ,  $\varepsilon_s$  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 ([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.

## 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $5 + 2 \cdot n_c$  columns of data, where  $n_c$  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 $n_c+5$	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 $n_c+6$ .... Item $2 \cdot n_c + 5$	Langmuir curve parameter, $\varepsilon_i$ for component $i$ . UNITS: dimensionless

DEFAULT: 0

The Palmer-Mansoori model adjusts the pore volumes,  $V(P, Y)$ , at the current pressure,  $P$  and gas phase composition  $Y = (y_1, y_2, \dots, y_{nc})$  using

$$V(P, Y) = V_0 \left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) (E - E_0) \right\}$$

where

$\varphi_0$  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  $c_m = \frac{1}{M} - \left[ \frac{K}{M} + f - 1 \right] \gamma$ .

The strain for component  $k$  is computed from the input Langmuir parameters by  $e_k = \frac{\varepsilon_k \beta_k a_k P}{1 + \sum \beta_j a_j P}$

where  $a_k = L_k / (\sum L_j)$  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 [LANGMEXT](#) or [LANGMUIR](#)).

The total strain is computed as

$$E = \sum e_k$$

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

$$\left\{ 1 + \frac{c_m}{\varphi_0} (P - P_0) + \frac{1}{\varphi_0} \left( \frac{K}{M} - 1 \right) (E - E_0) \right\}^n$$

It is possible to output a table of the pore volume and permeability multipliers to the PRT file by using the keyword [RPRKPAME](#) in the 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 [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 [RPTSCHED](#) or [RPTRST](#).
- 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
/
```

## ROCKPMAT Thermal Palmer-Mansoori rock model data

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $\varepsilon_l$   
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/m<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/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

$$\Delta V(P, T) = V(P_0, T_0) \left\{ \frac{c_m}{\phi_0} \Delta P + \frac{1}{\phi_0} \left( \frac{K}{M} - 1 \right) \alpha \Delta T \right\}$$

$$\Delta E^T = \Delta E^C - \frac{c_v}{\alpha} \left( \frac{\varepsilon_l \beta P}{1 + \beta P} \right)_p \Delta P$$

where

$\phi_0$  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

$c_m = \frac{1}{M} \left[ \frac{K}{M} + f - 1 \right] \gamma$  is the bulk elastic compressibility

This follows the conventional Palmer-Mansoori model, assuming that permeability varies with porosity as

$$\frac{k}{k_0} = \left( \frac{\phi}{\phi_0} \right)^n$$

**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:

```

ROCKPMAT
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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- Rock type identification number  
This should be a number between 1 and the maximum number of rocks, entered using [ROCKDIMS](#) in the RUNSPEC section.
- Initial temperature  
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
- Rock conductivity  
UNITS: kJ/m/day/K (METRIC), Btu/ft/day/°R (FIELD), J/cm/hr/K (LAB), kJ/m/day/K (PVT-M)
- Volumetric heat capacity of the rock  
UNITS: kJ/m<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/K (PVT-M)
- Temperature-dependent part of the volumetric heat capacity of the rock  
UNITS: kJ/m<sup>3</sup>/K<sup>2</sup> (METRIC), Btu/ft<sup>3</sup>/°R<sup>2</sup> (FIELD), J/cc/K<sup>2</sup> (LAB), kJ/m<sup>3</sup>/K<sup>2</sup> (PVT-M)  
DEFAULT: 0.0.
- Calculation method for heat loss  
This has the value 'V' for Vinsome and Westerveld or 'N' for numerical.  
DEFAULT: Vinsome.
- 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## ROCKTAB **Rock compaction data tables**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 13<sup>th</sup> 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 ( $P_{\text{eff fl}} = P_{\text{fluid}} - \text{Overburden}$ ) 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](#)≥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 /

```

## ROCKTABH Hysteretic rock compaction data tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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).

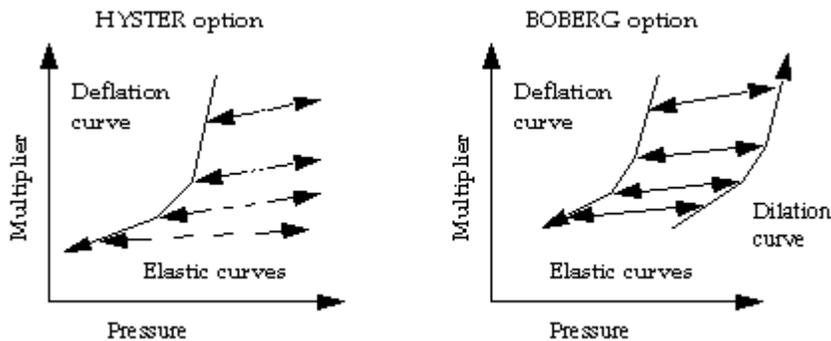


Figure 3.13. Hysteresis options

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 13<sup>th</sup> 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 ( $P_{eff\ fl} = P_{fluid} - \text{Overburden}$ ) 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, ( $P_{eff\ ov} = \text{Overburden} - P_{fluid}$ ), 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.

#### ECLIPSE 100

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:

```

ROCKTABH
-- Table 1
1000 .96 .98
2000 .97 .985
3000 .98 .99
4000 .99 .995 /
2000 .99 .99
3000 .995 .995
4500 .998 .998 /
3000 1.0 1.0
5000 1.01 1.01 /
4000 1.01 1.01
5500 1.02 1.02 /
/
-- Table 2
1000 .94 .97
2700 .94 .97 /
2000 .98 .99
3500 .98 .99 /
3000 1.0 1.0
4300 1.0 1.0 /
4000 1.02 1.01
5000 1.02 1.01 /
/

```

In this example the deflation curve for Table 1 is:

```

1000 0.96 0.98
2000 0.99 0.99
3000 1.00 1.00
4000 1.01 1.01

```

and the dilation curve (if used) is:

```

4000 0.99 0.995
4500 0.998 0.998
5000 1.01 1.01
5500 1.02 1.02

```

## ROCKTABW Water induced compaction tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
0.3 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 /

```

## ROCKTHSG Hysteretic rock compaction data tables in dual porosity runs

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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).

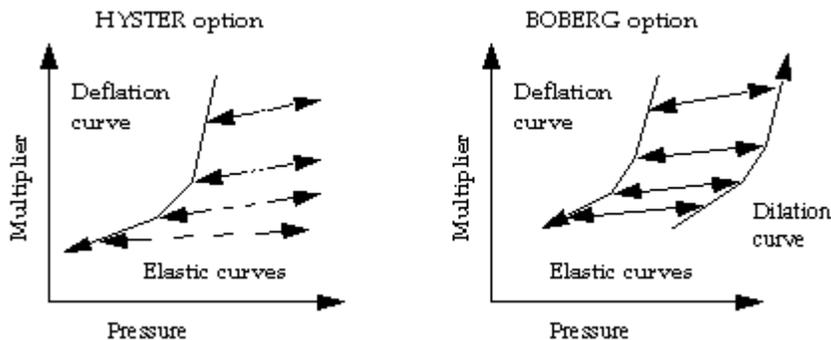


Figure 3.14. Hysteresis options

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](#).

ECLIPSE 300

**Note:** NTROCC can be set using the 13<sup>th</sup> 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 ( $P_{eff\ fl} = P_{fluid} - \text{Overburden}$ ) 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, ( $P_{eff\ ov} = \text{Overburden} - P_{fluid}$ ), which is then positive.

ECLIPSE 100

**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](#), [RKPVMNMX](#), [OVERBURD](#) and [ROCKOPTS](#), and "[Rock compaction](#)" in the *ECLIPSE Technical Description*.

## Example

With NTROCC=2 specified keyword [ROCKCOMP](#) and NPPVT≥4 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 /

```

```
4000 1.01
5000 1.01 /
/
```

In this example the deflation curve for Table 1 is:

```
1000 0.98
2000 0.99
3000 1.00
4000 1.01
```

and the dilation curve (if used) is:

```
4000 0.995
4500 0.998
5000 1.01
5500 1.02
```

## ROCKTRMX Transmissibility variation parameters in X-direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\frac{\kappa}{\kappa_0} = e^{\left[ \kappa_{\text{mult}} \left( \frac{\varphi - \varphi_0}{1 - \varphi_0} \right) \right]}$$

where

$\kappa_0$  is the initial transmissibility,

$\varphi_0$  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 [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](#):

```
ROCKTRMX  
120*0  
120*0.1  
120*0.5  
120*0.7  
120*1 /
```

## ROCKTRMY Transmissibility variation parameters in Y-direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\frac{\kappa}{\kappa_0} = e^{\left[ \kappa_{\text{mult}} \left( \frac{\varphi - \varphi_0}{1 - \varphi_0} \right) \right]}$$

where

$\kappa_0$  is the initial transmissibility,

$\varphi_0$  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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\frac{\kappa}{\kappa_0} = e^{\left[ \kappa_{\text{mult}} \left( \frac{\varphi - \varphi_0}{1 - \varphi_0} \right) \right]}$$

where

$\kappa_0$  is the initial transmissibility,

$\varphi_0$  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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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), 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 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](#).

ECLIPSE 300

---

**Note:** NTROCC can be set using the 13<sup>th</sup> 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 ( $P_{eff\ fl} = P_{fluid} - \text{Overburden}$ ) 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, ( $P_{eff\ ov} = \text{Overburden} - P_{fluid}$ ), 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`:

```
ROCKTSIG
1000 .98
2000 .99
3000 1.0
4000 1.0 /
1000 .97
2000 .99
3000 1.0
4000 1.01 /
```

## ROCKV

## Grid block rock volumes

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{m}^3$  (METRIC), rb (FIELD), rcc (LAB),  $\text{m}^3$  (PVT-M)

## Example

```
ROCKV
24*1.324E4 24*0 12*1876 /
```

## ROCKWNOD The water saturation node values for water-induced compaction tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
1000.0 0.92
        0.91
        0.90
        0.90 /
9000.0 1.0

```

```
1.0  
0.94  
0.94 /  
/
```

## ROMF

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/
/
```

## RPTGRID

## Controls on output from GRID section

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Mnemonic	Output	E100	E300
ALLNNC	Output of all global-grid non-neighbor connections from all sources	x	x
AQUCON	Output of numerical aquifer connections	x	x
AQUNUM	Output of numerical aquifer definitions	x	x
COALNUM	Output of coal region numbers	x	x
COALV	Output of grid block coal volumes (Coal Bed Methane)	x	x
COLLAPSE	Output of VE compression flag	x	
COORD	Output of coordinate lines	x	
COORDSYS	Output of coordinate systems	x	
DEPTH	Output of grid block center depths	x	x
DIFFMMF	Output of Matrix/Fracture diffusivity multipliers	x	
DIFFMX	Output of X diffusivity multipliers	x	x
DIFFMY	Output of Y diffusivity multipliers	x	x
DIFFMZ	Output of Z diffusivity multipliers	x	x
DIFFX or DIFFTX	Output of X diffusivities	x	x
DIFFY or DIFFTY	Output of Y diffusivities	x	x
DIFFZ or DIFFTZ	Output of Z diffusivities	x	x

Mnemonic	Output	E100	E300
DOMAINS	Output of domains (Parallel Option). See also the <a href="#">DOMAINS</a> keyword. The output consists of: <ul style="list-style-type: none"> <li>The solver direction, which ECLIPSE calculates automatically but which may be modified by using the <a href="#">SOLVDIRS</a> keyword.</li> <li>The split of the outer solver dimension into domains.</li> </ul> The split can be entered using the <a href="#">DOMAINS</a> keyword, otherwise ECLIPSE will calculate it automatically.	x	
DPCON or SIGMA	Output of dual porosity connections	x	
DPNUM	Output of dual porosity/single porosity regions	x	x
DX	Output of X direction grid block sizes	x	x
DY	Output of Y direction grid block sizes	x	x
DZ	Output of Z direction grid block sizes	x	x
DZMTRXV	Output of $D_{z_{mtrx}}$ values	x	
DZNET	Output of net Z-direction grid block sizes		x
EXTHOST or EXTFIN	Output the host cell map for Pebi local grid refinements supplied using the <a href="#">EXTFIN</a> keyword	x	
FAULTS	Output of fault data	x	x
FLUXNUM	Output of flux region numbers	x	x
HEATTX	X thermal transmissibility		x
HEATTY	Y thermal transmissibility		x
HEATTZ	Z thermal transmissibility		x
HM****	Outputs multipliers for gradient parameters		x
IOCN	Output of active cell numbers	x	x
IONROCK	Rock ion exchange capacity	x	
ISOLNUM	Output of independent reservoir region numbers	x	
JFUNCR	Output of Leverett J-Function data specified using <a href="#">JFUNCR</a> keyword.	x	x
KOVERD	Output of K/D values (Surfactant Model)	x	
LIMITS	Outputs limits (max/min) of grid arrays		x
LX	Output of matrix block size, X-direction (Viscous Displacement Option)	x	
LY	Output of matrix block size Y-direction	x	
LZ	Output of matrix block size Z-direction	x	
MIDS	Output of cell mid-point depths		x
MINPV or MINPVV	Output of cells with pore volume less than the input minimum values	x	x

Mnemonic	Output	E100	E300
MPFANUM	Output the <a href="#">MPFANUM</a> data when using the multi-point flux approximation		x
MPFORTH	Output the multi-point flux k-orthogonality error		x
MPFTRAN	Output the multi-point flux forward transmissibilities		x
MPFTREV	Output the multi-point flux reverse transmissibilities		x
MULTNUM	Output of regions for inter-region transmissibility multipliers	x	x
MULTPV	Output of the pore volume multipliers	x	x
MULTREGP	Output region pore volume multipliers	x	x
MULTREGT	Output of multipliers between regions	x	x
MULTX	Output of X transmissibility multipliers	x	x
MULTY	Output of Y transmissibility multipliers	x	x
MULTZ	Output of Z transmissibility multipliers	x	x
NINENUM	Output the <a href="#">NINENUM</a> data when using the nine-point scheme	x	x
NNC	Output of non-neighbor connections	x	x
NTG	Output of grid block net-to-gross ratios	x	x
OPERNUM	Output of regions for use with <a href="#">OPERATER</a> keyword	x	x
PERMX	Output of X direction permeabilities	x	x
PERMY	Output of Y direction permeabilities	x	x
PERMZ	Output of Z direction permeabilities	x	x
PERMXY, PERMYZ or PERMZ	Output of all tensor permeability coefficients		x
PINCH or PINCHREG	Output of pinchout data	x	
PINCHNUM	Output of pinchout region numbers	x	
POLYMER or FOAM	Output of 1/(porosity*area) (Polymer and Foam Models)	x	
PORO	Output of grid block porosities	x	x
PORV	Output of grid block pore volumes	x	x
PORVF	Output of grid block fluid pore volumes (Coal Bed Methane option)	x	x
ROCKFRAC	Output of grid block rock volume fractions (Coal Bed Methane option)	x	x
ROCKVOL	Output of bulk (net and gross) rock volumes to the print file. In addition, if <a href="#">FIPNUM</a> regions are defined, the bulk rock volumes for each region will be output.	x	x
SIGMAGDV	Output of <a href="#">SIGMAGD</a> values (gravity drainage option)		x

Mnemonic	Output	E100	E300
SIGMAK	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 <a href="#">LTOSIGMA</a> keyword is set to ALL.	x	
SIGMAV	Output of Sigma values	x	x
SOLVMUM	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 <a href="#">SOLVNUM</a> keyword.	x	x
THCAVE	Output of average thermal conductivities		x
THCGAS	Output of gas phase thermal conductivities		x
THCOIL	Output of oil phase thermal conductivities		x
THCONR	Output of thermal rock and fluid conductivities	x	x
THCROCK	Output of rock thermal conductivities		x
THCSOLID	Output of solid phase thermal conductivities		x
THCWATER	Output of water phase thermal conductivities		x
TOPS	Output of grid block top depths	x	x
TRANX	Output of X transmissibilities	x	x
TRANY	Output of Y transmissibilities	x	x
TRANZ	Output of Z transmissibilities	x	x
TRTHERM	Output of thermal transmissibilities	x	
VELFLO	Output of 1/(porosity*area)  Only allowed if velocity or flow output requested via summary mnemonics or restart output	x	
ZCORN	Output of cell corner depths	x	

Table 3.29: RPTGRID output controls

ECLIPSE 100

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.

Control	Setting	Definition
COORD control	0	No output
	1	Output to the PRINT file.
	2	COORD keyword written to the DEBUG file in floating point format (F12.2)
	3	COORD keyword written in exponent format (1PE12.6)
ZCORN control	0	No output
	1	Output to the PRINT file.

Control	Setting	Definition
	2	ZCORN keyword written to the DEBUG file in floating point format (F12.2)
	3	ZCORN keyword written in exponent format (1PE12.6)

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.

Character replacing the decimal point	Meaning
C	Coarsened cell
T	Compressed vertical equilibrium — cell is the top (representative) cell in a column of merged cells
V	Compressed vertical equilibrium — cell is in a column of merged cells but not the top (representative) cell
R	Cell is refined (contains an LGR)
F	In a <a href="#">USEFLUX</a> run using pressure/saturation boundary conditions, flow is specified in this cell
*	Cell contains a well connection

Table 3.31: Output format conventions

## Examples

### Example 1

```
RPTGRID
COORD TRANX TRANY TRANZ ALLNNC /
```

### Example 2

ECLIPSE 100

Changing the level of output

```
RPTGRID
COORD=2 ZCORN=2 /
```

## RPTGRIDL GRID section data output for local refined grids

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

#### ECLIPSE 300

---

**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.

---

#### ECLIPSE 100

---

**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.

# RPTHMD

## 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.

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

Item	Setting	Description
Item 1	If set = 0	Derivatives for all parameters are output.
	If set > 0	No derivatives are output and all derivative calculations are switched off (equivalent to <a href="#">NOHMD</a> ).
	DEFAULT: 0	
Item 2	If set = 0	Derivatives are output at each simulator timestep
	If set > 0	Derivatives will be output only at report times ( <a href="#">RPTONLY</a> )
	DEFAULT: 0	
Item 3	If set = 0	Derivatives will be output for all producers, injectors and any shut or stopped wells.
	If set > 0	Derivatives will only be output for active producers (a producer is active if it is open or stopped, but not shut.)
	DEFAULT: 0	
Item 4	If set = 0	The values of well rates, pressures and so on are written in addition to the derivatives
	If set > 0	The values of well rates, pressures and so on ('VALUES' record in the .HMD file) are not output.
	DEFAULT: 0	
Item 5	If set = 0	The 'VALUES ' and 'DERIVS' records in the .HMD file will be double precision.'
	If set > 0	The 'VALUES ' and 'DERIVS ' records in the .HMD file will be single precision.
	DEFAULT: 0	
Item 6	If set = 0	Group, field and cumulative production gradients may not be requested
	If set > 0	Cumulative gradients (WOPT, WWPT and WGPT for example) may be requested using the <a href="#">GRADWELL</a> keyword.  Group and field gradients (GOPR, GWPR and GGPR for example) may be requested using <a href="#">GRADGRUP</a> keyword.
	DEFAULT: 0	

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

```
RPTHMD
 0 1 1 1 /
```

## RPTHMG

## Configure output of derivatives for groups

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 6<sup>th</sup> switch of the RPTHMG 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
/
```

## RPTHMW

## Configure output of derivatives for wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

```
'W*'      ON      ON /  
/
```

## Example 2

Switch on output for every well

```
RPTHMW  
/
```

## RPTINIT

## Controls on output from GRID and EDIT sections to INIT file

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Mnemonic	Output	E100	E300
AREAX	Output of X direction Area (Velocity dependent flow option; Velocity dependent relative permeabilities (E300))	x	x
AREAY	Output of Y direction Area (Velocity dependent flow option; Velocity dependent relative permeabilities (E300))	x	x
AREAZ	Output of Z direction Area (Velocity dependent flow option; Velocity dependent relative permeabilities (E300))	x	x
BTOBALFV	Output of Dual Porosity block-to-block matrix-fracture coupling multiplier	x	
COALV	Output of grid block coal volumes (Coal Bed Methane)	x	x
DEPTH	Output of grid block center depths	x	x
DIFFX	Output of X direction diffusivities	x	x
DIFFY	Output of Y direction diffusivities	x	x
DIFFZ	Output of Z direction diffusivities	x	x
DIFFMX	Output of X direction diffusivity multipliers	x	x
DIFFMY	Output of Y direction diffusivity multipliers	x	x
DIFFMZ	Output of Z direction diffusivity multipliers	x	x
DIFFMMF	Output of Matrix/Fracture diffusivity multipliers	x	
DIFFMX-	Output of X- direction diffusivity multipliers	x	x

Mnemonic	Output	E100	E300
DIFFMY-	Output of Y- direction diffusivity multipliers	x	x
DIFFMZ-	Output of Z- direction diffusivity multipliers	x	x
DR	Output of R direction grid block sizes	x	x
DTHETA	Output of THETA direction grid block sizes	x	x
DX	Output of X direction grid block sizes	x	x
DY	Output of Y direction grid block sizes	x	x
DZ	Output of Z direction grid block sizes	x	x
DZMTRXV DZMATRIX DZMTRXV	Output of $D_{zmtx}$ values	x	
DZNET	Output of net DZ values		x
FRACTURE	Output of fracture propagation data (fracture modeling only)	x	
HEATCR	Output of rock heat capacity		x
HEATCRT	Output of temperature dependence of the rock heat capacity		x
HEATTX	Output of X direction Thermal transmissibilities		x
HEATTY	Output of Y direction Thermal transmissibilities		x
HEATZ	Output of Z direction Thermal transmissibilities		x
IONROCK	Rock ion exchange capacity	x	
LX	Output of matrix block size in X direction (Viscous Displacement Option)	x	
LY	Output of matrix block size in Y direction (Viscous Displacement Option)	x	
LZ	Output of matrix block size in Z direction (Viscous Displacement Option)	x	
MIDS	Output of midpoint depths		x
MINPVV	Output of cells with pore volume less than the input minimum values	x	x
MULTPV	Output of the pore volume multipliers	x	x
MULTX	Output of X direction transmissibility multipliers	x	x
MULTY	Output of Y direction transmissibility multipliers	x	x
MULTZ	Output of Z direction transmissibility multipliers	x	x
MULTX-	Output of X- direction transmissibility multipliers	x	x
MULTY-	Output of Y- direction transmissibility multipliers	x	x
MULTZ-	Output of Z- direction transmissibility multipliers	x	x
NTG	Output of grid block net-to-gross ratios	x	x
PERMX	Output of X direction permeabilities	x	x
PERMY	Output of Y direction permeabilities	x	x

Mnemonic	Output	E100	E300
PERMZ	Output of Z direction permeabilities	x	x
PORO	Output of grid block porosities	x	x
PORV	Output of grid block pore volumes	x	x
PORVF	Output of grid block fluid pore volumes (Coal Bed Methane option)	x	x
ROCKFRAC	Output of grid block rock volume fractions (Coal Bed Methane option)	x	x
ROCKDEN	Output of rock density		x
ROCKV	Output of rock volumes		x
SIGMAGDV	Output of SIGMAGD values (gravity drainage option)	x	
SIGMAV	Output of Sigma values	x	
SIGMA*K	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.	x	
THCGAS	Output of Thermal conductivity for gas phase		x
THCOIL	Output of Thermal conductivity for oil phase		x
THCONR	Output of thermal rock and fluid conductivities (For ECLIPSE 300 only, this header will be renamed THCAVE, similar to RPTGRID keyword)	x	x
THCONSF	Output saturation dependence of conductivity		x
THCSOLID	Output of Thermal conductivity for solid phase		x
THCWATER	Output of Thermal conductivity for water phase		x
TOPS	Output of grid block top depths	x	x
TRANX	Output of X direction transmissibilities	x	x
TRANX	Output of Y direction transmissibilities	x	x
TRANZ	Output of Z direction transmissibilities	x	x

Table 3.33: RPTINIT output controls

## Example

```
RPTINIT
DZ TRANX PORO PERMY /
```

## RPTISOL

## Generates a grid array of isolated reservoir numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
x	SCHEDULE

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

```
RPTONLY
```

## RPTONLYO Turns off RPTONLY restricted summary output

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
x	SCHEDULE

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 RPTONLYO 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 RPTONLYO (or RPTONLY) keyword must be specified at or after the restart time. Any RPTONLYO (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 RPTONLYO keyword has no associated data.

### Example

```
RPTONLYO
```

## RPTOPT

## Optimization reporting options

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE
x	OPTIMIZE

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.

Mnemonic	Output
SEPARATE	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.
CLRFILES	Removes SEPARATE results files from any previous optimization for this model <i>if necessary</i> (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).

Table 3.34: RPTOPT output controls

## Example

```
RPTOPT
-- Output separate summary and restart files for each
-- optimization iteration, and clear previous results files
'SEPARATE' 'CLRFILES' /
```

## RPTPRINT

## Control printed simulation reports

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

1. Report step summary.  
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
3. Report fluid in place by region.  
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
6. Report status of wells.  
DEFAULT: 1
7. Report status of well completions.  
DEFAULT: 1
8. Report solution data, as specified by mnemonics in [RPTSCHED](#).  
DEFAULT: 1
9. Report convergence of non-linear equations.  
DEFAULT: 0
10. Report well potentials.  
DEFAULT: 0
11. Report inter-region flows.  
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
12. Report component flows.  
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

13. Report aquifer flows.

DEFAULT: 0

14. Report heater connections in the Thermal option.

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

Mnemonic	Output	E100	E300
ALSURFST	Outputs table of water-oil surface tension multipliers as a function of alkaline concentration		x
AQUTAB	Outputs the Carter-Tracy influence tables	x	
DENSITY, GRAVITY, SDENSITY, BDENSITY	Output of surface densities/gravities	x x x	x x x
DIFFC or DIFFCOAL	Output of the diffusion data	x	
DIFFCBM	Output of diffusion data (Coal Bed Methane)		x
DETAILVD	Output of the DETAILVD tables		x
ENDPT	Output of endpoint scaling values	x	
ENDPTVD	Output of endpoint versus depth tables		x
EPSODD3P	Output of end-point scaling control table for the ODD3P option		x
ESPNOE	Output of effective salinity nodes for Polymer Flood Model		x
ESSNOE	Output of effective salinity nodes for Surfactant Model		x
FHERCHBL	Output of Herschel-Bulkley data (fracture modeling only)	x	
FOAM	Output of the foam properties	x	
FRACMOVE	Output of fracture propagation (fracture modeling only)	x	
GINODE	Output of Gi-model saturated fluid properties	x	
HSGRO	Output of hysteresis gas residual end-point saturation for gas-oil system for the ODD3P option		x

Mnemonic	Output	E100	E300
HSGRW	Output of hysteresis gas residual end-point saturation for gas-water system for the ODD3P option		x
HSORG	Output of hysteresis oil residual end-point saturation for oil-gas system for the ODD3P option		x
HSORW	Output of hysteresis oil residual end-point saturation for oil-water system for the ODD3P option		x
HSWRG	Output of hysteresis water residual end-point saturation for water-gas system for the ODD3P option		x
HSWRO	Output of hysteresis water residual end-point saturation for water-oil system for the ODD3P option		x
IKRWH	Output of water relative permeabilities at maximum water saturation for WAG hysteresis three-phase water relative permeabilities when end-point scaling has been specified.	x	x
IKRWRH	Output of water relative permeabilities at residual oil for WAG hysteresis three-phase water relative permeabilities when end-point scaling has been specified.	x	x
KRG	Output of maximum gas relative permeabilities	x	x
KRGR	Output of gas relative permeabilities at residual	x	x
KRO	Output of maximum oil relative permeability		x
KROR	Output of oil relative permeabilities at residual		x
KRW	Output of maximum water relative permeabilities	x	x
KRWH	Output of water relative permeabilities at maximum water saturation for WAG hysteresis two-phase water relative permeabilities when end-point scaling has been specified.	x	x
KRWR	Output of water relative permeabilities at residual	x	x
KRWRH	Output of water relative permeabilities at residual oil for WAG hysteresis two-phase water relative permeabilities when end-point scaling has been specified.	x	x
LANGMEXT	Output of the extended Langmuir Isotherms data		x
LANGMPL	Output of the pressure scaling factor for the adsorption capacity for each grid cell.	x	x
LANGMULT	Output of the Langmuir multipliers		x
LANGMUIR, LANGSOLV, COALADS	Output of the Langmuir Isotherms	x	
LEAKOFF	Output of fracture fluid loss (fracture modeling only)	x	
MLANG	Output of the maximum surface gas concentration	x	
MLANGSLV	Output of the maximum surface solvent concentration	x	
MISC, PMISC	Output of the miscibility function and the pressure miscibility function	x	
OVERBURD	Overburden pressure data		x

Mnemonic	Output	E100	E300
PCG	Output of scaled maximum gas capillary pressures (provided data is supplied via the <a href="#">PCG</a> keyword), or the scaling factor for the <a href="#">JFUNC</a> keyword	x	x
PCODD3P	Output of capillary pressure calculation selection tables for the ODD3P option		x
PCODD3PG	Output of capillary pressure calculation gas saturation function for the ODD3P option		x
PCODD3PW	Output of capillary pressure calculation water saturation function for the ODD3P option		x
PCW	Output of scaled maximum water capillary pressures (provided data is supplied via the <a href="#">PCW</a> keyword), or the scaling factor for the <a href="#">JFUNC</a> keyword	x	x
PLYADS, PLYMAX, PLYROCK, PLYSHEAR, PLYDHFLF, PLYVISC, PLYVISCs, PLYVISCT, PLYVSCST, SALTNODE, TEMPNODE, PLMIXPAR	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 <a href="#">PLYKRRF</a> , <a href="#">PLYRMDEN</a> and <a href="#">PLYCAMAX</a> .	x	
PLYVISC	Output of water phase viscosity multipliers supplied by the <a href="#">PLYVISC</a> keyword		x
PLYVISCs	Output of water phase viscosity multipliers supplied by the <a href="#">PLYVISCs</a> keyword		x
PSGRO	Output of primary gas residual end-point saturation for gas-oil system for the <a href="#">ODD3P</a> option		x
PSGRW	Output of primary gas residual end-point saturation for gas-water system for the <a href="#">ODD3P</a> option		x
PSORG	Output of primary oil residual end-point saturation for oil-gas system for the <a href="#">ODD3P</a> option		x
PSORW	Output of primary oil residual end-point saturation for oil-water system for the <a href="#">ODD3P</a> option		x
PSWRG	Output of primary water residual end-point saturation for water-gas system for the <a href="#">ODD3P</a> option		x
PSWRO	Output of primary water residual end-point saturation for water-oil system for the <a href="#">ODD3P</a> option		x
PVDG, PVTG	Output of gas PVT tables	x	
PVDO, PVTO	Output of oil PVT tables	x	x
PVDS	Output of Solvent PVT properties	x	

Mnemonic	Output	E100	E300
PVTW	Output of water PVT tables	x	x
PVTWSALT	Output of brine properties	x	x
ROCK	Output of rock properties	x	
ROCKTAB	Output of rock properties	x	x
ROCKTABH	Output of rock properties (Hysteretic option only)	x	x
SGCR	Output of scaled critical gas saturations	x	x
SGFN	Output of gas saturation tables	x	x
SGF3	Output of gas saturation tables for the <a href="#">IKU3P</a> and <a href="#">ODD3P</a> options. Output of three-phase gas relative permeability table for the <a href="#">IKU3P</a> option.		x
SGL	Output of scaled connate gas saturations	x	x
SGU	Output of scaled maximum gas saturations	x	x
SOF2, SOF3, TLMIXPAR, SORWMIS	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.	x	x
SOGCR	Output of scaled critical oil-in-gas saturations	x	x
SOWCR	Output of scaled critical oil-in-water saturations	x	x
SOLU	Output of aqueous phase property data ( <a href="#">CO2SOL</a> and <a href="#">GASSOL</a> options)		x
SPECHEAT, VISCVT	Output of the temperature-dependent properties.	x	
SSFN	Output of Solvent/Gas relative permeabilities.	x	
STOG, STOW	Output of Surface Tension properties.	x	
STONE1EX	Output of Stone 1 exponent values.	x	x
SURF	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).	x	
SURFSTES	Outputs table of water-oil surface tension as function of surfactant concentration and effective salinity		x
SWCR	Output of scaled critical water saturations.	x	x
SWFN	Output of water saturation tables.	x	x
SWF3	Output of water saturation tables for the <a href="#">IKU3P</a> and <a href="#">ODD3P</a> options. Output of three-phase water relative permeability table for the <a href="#">IKU3P</a> option.		x
SWL	Output of scaled connate water saturations.	x	x

Mnemonic	Output	E100	E300
SWU	Output of scaled maximum water saturation.	x	x
TRACER	Output of passive fluid tracer names.	x	
TRADS, TRDCY, TRROCK	Output of environmental tracer properties.	x	
VDKRG	Velocity dependent relative permeability data for gas.		x
VDKRO	Velocity dependent relative permeability data for oil.		x
VEFRACV, VEFRAC	Output of Vefrac values.	x	

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 /
```

# RPTREGS Controls on output from REGIONS section

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

mnemonic=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

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.

Mnemonic	Output	E100 Integer Control	E300
ALPHANUM	Output of ALPHA region numbers		x
DSTNUM	Output of tertiary hysteresis saturation region numbers for the <a href="#">ODD3P</a> option		x
ENDNUM	Output of endpoint v depth region numbers	19	x
EOSNUM	Output of equation of state region numbers		x
EQLNUM	Output of equilibration region numbers	3	x
FIPNUM	Output of fluid in place region numbers	4	x
FIPOWG	Output original oil, water and gas zones	28	
FRACNUM	Output of fracture region numbers (fracture modeling only)	31	
HMxxxxxx	Output of gradient parameter regions when the Gradient Option is in use.  (If set >1, the parameter regions will be written to the Initial file when the INIT keyword is specified, this only applies to ECLIPSE 100).	25	x
HWSNUM	Output of (high salinity) water-wet saturation region numbers when the <a href="#">SURFACTW</a> option has been specified. This keyword is an alias for the <a href="#">SURFNUM</a> keyword.	x	

Mnemonic	Output	E100 Integer Control	E300
IMBNUM	Output of imbibition region numbers	11	x
IMBNUMMF	Output of matrix/fracture imbibition regions	23	
IMBNUMX	Output of X direction imbibition region numbers	12	x
IMBNUMX-	Output of -X direction imbibition region numbers	13	x
IMBNUMY	Output of Y direction imbibition region numbers	14	x
IMBNUMY-	Output of -Y direction imbibition region numbers	15	x
IMBNUMZ	Output of Z direction imbibition region numbers	16	x
IMBNUMZ-	Output of -Z direction imbibition region numbers	17	x
ISTNUM	Output of secondary hysteresis saturation region numbers for the ODD3P option		
LSNUM	Output of low salinity (oil-wet) saturation region numbers when the LOWSALT option has been specified. This keyword is an alias for the LWSLTNUM keyword.	x	
LWSLTNUM	Output of low salinity (oil-wet) saturation region numbers when the LOWSALT option has been specified. This keyword is an alias for the LSNUM keyword.	x	
LWSNUM	Output of low salinity water-wet saturation region numbers when the SURFACTW and the LOWSALT options have been specified. This keyword is an alias for the LSLTWNUM keyword.	x	
LSLTWNUM	Output of low salinity water-wet saturation region numbers when the SURFACTW and the LOWSALT options have been specified. This keyword is an alias for the LWSNUM keyword.	x	
KRNUMMF	Output of matrix/fracture saturation regions	22	
KRNUMX	Output of X direction $K_r$ region numbers	5	x
KRNUMX-	Output of -X direction $K_r$ region numbers	6	x
KRNUMY	Output of Y direction $K_r$ region numbers	7	x
KRNUMY-	Output of -Y direction $K_r$ region numbers	8	x
KRNUMZ	Output of Z direction $K_r$ region numbers	9	x
KRNUMZ-	Output of -Z direction $K_r$ region numbers	10	x
MISCNUM	Output of miscibility region numbers	21	x
PMANUM	Pressure maintenance region numbers		x
PSTNUM	Output of primary saturation region numbers for the ODD3P option		x
PVTNUM	Output of PVT region numbers	1	x
RESIDNUM	Output of VE residual flow region numbers	26	
ROCKNUM	Output of Rock Compaction region numbers	18	x
SATNUM	Output of saturation region numbers	2	x

Mnemonic	Output	E100 Integer Control	E300
SDRGO	Output of gas phase saturation direction indicators for gas-oil system for the ODD3P option		x
SDRGW	Output of gas phase saturation direction indicators for gas-water system for the ODD3P option		x
SDROG	Output of oil phase saturation direction indicators for oil-gas system for the ODD3P option		x
SDROW	Output of oil phase saturation direction indicators for oil-water system for the ODD3P option		x
SDRWG	Output of water phase saturation direction indicators for water-gas system for the ODD3P option		x
SDRWO	Output of water phase saturation direction indicators for water-oil system for the ODD3P option		x
SURFNUM	Output of the surfactant saturation regions	24	
SURFWNUM	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.	27	
WH2NUM	Output of two-phase water relative permeability saturation region numbers for the WAGHYSTR option.	x	x
WH3NUM	Output of three-phase water relative permeability saturation region numbers for the WAGHYSTR option.	x	x
TNUM	Output of tracer versus depth region numbers	20	
TRACKREG	Output of enhanced TRACK region numbers		x

Table 3.36: RPTREGS output controls

**Note:** For ECLIPSE 100, LWSLTNUM and its alias LSNUM, SURFWNUM and its alias HWSNUM, and LSLTWNUM 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

```
RPTREGS
  FIPNUM ENDNUM SATNUM /
```

### Example 2 (ECLIPSE 100)

The old integer format:

```
RPT5*0 1 /
```

REGS  
1 1 1 5

# RPTRST

## Controls on output to the RESTART file

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.

Mnemonic	Output	E100	E300
ACIP	Output pressure-wave impedance (Petro-elastic Model only): this produces an array with the name ACOUIMPP  If you request this item, you must also request the DEN item.  These values are only output on the global grid cells (that is, not on local grid refinement cells).	x	

Mnemonic	Output	E100	E300
ACIS	Output shear-wave impedance (Petro-elastic Model only): this produces an array with the name ACOUIMPS  If you request this item, you must also request the DEN item.  These values are only output on the global grid cells (that is, not on local grid refinement cells).	x	
AIM	Output AIM status array (1 for explicit cells; 2 for implicit cells)		x
ALSURF	Output alkaline concentration when surfactant and alkaline models are activated.		x
ALSTML	Output alkaline surface tension multiplier when surfactant and alkaline models are activated.		x
ALLPROPS	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 (1OVERBO, 1OVERBG and 1OVERBW) and phase relative permeabilities (OILKR, GASKR and WATKR).  If set =2 the old output mnemonic names (for example 1 / FVFOIL instead of 1OVERBO) are used.	x	
AMF	Output of aqueous mole fractions (CO2SOL, GASSOL, THERMAL options KWTABTn / CO2STORE and the COMPW option)		x
AQSP	Output of aqueous speciation (CO2STORE option)		x
AQPH	Output of aqueous pH level (CO2STORE option)		x
AREAC	Asphaltene reaction rates		x
ASPADS	Asphaltene cumulative adsorption volume fraction (pre-2011.1 model only)		x
ASPDOT	Asphaltene net cumulative volume fraction deposit (pre-2011.1 model only)		x
ASPENT	Asphaltene instantaneous entrainment volume fraction (pre-2011.1 model only)		x
ASPFLO	Oil velocities (useful to check the asphaltene entrainment) (Asphaltene pre-2011.1 model only)		x
ASPFLT	Average oil flow rate at the previous timestep (Asphaltene 2011.1 model only)		x
ASPFRD	Asphaltene precipitate fraction dissolved		x
ASPKDM	Asphaltene permeability multiplier (damage) (Asphaltene pre-2011.1 model only)		x
ASPLIM	Asphaltene precipitate fraction limit		x
ASPLUG	Asphaltene instantaneous plugging volume fraction (Asphaltene pre-2011.1 model only)		x
ASPRET	Asphaltene precipitate fraction (including flocs)		x

Mnemonic	Output	E100	E300
ASPREW	Asphaltene precipitate fraction (no flocs)		x
ASPVEL	Average oil velocity at the previous timestep (Asphaltene 2011.1 model only)		x
ASPVOM	Asphaltene relative oil viscosity multiplier		x
BASIC	<p>Output of basic Restart files</p> <ul style="list-style-type: none"> <li>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).</li> <li>If set to 2, restart files are created at every report time until this switch is reset, and all are kept.</li> <li>If set to 3, restart files are created every <math>n^{\text{th}}</math> report time. The frequency of the restarts is governed by the mnemonic "FREQ=<math>n</math>".</li> <li>If set to 4, a Restart file is written at the first report step of each year. Optionally, if the mnemonic FREQ is set <math>&gt;1</math> the restart is written only every <math>n^{\text{th}}</math> year.</li> <li>If set to 5, a Restart file is written at the first report step of each month. Optionally, if the mnemonic FREQ is set <math>&gt;1</math> the restart is written only every <math>n^{\text{th}}</math> month.</li> <li>If set to 6, a Restart file is written at every timestep.</li> </ul> <p><b>Notes:</b></p> <p>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 <b>LOAD</b> or <b>RESTART</b> 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.</p> <p>ECLIPSE 300 assumes the case 'BASIC = 2' if the keyword BASIC is used. To subsequently turn off Restarts, use 'BASIC=0'.</p> <p>Value 1 is not available with ECLIPSE 300.</p>	x	x
BFORO	<p>Output of oil Forchheimer B-Factor / phase potential difference (as described in <a href="#">Construction of flows</a> in the <i>ECLIPSE Technical Description</i>). This is only output if VDFLOW, VDFLOWR or VELDEP has been used to activate a non-Darcy flow model.</p> <p>Note that this is not the Forchheimer beta parameter.</p>		x
BG	Gas formation volume factor	x	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

Mnemonic	Output	E100	E300
BSOL	Output of solid reservoir molar densities ( <a href="#">SOLID</a> option)		x
BTFORG	Output gas Forchheimer beta parameter		x
BTFORO	Output oil Forchheimer beta parameter		x
BW	Water formation volume factor	x	x
BWAT	Output of water reservoir molar densities		x
CELLINDX	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. <ul style="list-style-type: none"> <li>The numbering on the global grid is from 1 to NX*NY*NZ, with the numbering cycling fastest in the X, then Y, then Z directions.</li> <li>The numbering on each local grid is from 1 to LX*LY*LZ, with the numbering cycling fastest in the X, then Y, then Z directions.</li> </ul>	x	x
CFL	Outputs CFL numbers (see <a href="#">CFLLIMIT</a> keyword)		x
CGAS	Output of grid-block adsorbed gas concentration (for the Coal Bed Methane option)		x
COHESION	Geomechanics: outputs cohesion (plasticity)		x
COLR	Output color of sub-grids (see " <a href="#">The ECLIPSE 300 WARP linear solver</a> " in the of the <i>ECLIPSE Technical Description</i> )		x
COILR	Residual oil compressibility ( <a href="#">SOR</a> option)		x
COMPRESS	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.	x	
CONV	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). <b>Note:</b> ECLIPSE 100 outputs various convergence problem indicators as described in " <a href="#">Output of cells with convergence problems</a> " in the <i>ECLIPSE Technical Description</i> . ECLIPSE 300 outputs various convergence problem indicators as described in " <a href="#">Output of cells with convergence problems</a> " in the <i>ECLIPSE Technical Description</i> .	x	x
DEN	Output of phase reservoir densities	x	
DENG	Output of gas mass densities at reservoir conditions		x
DENO	Output of oil mass densities at reservoir conditions		x
DENS	Output of solid mass densities ( <a href="#">SOLID</a> option) at reservoir conditions		x
DEFFSTRS	Delta effective stresses = change from initial conditions: this produces output with the names DEFFSTXX, DEFFSTXY, DEFFSTXZ, DEFFSTYY, DEFFSTYZ, DEFFSTZZ.		x

Mnemonic	Output	E100	E300
DENW	Output of water mass densities at reservoir conditions		x
DPRSTRES	Delta principal stresses = change from initial conditions; this produces output with the names DPRSTRS1, DPRSTRS2, DPRSTRS3 followed by their respective directions		x
DRAIN(AGE)	Output of the drainage regions as DRAINAGE.  If set >1, an additional array, DRAINMIN, is output marking the sinks (typically wells) in the drainage region calculation.  (These may also be output to the Debug file - See <a href="#">item 25</a> in the OPTIONS keyword.)	x	
DROCKDIS	Geomechanics: outputs Delta rock displacements = change from initial conditions: DROCKDIX, DROCKDIY, DROCKDIZ. These values are interpolated to the centers of the grid blocks.		x
DSTRAIN	Geomechanics: outputs Delta strain components = change from initial conditions: DSTRAINXX, DSTRAINXY, DSTRAINXZ, DSTRAINYY, DSTRAINYZ, DSTRAINZZ.		x
DTOTSTRS	Geomechanics: outputs Delta total stresses = change from initial conditions: DTOTSTXX, DTOTSTXY, DTOTSTXZ, DTOTSTYY, DTOTSTYZ, DTOTSTZZ.		x
DYNREG	Output of Dynamic Regions	x	x
EFFSTRES	Geomechanics: outputs rock stresses: EFFSTRXX, EFFSTRXY, EFFSTRXZ, EFFSTRYY, EFFSTRYZ, EFFSTRZZ.		x
ENERGY	Internal energy / bulk volume ( <a href="#">THERMAL</a> option)		x
ESALTS	Effective salinity for the Surfactant Model calculated as specified by the <a href="#">SALTEFF</a> keyword		x
ESALTP	Effective salinity for the Polymer Flood Model calculated as specified by the <a href="#">SALTEFF</a> keyword		x
FFACTG	$FFACTG = FACTLI \times T_{crit}^{mix} / T_{reservoir}$  where $T_{crit}^{mix}$ is defined in the FACTLI keyword,  $Z_j$ used in the FACTLI definition is the gas phase mole fraction		x
FFACTO	$FFACTO = FACTLI \times T_{crit}^{mix} / T_{reservoir}$  where $T_{crit}^{mix}$ is defined in the FACTLI keyword,  $Z_j$ used in the FACTLI definition is the oil phase mole fraction		x
FFORO	Forchheimer Non-Darcy flow Factor for Oil ( <a href="#">VELDEP</a> option)		x

Mnemonic	Output	E100	E300
FIP	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.	x	x
FLOCn	Output of inter-block component flows (for component n, where n is an integer)		x
FLOE	Output of inter-block energy flow (ECLIPSE 300 Thermal option)		x
FLOGAS	Output of inter-block gas flows		x
FLOOIL	Output of inter-block oil flows		x
FLOWAT	Output of inter-block water flows		x
FLWS	Output of interblock flows  (Includes non-neighbor connection flows and flows between global and local grids.)	x	
FLORES	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+.	x	x
FMISC	Output of relative permeability miscibility factor (Miscible option)		x
FOAM	Output of foam surfactant density		x
FOAMST	Output of oil-gas surface tension for the Foam Model		x
FOAMCNM	Output of capillary number for the Foam Model		x
FOAMMOB	Output of gas mobility multiplier for the Foam Model		x
FPC	Output of capillary pressure miscibility factor (Miscible option)		x
FREQ=n	Controls the frequency of the restart files if BASIC is set >2	x	x
FUGG	Output of component fugacities in gas (Activity Driven Diffusion)		x
FUGO	Output of component fugacities in oil (Activity Driven Diffusion)		x
GASPOT	Output of gas potentials (or “depth corrected pressures”)		x
GENPLSTN	Geomechanics: outputs generalized plastic strains		x
GENPSTRS	Geomechanics: outputs stiffness matrix multiplied by plastic strain.		x
GIMULT	Outputs RSGI, RVGI, BOGI and BGGI values	x	
GOSTRES	Geomechanics: outputs initial rock stresses induced by gravity and overburden: GOESTRXX, GOESTRXY, GOESTRZX, GOESTRYY, GOESTRZY.		x
HGAS	Output gas enthalpy values (ECLIPSE 300 Thermal)		x
HOIL	Output oil enthalpy values (ECLIPSE 300 Thermal)		x

Mnemonic	Output	E100	E300
HSOL	Output solid enthalpy values (ECLIPSE 300 Thermal and <a href="#">SOLID</a> option)		x
HWAT	Output water enthalpy values (ECLIPSE 300 Thermal)		x
HYDH	Output hydraulic head. (Reference data must be supplied in item 1 of the <a href="#">HYDRHEAD</a> keyword.)	x	
HYDHFV	Output freshwater hydraulic head. (Reference data must be supplied in items 1 and 2 of <a href="#">HYDRHEAD</a> keyword.)	x	
JV	Output of ln(K-Value) values		x
KRG	Output of gas relative permeabilities: outputs the array GASKR (E100) or KRG (E300)	x	x
KRO	Output of oil relative permeabilities: outputs the array OILKR (E100) or KRO (E300)	x	x
KRW	Output of water relative permeabilities: outputs the array WATKR (E100) or KRW (E300)	x	x
KRGDM	Output of gas relative permeability damage multiplier		x
KRODM	Output of oil relative permeability damage multiplier		x
KRWDM	Output of water relative permeability damage multiplier		x
LGLCn	Output of number of <a href="#">LAGTRAC</a> streamlines generated for component n to pass through a cell		x
LGLCWAT	Output of number of <a href="#">LAGTRAC</a> streamlines generated for water to pass through a cell		x
LGLCHC	Output of number of <a href="#">LAGTRAC</a> streamlines generated for hydrocarbons (that is to solve <a href="#">LUMPING</a> tracers) to pass through a cell		x
MLSC	Output of total molar density for hydrocarbon components (moles per reservoir volume)		x
MWAT	Output of total molar density for water components (moles per reservoir volume)		x
NCNG	Output of capillary numbers for gas ( <a href="#">VELDEP</a> option). The normalized capillary number is output when used with keyword <a href="#">VDKRG</a> .		x
NCNO	Output of capillary numbers for oil ( <a href="#">VELDEP</a> option). The normalized capillary number is output when used with keyword <a href="#">VDKRO</a> .		x
NOGRAD	Output of total derivatives for the Gradient option  If set > 0, extra information is not written to the Restart file for the Gradient option.	x	

Mnemonic	Output	E100	E300
NORST	<p>Requests a graphics only Restart file.</p> <ul style="list-style-type: none"> <li>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).</li> <li>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).</li> </ul> <p>The restart files dumped while this control is set &gt; 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 <a href="#">RESTART</a>).</p>	x	
NPMREB	<p>Output of the number of passes through the rebound region during the calculations of the pore volumes of the Palmer-Mansoori rock model (keyword <a href="#">ROCKPAMA</a> or <a href="#">ROCKPAME</a>). Also gives the sign of the derivatives during the last calculation (positive &gt;=0). Requires <a href="#">CBMOPTS</a> item 4 to be set to PMREB.</p>	x	x
OILPOT	Output of oil potentials (or “depth corrected pressures”)		x
PART	Output of domain partitions (Parallel option)		x
PBPD	Output of bubble point and dew point pressures	x	
PCGW	Output of gas-water capillary pressures ( <a href="#">GASWAT</a> cases)		x
PCOG	Output of gas-oil capillary pressures	x	x
PCOW	Output of oil-water capillary pressures	x	x
PERMSTAB	Geomechanics: outputs permeability multipliers (stress dependence)		x
PERM_MDX PERM_MDY PERM_MDZ	<p>Output directional permeability multipliers (Rock Compaction option) when <a href="#">RKTRMDIR</a> or <a href="#">ROCKTRMX</a>, <a href="#">ROCKTRMY</a> and <a href="#">ROCKTRMZ</a> are defined in ECLIPSE 300.</p> <p>This information can be output in ECLIPSE 100 via the <a href="#">ROCKC</a> mnemonic (described below).</p>		x
PERM_MOD	<p>Output permeability multipliers (Rock Compaction option).</p> <p>This information can be output in E100 via the <a href="#">ROCKC</a> mnemonic (described below).</p>		x
PGAS	Output of gas phase pressure		x
PGOSTRES	Geomechanics: outputs initial principal stresses induced by gravity and overburden: <a href="#">GVPRSTS1</a> , <a href="#">GVPRSTS2</a> , and <a href="#">GVPRSTS3</a> .		x
PKRG	Processed gas relative permeability ( <a href="#">ENKRVC</a> and <a href="#">ENKRVT</a> cases)		x
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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRO	Processed oil relative permeability ( <a href="#">ENKRVC</a> and <a href="#">ENKRVT</a> cases)		x

Mnemonic	Output	E100	E300
PKRORG	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 ENKRVC and ENKRVT keywords respectively.		x
PKRORW	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 ENKRVC or ENKRVT keywords respectively.		x
PKRW	Processed water relative permeability (ENKRVC and ENKRVT cases)		x
PKRWR	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 ENKRVC or ENKRVT keywords respectively.		x
PLASDISP	Geomechanics: outputs plastic displacements		x
POIL	Output of oil phase pressure		x
POIS	Output Poisson's ratio (Petro-elastic Model only): this produces an array with the name POISSONR  If you request this item, you must also request the DEN item  These values are only output on the global grid cells (that is, not on local grid refinement cells).	x	
POLY	Output of polymer concentration (Polymer option)		x
POLYVM	Output of polymer viscosity multiplier (Polymer option)		x
PORO	Output of effective porosities (Gradient Option only)	x	
PORV	Output of reservoir pore volumes at reference conditions	x	x
POT	Output of phase potentials (or "depth corrected pressures"): this produces the arrays OIL_POTN, GAS_POTN and WAT_POTN.  (Including the initial contact corrected potential output to the array POT_CORR.)	x	
PORV_MOD	Output pore volume multipliers (Rock compaction option)  This information can be output in E100 via the ROCKC mnemonic (described below).		x
PPCG	Output of processed capillary pressure gas-oil end-points		x
PPCW	Output of processed capillary pressure oil-water end-points		x
PRES_EFF	Output of effective fluid pressures for rock compaction (when <a href="#">OVERBURD</a> and <a href="#">ROCKOPTS</a> are specified).		x
PRES	ECLIPSE 100 Output of water and gas phase pressures  ECLIPSE 300 Output of grid block pressures	x	x
PRESMIN	Output of minimum pressures (Irreversible rock compaction option)		x
PRESSURE	Output of grid block pressures (equivalent to PRES)		x

Mnemonic	Output	E100	E300
PRSTRESS	Geomechanics: outputs principal stresses: PRSTRES1, PRSTRES2, PRSTRES3 followed by their respective directions.		x
PSAT	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.		x
PSGCR	Output of the processed scaled critical gas saturation end-points for each grid cell if composition or temperature-dependent end-point scaling selected via the use of the ENPTVC or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.		x
PSGL	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.		x
PSGU	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.		x
PSOGCR	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.		x
PSOWCR	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.		x
PSWCR	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.		x
PSWL	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.		x
PSWU	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.		x
PTCSTRES	Geomechanics: outputs initial principal stresses induced by tectonic loadings: T CPRSTS1, T CPRSTS2 and T CPRSTS3.		x
PVDPH	Derivative of pore volume with respect to porosity		x
PWAT	Output of water phase pressure		x
RATP	Ratio of actual to target throughputs		x

Mnemonic	Output	E100	E300
RATS	Ratio of actual to target solution changes		x
RATT	Ratio of actual to target temperature changes (Thermal option only)		x
REAC	Chemical reaction rates (See " <a href="#">Chemical Reactions</a> " in the <i>ECLIPSE Technical Description</i> )		x
RESTART	Output of data required for flexible restart		x
RFIP	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.	x	x
RK or PERMREDN	Polymer Flood: Permeability reduction for the aqueous phase due to polymer retention in the rock.	x	
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 interpolated to the centers of the grid blocks.		x
ROMLS	Residual oil composition (SOR option)		x
RPORV	Output of reservoir pore volumes at Reservoir conditions	x	x
RS	Output dissolved GOR ( $R_s$ ) (In compositional runs this is interpreted as the GOR at surface conditions of the liquid phase in the reservoir)		x
RSSAT	Output of saturated $R_s$ values (dissolved gas-oil ratio).  Only in Blackoil mode for ECLIPSE 300.	x	x
RSW	Output of the aqueous concentration of soluble components. Each component is distinguished, as with mnemonic AMF, by its associated index.		x
RV	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.)		x
RVSAT	Output of saturated $R_v$ values (vaporized oil-gas ratio).  Only in Blackoil mode for ECLIPSE 300.	x	x

Mnemonic	Output	E100	E300
SAVE	<p>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.</p> <ul style="list-style-type: none"> <li>If set to 1 or 2, restart data is written to the Save file every report time until this switch is reset.</li> <li>If set to 3, restart data is written every <math>n^{\text{th}}</math> report time. The frequency of the restarts is governed by the mnemonic.</li> <li>If set to 4, restart data is written at the first report step of each year. Optionally, if SFREQ is set <math>&gt;1</math> the restart is written only every <math>n^{\text{th}}</math> year.</li> <li>If set to 5, restart data is written at the first report step of each month. Optionally, if SFREQ is set <math>&gt;1</math> the restart is written only every <math>n^{\text{th}}</math> month.</li> <li>If set to 6, this setting is reserved for use by FrontSim.</li> </ul> <p>Note that the Save file <b>must</b> be requested either by using the <a href="#">SAVE</a> keyword in the RUNSPEC section or by requesting a Save file in the <a href="#">LOAD</a> keyword, otherwise no restart data is output to the Save file.</p>	x	
SDENO	Output of grid block oil surface densities (for API tracking)	x	
SFIP	Output of fluids-in-place at surface/separator conditions (see " <a href="#">Initial fluid in place</a> " and item 201 of <a href="#">OPTIONS3</a> )	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 $S_{\text{gtrap}}$ 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 ( <a href="#">CO2STORE</a> option only)		x
SMMULT	Output of solid mobility multiplier ( <a href="#">SOLID</a> option)		x
SOIL	Output of grid block oil saturations		x
SOILM	Mobile oil saturation ( <a href="#">SOR</a> option)		x

Mnemonic	Output	E100	E300
SOILMAX	Output of maximum oil saturation (Hysteresis option)		x
SOILR	Residual oil saturation (SOR option)		x
SOLADS	Adsorbed solid concentration (Solid option)		x
SOLADW	Adsorbed wetting-solid concentration (Solid option)		x
SOLWET	Solid wettability interpolation parameter (Solid option)		x
SSFRACT	Output of suspended solids fraction (Solid option)		x
SSOLID	Output of grid block solid saturations (Solid option)		x
STATE	Output of states present		x
STEN	Output of oil/gas surface tensions (Miscible option)		x
STRAIN	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 " <a href="#">Elastic Stress Equations</a> " of the <i>ECLIPSE Technical Description</i> .		x
STREAM=n	Reserved for use by FrontSim	x	
SUBG	Output the sub-grid (see " <a href="#">The ECLIPSE 300 WARP linear solver</a> " in the <i>ECLIPSE Technical Description</i> )		x
SURF	Surfactant concentration - Surfactant option		x
SURFBLK	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.	x	
SURFCNM	log (capillary number) - Surfactant option		x
SURFKR	Relative permeability interpolation parameter - Surfactant option		x
SURFCP	Capillary pressure multiplier - Surfactant option		x
SURFST	Oil-water surface tension - Surfactant option		x
SWAT	Output of grid block water saturations		x
SWATMIN	Output of minimum water saturation (Hysteresis option)		x
TCBULK	Output of the average thermal conductivity (using the mnemonic THCAVE specified in the <a href="#">RPTGRID</a> keyword) multiplied by saturation dependent thermal transmissibility multiplier $\lambda$ ( <a href="#">THERMAL</a> option)		x
TCMULT	Saturation dependent thermal transmissibility multiplier $\lambda$ ( <a href="#">THERMAL</a> option)		x
TCSTRES	Geomechanics: outputs initial rock stresses induced by tectonic loadings: TCESTRXX, TCESTRXY, TCESTRXZ, TCESTRYY, TCESTRYZ, TCESTRZZ		x
TEMP	Output of reservoir temperatures ( <a href="#">THERMAL</a> option)		x
TOTCOMP	Outputs total compressibility of the fluid system (excluding the rock compressibility)		x

Mnemonic	Output	E100	E300
TOTSTRES	Geomechanics: outputs total stresses: TOTSTRXX, TOTSTRXY, TOTSTRXZ, TOTSTRYY, TOTSTRYZ, TOTSTRZZ.		x
TRAS	Output the product of tracer concentration with the carrier saturation	x	
TREACM	Tracer dependent reaction rate multipliers (See " <a href="#">Chemical Reactions</a> " in the <i>ECLIPSE Technical Description</i> )		x
TRMFxxxx	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.		x
TSUB	Output of the water saturation temperature minus the grid block temperature (THERMAL option)		x
VELGAS	Output of gas velocities  This gives phase velocity $u_p$ as defined in " <a href="#">ECLIPSE 100 Velocity calculations</a> " in the <i>ECLIPSE Technical Description</i> .	x	
VELOCITY	Output of all fluid velocities.  This gives phase velocity $u_p$ as defined in " <a href="#">ECLIPSE 100 Velocity calculations</a> " in the <i>ECLIPSE Technical Description</i> .  For polymer shear-thinning in the Polymer Flood Model, only water velocity is output.  For shear thinning in the Foam Model, only gas velocity is output.	x	
VELOIL	Output of oil velocities  This gives phase velocity $u_p$ as defined in " <a href="#">ECLIPSE 100 Velocity calculations</a> " in the <i>ECLIPSE Technical Description</i> .	x	
VELWAT	Output of water velocities  This gives phase velocity $u_p$ as defined in " <a href="#">ECLIPSE 100 Velocity calculations</a> " in the <i>ECLIPSE Technical Description</i> .	x	
VGAS	Output of gas viscosities		x

Mnemonic	Output	E100	E300
VISC	<p>Argument VISC allows to output extra viscosities when Polymer is active. You can obtain the following output:</p> <ul style="list-style-type: none"> <li>• EPVIS: effective polymer viscosity (Polymer option).</li> <li>• EMVIS: effective mixture (water/polymer) viscosity (Polymer option).</li> <li>• EWV_POL: effective water viscosity (Polymer option).</li> <li>• EWV_SAL: effective water viscosity due to salt concentration (Brine option).</li> <li>• PLADALK: polymer adsorption multiplier as a function of alkaline concentration.</li> <li>• SFIPPLY/SFIPPOL: polymer in place (mass)</li> <li>• RRFAC: residual resistance factor.</li> <li>• TRXBYRRF: I-direction transmissibility divided by the residual resistance factor.</li> <li>• TRYBYRRF: J-direction transmissibility divided by the residual resistance factor.</li> <li>• TRZBYRRF: K-direction transmissibility divided by the residual resistance factor.</li> <li>• 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).</li> <li>• POLYMAX: maximum polymer concentration.</li> <li>• RRFTMAX: RRF maximum temperature.</li> <li>• POLYMER: polymer concentration.</li> </ul>	x	
VOIL	Output of oil viscosities		x
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas)		x
VWAT	Output of water viscosities		x
WATPOT	Output of water potentials (or “depth corrected pressures”)		x
XFW	Output of liquid component mass fractions (weight fraction of each component in liquid)		x
XGAS	Output of gas fractional saturations (Sudation option)		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid)		x
XWAT	Output of water fractional saturations (Sudation option)		x
YFW	Output of vapor component mass fractions (weight fraction of each component in gas)		x

Mnemonic	Output	E100	E300
YMF	Output of vapor component mole fractions (fraction of each component in gas)		x
ZMF	Output of total component mole fractions		x

Table 3.37: RPTRST output controls

The following table lists the deprecated mnemonics.

Mnemonic	Output	E100	E300
PKRGE	This has been superseded by the mnemonic PKRGR (ENKRVC and ENKRVT cases).		x
PKROGE	This has been superseded by the mnemonic PKRORG (ENKRVC and ENKRVT cases).		x
PKROWE	This has been superseded by the mnemonic PKRORW (ENKRVC and ENKRVT cases).		x
PKRWE	This has been superseded by the mnemonic PKRWR (ENKRVC and ENKRVT cases).		x
SGCR	This has been superseded by the mnemonic PSGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SGL	This has been superseded by the mnemonic PSGL (ENKRVC and ENKRVT cases).		x
SGU	This has been superseded by the mnemonic PSGU (ENKRVC and ENKRVT cases).		x
SOGCR	This has been superseded by the mnemonic PSOGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SOWCR	This has been superseded by the mnemonic PSOWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWCR	This has been superseded by the mnemonic PSWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWL	This has been superseded by the mnemonic PSWL (ENKRVC and ENKRVT cases).		x
SWU	This has been superseded by the mnemonic PSWU (ENKRVC and ENKRVT cases).		x
WMF	The mnemonic has been superseded by AMF.		x

Table 3.38: Deprecated RPTRST output controls

## XMF/YMF output

### ECLIPSE 300

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 FLOC3I+ 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

BASIC=2	Additional Output
PRESSURE	Output of grid block pressures
RS	Output of dissolved GOR (Rs)
RV	Output of vaporized OGR (Rv)
SWAT	Output of grid block water saturations
SGAS	Output of grid block gas saturations
OILAPI	Output of grid block API values (for API tracking)

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,

	BASIC=2	Additional Output
Asphaltene 2011.1 model	ASPFLT	Average oil flow rate at the previous timestep
Asphaltene 2011.1 model	ASPVEL	Average oil velocity at the previous timestep
Residual oil model	COILR	Residual oil compressibility
Compositional models	LIFACT	Li Factors
	MLSC 1, 2, . . . , n	Hydrocarbon moles/reservoir volume for each component, 1,2,...,n (“total molar density”)
	MWAT	Water moles/reservoir volume for each water component (“total molar density of water”)

	<b>BASIC=2</b>	<b>Additional Output</b>
FREEZEPC or DPCDT options	PCOINIT	Initial oil capillary pressure
FREEZEPC or DPCDT options	PCWINIT	Initial water capillary pressure
FREEZEPC or DPCDT options	PCGINIT	Initial gas capillary pressure
Rock compaction options	PRESMIN	Minimum pressure
Rock compaction options	PRESMAX	Maximum pressure
Residual oil model	ROMOL	Residual oil moles
Sudation (GRAVDR) option	SGASINIT	Initial gas saturation
Hysteresis options (HYST)	SGASMAX	Maximum gas saturation
DPCDT option	SGT	Gas saturation at time t
HYST option	SOILMAX	Maximum oil saturation
HYST option	SOILMIN	Minimum oil saturation
Residual oil model	SOILRA	Initial residual oil saturation or residual oil volume, depending on which residual oil compressibility model is used
Asphaltene 2011.1 model	SOLIDT	Solid saturation at previous timestep
Sudation (GRAVDR) option	SWATINIT	Initial water saturation
Hysteresis options (HYST)	SWATMIN	Minimum water saturation
DPCDT option	SWT	Water saturation at time t

Table 3.40: Extra output for BASIC=2 in ECLIPSE 300

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.

#### ECLIPSE 300

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.

<b>BASIC=2</b>	<b>Additional output</b>
RKRG	Reference scaled maximum gas relative permeability for each cell.
RKRGR	Reference scaled gas relative permeability at the residual oil saturation for each cell.
RKRO	Reference scaled maximum oil relative permeability for each cell.
RKRORG	Reference scaled oil relative permeability at the residual gas saturation for each cell.
RKRORW	Reference scaled oil relative permeability at the residual water saturation for each cell.
RKRW	Reference scaled maximum water relative permeability.
RKRWR	Reference scaled water relative permeability at the residual oil saturation for each cell.
RPCG	Reference scaled maximum gas-oil capillary pressure for each cell.
RPCW	Reference scaled maximum water-oil capillary pressure for each cell.

<b>BASIC=2</b>	<b>Additional output</b>
RSOGCR	Reference scaled critical oil-in-gas saturation for each grid cell.
RSOWCR	Reference scaled critical oil-in-water saturation for each grid cell.
RSGCR	Reference scaled critical gas saturation for each grid cell.
RSGL	Reference scaled minimum gas saturation for each grid cell.
RSGU	Reference scaled maximum gas saturation for each grid cell.
RSWCR	Reference scaled critical water saturation for each grid cell.
RSWL	Reference scaled minimum water saturation for each grid cell.
RSWU	Reference scaled maximum water saturation for each grid cell.

Table 3.41: Extra output for BASIC=2 in ECLIPSE 300 for processed end-points.

## 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 /
```

## RPTRUNSP Controls output of RUNSPEC data

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## RPTSCHED Controls on output from SCHEDULE section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

Mnemonic	Output	E100	E300
AIM	Output of AIM status array (1 for implicit cells; 2 for explicit cells).		x
ALKALINE	Output of alkaline concentrations and alkaline adsorption when activated.	x	
ALSURF	Output alkaline concentration when surfactant and alkaline models are activated.		x
ALSTML	Output alkaline surface tension multiplier when surfactant and alkaline models are activated.		x
AMF	Output of aqueous mole fractions ( <a href="#">CO2SOL</a> , <a href="#">GASSOL</a> and <a href="#">THERMAL</a> options <a href="#">KWTABTn</a> / <a href="#">CO2STORE</a> and the <a href="#">COMPW</a> option).		x
ANIONS	Output of anion concentration.	x	
AQSP	Output of aqueous speciation ( <a href="#">CO2STORE</a> option).		x
AQPH	Output of aqueous pH level ( <a href="#">CO2STORE</a> option).		x

Mnemonic	Output	E100	E300
AQUCT or AQUFET or AQUFETP	Output of Fetkovich or Carter-Tracy aquifer status. Also outputs <a href="#">AQUCHWAT</a> , <a href="#">AQUCHGAS</a> and <a href="#">AQUFLUX</a> status.  Note: <a href="#">AQUFET</a> applies only for ECLIPSE 100.	x	x
AREAC	Asphaltene reaction rates.		x
ASPADS	Asphaltene cumulative adsorption volume fraction (pre-2011.1 model only).		x
ASPDOT	Asphaltene net cumulative volume fraction deposit (pre-2011.1 model only).		x
ASPENT	Asphaltene instantaneous entrainment volume fraction (pre-2011.1 model only).		x
ASPFLO	Oil velocities (useful to check the asphaltene entrainment) (pre-2011.1 model only).		x
ASPFLT	Average oil flow rate at the previous timestep (2011.1 model only).		x
ASPFRD	Asphaltene precipitate fraction dissolved.		x
ASPKDM	Asphaltene permeability multiplier (damage) (pre-2011.1 model only).		x
ASPLIM	Asphaltene precipitate fraction limit.		x
ASPLUG	Asphaltene instantaneous plugging volume fraction (pre-2011.1 model only).		x
ASPRET	Asphaltene precipitate fraction (including flocs).		x
ASPREW	Asphaltene precipitate fraction (no flocs).		x
ASPVEL	Average oil velocity at the previous timestep (2011.1 model only).		x
ASPVOM	Asphaltene relative oil viscosity multiplier.		x
BFORG	Output of gas Forchheimer B-Factor / phase potential difference (as described in <a href="#">Construction of flows</a> in the <i>ECLIPSE Technical Description</i> ). This is only output if <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> (or <a href="#">VELDEP</a> 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 <a href="#">Construction of flows</a> in the <i>ECLIPSE Technical Description</i> ). This is only output if <a href="#">VDFLOW</a> , <a href="#">VDFLOWR</a> or <a href="#">VELDEP</a> 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 ( <a href="#">SOLID</a> 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

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. <ol style="list-style-type: none"> <li>report is sent to Print file.</li> <li>in addition to being sent to the Print file, a copy of the report is sent to the short form output (terminal)</li> <li>report is sent to short form output only.</li> <li>as 2 but with actual cpu and elapsed times output in seconds.</li> </ol> <p>See "<a href="#">Timing Reports</a>" in the <i>ECLIPSE Technical Description</i> for an explanation of the outputs. Internal clocks within TS are not recorded during intermediate timesteps when set to 0.</p>	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

Mnemonic	Output	E100	E300
ESALSUR	Output of effective salinity for phase behavior and surfactant properties.	x	
FFORG	ECLIPSE 300: output of Forchheimer Non-Darcy flow factor for Gas (VELDEP option). ECLIPSE 100: Maximum deviation from Darcy Flow.	x	x
FFORO	Output of Forchheimer Non-Darcy flow factor for Oil (VELDEP option). ECLIPSE 100: not applicable yet.		x
FIP	Output of fluid in place and inter-region flow reports: 1. fluids in place are reported for the whole field. 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. 3. in addition, a balance sheet is produced for all sets of fluid-in-place regions defined with the FIP keyword.	x	x
FIPFOAM	Output of foam fluid in place reports (for the Foam Model): 1. gives a report for the whole field 2. in addition gives a report for each fluid in place region	x	
FIPRESV	Output of region reservoir volume reports.	x	
FIPSALT	Output of salt fluid in place reports (for the Brine option and salt-sensitive Polymer Flood option): 1. gives a report for the whole field 2. in addition gives a report for each fluid-in-place region	x	
FIPSOL	Output of the Solvent fluid in place reports: 1. gives a report for the whole field 2. gives a report for each fluid-in-place region	x	
FIPSURF	Output of surfactant fluid in place reports (for the Surfactant Model): 1. gives a report for the whole field 2. in addition gives a report for each fluid-in-place region	x	
FIPTEMP or FIPHEAT	Output of energy in place reports (for the Temperature option): 1. gives a report for the whole field 2. in addition gives a report for each fluid-in-place region	x	
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	

Mnemonic	Output	E100	E300
FIPVE	Output of fluid in place in original oil, water and gas zones (for the <a href="#">VE</a> option).	x	
FLOCn	Output of inter-block component flows (for component n, where n is an integer).		x
FLOE	Output of inter-block energy flow (ECLIPSE 300 Thermal option).		x
FLOGAS	Output of inter-block gas flows: NNC flows are always given in ECLIPSE 300 =3 gives NNC flows in ECLIPSE 100	x	x
FLOOIL	Output of inter-block oil flows: NNC flows are always given in ECLIPSE 300 =3 gives NNC flows in ECLIPSE 100	x	x
FLOSOL	Output of inter-block solvent flows.	x	
FLOWAT	Output of inter-block water flows: NNC flows are always given in ECLIPSE 300 =3 gives NNC flows in ECLIPSE 100	x	x
FMISC	Output of relative permeability miscibility factor (Miscible option).		x
FOAM	Output of grid block foam surfactant 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 oil-gas surface tension for foam.		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	
GOWAT	Output of gas displaced into oil and water zones: (for the <a href="#">VE</a> option) 1= by regions and field >1 = by grid blocks	x	
GENPLSTN	Geomechanics: outputs generalized plastic strains.		x
GENPSTRS	Geomechanics: outputs plastic strain multiplied by stiffness matrix.		x

Mnemonic	Output	E100	E300
GI	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.	x	
GMEG	Output gas phase energy mobility (ECLIPSE 300 Thermal).		x
GMEO	Output oil phase energy mobility (ECLIPSE 300 Thermal).		x
GMEW	Output water phase energy mobility (ECLIPSE 300 Thermal).		x
GOC	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	x	
GOCDIFF	Output of current G.=OC depth - initial GOC depth (for VE option).	x	
GRAD	Summary of gradient calculations for the Gradient option.	x	
HGAS	Output gas enthalpies (ECLIPSE 300 Thermal).		x
HOIL	Output oil enthalpies (ECLIPSE 300 Thermal).		x
HSOL	Output solid enthalpies (ECLIPSE 300 Thermal and SOLID option).		x
HWAT	Output water enthalpies (ECLIPSE 300 Thermal).		x
JV	Output of ln(K-Value) values.		x
KRG	Output of grid block gas relative permeabilities.	x	x
KRN	Output of grid block solvent relative permeabilities (for the Solvent Model).	x	
KRO	Output of grid block oil relative permeabilities.	x	x
KRW	Output of grid block water relative permeabilities.	x	x
MLSC	Output of total molar density for hydrocarbon components (moles per reservoir volume).		x
MWAT	Output of total molar density for water components (moles per reservoir volume).		x
MULT	Output of grid block pore volumes and transmissibilities if they have been modified.	x	x
NCNG	Output of capillary numbers for gas (VELDEP option). The normalized capillary number is output when used with keyword VDKRG.		x
NCNO	Output of capillary numbers for oil (VELDEP option). The normalized capillary number is output when used with keyword VDKRO.		x
NEWTON	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.	x	

Mnemonic	Output	E100	E300
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 <a href="#">ROCKPAMA</a> or <a href="#">ROCKPAME</a> ). Also gives the sign of the derivatives during the last calculation (positive $\geq 0$ ). Requires <a href="#">CBMOPTS</a> item 4 to be set to <code>PMREB</code> .	x	x
ODGAS	Output of oil displaced into gas zone (for VE option)  1 = by regions and field, >1 = by grid blocks	x	
ODWAT	Output of oil displaced into water zone (for VE option:  1 = by regions and field, >1 = by grid blocks	x	
OILAPI	Output of grid block oil API values (for API Tracking).	x	
OILPOT	Output of oil potentials.		x
PART	Output of domain partitions (Parallel option).		x
PB or PBUB RSSAT	Output of grid block bubble point pressures:  1. outputs bubble point pressures only 2. in addition, outputs the saturated dissolved gas-oil ratios	x	x
PBLK or POLYMER	Output of grid block polymer concentration values (for Polymer Flood option).	x	
PCOG	Output of gas-oil capillary pressures.		x
PCOW	Output of oil-water capillary pressures.		x
PCGW	Output of gas-water capillary pressures.		x
PD or PDEW RVSAT	Output of grid block dew point pressures:  1. outputs dew point pressures only 2. in addition, outputs the saturated vaporized oil-gas ratios	x	x
PERMSTAB	Geomechanics: outputs permeability multipliers (stress dependence).		x
PERM_MDX PERM_MDY PERM_MDZ	Output of directional permeability modifiers (Rock Compaction option) when <a href="#">RKTRMDIR</a> or <a href="#">ROCKTRMX</a> , <a href="#">ROCKTRMY</a> and <a href="#">ROCKTRMZ</a> are defined in ECLIPSE 300.  This information can be output in ECLIPSE 100 using the <a href="#">ROCKC</a> mnemonic (described below).		x
PERM_MOD	Output of permeability modifiers for Rock Compaction option ( <a href="#">ROCKCOMP</a> or <a href="#">ROCKPAME</a> ).  This information can be output in E100 via the <a href="#">ROCKC</a> mnemonic (described below).		x
PGAS	Output of grid block gas phase pressures	x	x

Mnemonic	Output	E100	E300
PGASD or POTG	Output of grid block gas phase pressures (corrected to datum depth). (See also the <a href="#">DATUM</a> keyword).	x	
PKRG	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRGR	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRO	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRORG	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRORW	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRW	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRWR	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PLASDISP	Geomechanics: outputs plastic displacements.		x
PLYADS	Polymer Flood: output of current adsorbed polymer concentration on the formation rock.	x	
POILD or POTO	Output of grid block oil phase pressures (corrected to datum depth).	x	
POLY	Output of polymer concentration (Polymer option)		x
POLYVM	Output of polymer viscosity multiplier (Polymer option)		x
PORV_MOD	Output of pore volume modifiers for rock compaction ( <a href="#">ROCKCOMP</a> case or <a href="#">ROCKPAME</a> ).  This information can be output in E100 via the <a href="#">ROCKC</a> mnemonic (described below).		x
POTC	Output of initial contact corrected potential.	x	

Mnemonic	Output	E100	E300
PPCG	Output of the processed maximum gas-oil capillary pressures if composition or temperature-dependent end-point scaling is selected via the use of the <a href="#">ENPCVC</a> or <a href="#">ENPCVT</a> keywords respectively.		x
PPCW	Output of the processed maximum oil-water capillary pressures if composition or temperature-dependent end-point scaling is selected via the use of the <a href="#">ENPCVC</a> or <a href="#">ENPCVT</a> keywords respectively.		x
PRES or POIL or PRESSURE	Output of grid block pressures.	x	x
PRESMIN	Output of minimum pressures (Irreversible rock compaction option).		x
PRSTRESS	Geomechanics: outputs principal stresses: PRSTRES1, PRSTRES2, PRSTRES3 followed by their respective directions.		x
PSAT	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.		x
PSGCR	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively or the mobile fluid correction options or <a href="#">TZONE</a> are activated.		x
PSGL	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively.		x
PSGU	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively.		x
PSOGCR	Output of the critical oil-in-gas saturations for each grid cell if composition or temperature-dependent end-point scaling is selected via the use of the <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively or the mobile fluid correction options or <a href="#">TZONE</a> are activated.		x
PSOWCR	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively or the mobile fluid correction options or <a href="#">TZONE</a> are activated.		x
PSWCR	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively or the mobile fluid correction options or <a href="#">TZONE</a> are activated.		x

Mnemonic	Output	E100	E300
PSWL	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 ENPTVC or ENPTVT keywords respectively.		x
PSWU	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.		x
PWAT	Output of grid block water phase pressures.	x	
PWATD or POTW	Output of grid block water phase pressures (corrected to datum depth).	x	
REAC	Chemical reaction rates (See " <a href="#">Chemical reactions</a> " in the <i>ECLIPSE Technical Description</i> ).		x
RECOV	Output of recovery mechanisms (See " <a href="#">Recovery mechanisms</a> " in the <i>ECLIPSE Technical Description</i> ).	x	
RESTART	<p>Output of Restart files:</p> <p>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).</p> <p>If set &gt; 1, Restart files are created at every report time until this switch is reset, and all are kept.</p> <p>If set &gt; 2, the inter block flows of each phase are written to the Restart files.</p> <p>If set &gt; 3, the current fluids-in-place and phase potentials in each grid cell are written to the Restart files.</p> <p>If set = 6, Restart files are created at every timestep.</p> <p>See also the keywords <a href="#">RESTART</a>, <a href="#">RPTRST</a>, <a href="#">FMTOUT</a>, <a href="#">FMTIN</a>, <a href="#">UNIFIN</a> and <a href="#">UNIFOUT</a>. The value of this control will be ignored if the mnemonic BASIC in keyword <a href="#">RPTRST</a> has previously been set &gt; 2.</p> <p>Output of the initial Restart file (at 0.0 Days) is governed by the mnemonic RESTART in keyword <a href="#">RPTSOL</a>, but there must be at least one timestep.</p> <p>In ECLIPSE 300 the keyword <a href="#">RPTRST</a> controls output to the Restart file.</p>	x	
RK	Output of the permeability reduction factor currently being applied to water phase relative permeability (for Polymer Flood option).	x	

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

Mnemonic	Output	E100	E300
SOILM	Mobile oil saturation (SOR option).		x
SOILR	Residual oil saturation (SOR option).		x
SOLADS	Adsorbed solid concentration (Solid option)		x
SOLADW	Adsorbed wetting-solid concentration (Solid option)		x
SOLWET	Solid wettability interpolation parameter (Solid option)		x
SPENGAS	Output gas internal energy (ECLIPSE 300 Thermal).		x
SPENOIL	Output oil internal energy (ECLIPSE 300 Thermal).		x
SPENWAT	Output water internal energy (ECLIPSE 300 Thermal).		x
SSFRAC	Output of suspended solids fraction (SOLID option).		x
SSOL	Output of the current Solvent saturation.	x	
SSOLID	Output of grid block solid saturations (SOLID option).		x
STATE	Output of states present.		x
STEN	Output of oil/gas surface tensions (Miscible option).		x
STRAIN	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 "Elastic stress equations" in the <i>ECLIPSE Technical Description</i> .		x
SUMMARY	Output of solution summary at end of each timestep:  If set to 1: report is sent to Print file.  If set to 2: in addition, a copy of the report is sent to the short form output (terminal).  If set to 3: report is sent to short form output only.  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.	x	
SURF	Surfactant concentration - Surfactant option.		x
SURFADS	Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model).	x	
SURFBLK	Output of the surfactant concentrations, the SURFBLK capillary number (for the Surfactant Model) and water-oil surface tension (SURFST or SURFSTES).	x	
SURFCNM	log (capillary number) - Surfactant option.		x
SURFKR	Relative permeability interpolation parameter - Surfactant option.		x
SURFCP	Capillary pressure multiplier - Surfactant option.		x
SURFST	Oil-water surface tension - Surfactant option.		x
SV	Output vapor mole fractions (ECLIPSE 300 Thermal).		x
SW	Output water mole fractions (ECLIPSE 300 Thermal).		x
SWAT	Output of grid block water saturations.	x	x

Mnemonic	Output	E100	E300
T2	Output temperature at the two-phase to three-phase transition (ECLIPSE 300 Thermal).		x
T3	Output bubble point (ECLIPSE 300 Thermal).		x
TBLK or TRACER	Output of grid block tracer concentrations (for Tracer Tracking).	x	
TCMULT	Output saturation dependent thermal conductivity transmissibility multiplier (ECLIPSE 300 Thermal).		x
TEMP	Output of the grid block temperature (for the Temperature option or ECLIPSE 300 Thermal).	x	x
TET	Output temperatures at last time level (ECLIPSE 300 Thermal).		x
TO	Output temperature at which the oil phase leaves the system (ECLIPSE 300 Thermal).		x
TOTCOMP	Output the total compressibility of the fluid system (excluding the rock).		x
TOTSTRES	Geomechanics: outputs total stress tensor: TOTSTRXX , TOTSTRXY, TOTSTRXZ, TOTSTRYY, TOTSTRYZ , TOTSTRZZ.		x
TW	Output temperature at which the water phase leaves the system (ECLIPSE 300 Thermal).		x
TRADS or TRDCY	Output of tracer adsorption and decay (for Environmental Tracers).	x	
TREACM	Tracer-dependent reaction rate multipliers (See " <a href="#">Chemical Reactions</a> " in the <i>ECLIPSE Technical Description</i> ).		x
TSUB	Output the water saturation temperature minus the grid block temperature (THERMAL option).		x
TUNING	Output of simulator control parameters set by <a href="#">TUNING</a> keyword.	x	
UGAS	Output gas phase specific volumes (ECLIPSE 300 Thermal).		x
UOIL	Output oil phase specific volumes (ECLIPSE 300 Thermal).		x
UWAT	Output water phase specific volumes (ECLIPSE 300 Thermal).		x
V2	Output vapor mole fraction at the system two phase to three phase transition (ECLIPSE 300 Thermal).		x
VFPPROD	Output of vertical flow performance tables. (In order to obtain this, RPTSCHED must be placed before <a href="#">VFPPROD</a> and <a href="#">VFPINJ</a> .)  This also turns on network convergence messages in the Network option.  If set > 1: a table is also written comparing the maximum dimensions as defined in the <a href="#">VFPPDIMS</a> keyword with those observed in the input tables.  If set > 2: only the table comparing the maximum dimensions is written.	x	
VGAS	Output of grid block gas viscosity.	x	x
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas).		x

Mnemonic	Output	E100	E300
VOIL	Output of grid block oil viscosity.	x	x
VWAT	Output of grid block water viscosity.	x	x
WATPOT	Output of water potentials.		x
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. In ECLIPSE 300, the keyword <code>RPTPRINT</code> controls well report output to the Print file.	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.

Mnemonic	Output	E100	E300
IFTGO	Output of gas-oil interfacial tension		x
IFTOW	Output of oil-water interfacial tension		x
IFTGW	Output of gas-water interfacial tension		x
KRGOE	Output of equivalent relative permeabilities to gas for gas-oil system		x
KRGWE	Output of equivalent relative permeabilities to gas for gas-water system		x
KROGE	Output of equivalent relative permeabilities to oil for oil-gas system		x
KROWE	Output of equivalent relative permeabilities to oil for oil-water system		x
KRWGE	Output of equivalent relative permeabilities to water for water-gas system		x
KRWOE	Output of equivalent relative permeabilities to water for water-oil system		x
KRGOH	Output of turning point relative permeabilities to gas for gas-oil system		x
KRGWH	Output of turning point relative permeabilities to gas for gas-water system		x
KROGH	Output of turning point relative permeabilities to oil for oil-gas system		x
KROWH	Output of turning point relative permeabilities to oil for oil-water system		x
KRWGH	Output of turning point relative permeabilities to water for water-gas system		x
KRWOH	Output of turning point relative permeabilities to water for water-oil system		x
KRGOR	Output of representative gas-oil relative permeabilities		x
KRGWR	Output of representative gas-water relative permeabilities		x
KROGR	Output of representative oil-gas relative permeabilities		x
KROWR	Output of representative oil-water relative permeabilities		x
KRWGR	Output of representative water-gas relative permeabilities		x
KRWOR	Output of representative water-oil relative permeabilities		x
KRGOT	Output of opposite saturation direction turning point relative permeabilities to gas for gas-oil system		x
KRGWT	Output of opposite saturation direction turning point relative permeabilities to gas for gas-water system		x
KROGT	Output of opposite saturation direction turning point relative permeabilities to oil for oil-gas system		x
KROWT	Output of opposite saturation direction turning point relative permeabilities to oil for oil-water system		x
KRWGT	Output of opposite saturation direction turning point relative permeabilities to water for water-gas system		x

<b>Mnemonic</b>	<b>Output</b>	<b>E100</b>	<b>E300</b>
KRWOT	Output of opposite saturation direction turning point relative permeabilities to water for water-oil system		x
PCOWR	Representative oil-water capillary pressure		x
PCWOR	Representative water-oil capillary pressure		x
PCOGR	Representative oil-gas capillary pressure		x
PCGOR	Representative gas-oil capillary pressure		x
PCGWR	Representative gas-water capillary pressure		x
PCWGR	Representative water-gas capillary pressure		x
SONRM	Output of normalized oil saturations		x
SGNRM	Output of normalized gas saturations		x
SWNRM	Output of normalized water saturations		x
SDRGO	Output of gas phase saturation direction indicators for gas-oil system		x
SDRGW	Output of gas phase saturation direction indicators for gas-water system		x
SDROG	Output of oil phase saturation direction indicators for oil-gas system		x
SDROW	Output of oil phase saturation direction indicators for oil-water system		x
SDRWG	Output of water phase saturation direction indicators for water-gas system		x
SDRWO	Output of water phase saturation direction indicators for water-oil system		x
SGONE	Output of normalized gas equivalent opposite direction saturations for gas-oil system		x
SGWNE	Output of normalized gas equivalent opposite direction saturations for gas-water system		x
SOGNE	Output of normalized oil equivalent opposite direction saturations for oil-gas system		x
SOWNE	Output of normalized oil equivalent opposite direction saturations for oil-water system		x
SWGNE	Output of normalized water equivalent opposite direction saturations for water-gas system		x
SWONE	Output of normalized water equivalent opposite direction saturations for water-oil system		x
SGONH	Output of normalized gas hysteresis saturations for gas-oil system		x
SGWNH	Output of normalized gas hysteresis saturations for gas-water system		x
SOGNH	Output of normalized oil hysteresis saturations for oil-gas system		x
SOWNH	Normalized oil hysteresis saturations for oil-water system		x
SWGNH	Output of normalized water hysteresis saturations for water-gas system		x
SWONH	Output of normalized oil hysteresis saturations for water-oil system		x
SGROB	Output of residual gas saturation in gas-oil system used to calculate normalized saturations		x
SGRWB	Output of residual gas saturation in gas-water system used to calculate normalized saturations		x

Mnemonic	Output	E100	E300
SORGB	Output of residual oil saturation in oil-gas system used to calculate normalized saturations		x
SORWB	Output of residual oil saturation in oil-water system used to calculate normalized saturations		x
SWRGB	Output of residual water saturation in water-gas system used to calculate normalized saturations		x
SWROB	Output of residual water saturation in water-oil system used to calculate normalized saturations		x
SGROP	Output of process dependent residual gas saturation in gas-oil system		x
SGRWP	Output of process dependent residual gas saturation in gas-water system		x
SORWP	Output of process dependent residual oil saturation in oil-water system		x
SORGP	Output of process dependent residual oil saturation in oil-gas system		x
SWRGP	Output of process dependent residual water saturation in water-gas system		x
SWROP	Output of process dependent residual water saturation in water-oil system		x
SGONT	Output of normalized gas turning point saturations for gas-oil system		x
SGWNT	Output of normalized gas turning point saturations for gas-water system		x
SOGNT	Output of normalized oil turning point saturations for oil-gas system		x
SOWNT	Output of normalized oil turning point saturations for oil-water system.		x
SWGNT	Output of normalized water turning point saturations for water-gas system		x
SWONT	Output of normalized water turning point saturations for water-oil system		x

Table 3.43: ODD3P RPTSCHED output controls

The following table lists the deprecated mnemonics.

Mnemonic	Output	E100	E300
SGCR	This mnemonic has been superseded by PSGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SGL	This mnemonic has been superseded by PSGL (ENKRVC and ENKRVT cases).		x
SGU	This mnemonic has been superseded by PSGU (ENKRVC and ENKRVT cases).		x
SOGCR	This mnemonic has been superseded by PSOGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SOWCR	This mnemonic has been superseded by PSOWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWCR	This mnemonic has been superseded by PSWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWL	This mnemonic has been superseded by PSWL (ENKRVC and ENKRVT cases).		x
SWU	This mnemonic has been superseded by PSWU (ENKRVC and ENKRVT cases).		x
WMF	The mnemonic has been superseded by AMF.		x

Table 3.44: Deprecated RPTSCHED output controls

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.

## 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

Mnemonic	Output
BASIC	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.
FLows	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.
LiNES	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.
NOADDLiN	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.
TRACERS	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

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 /
```

## RPTSMRY

**Controls on output from SUMMARY section**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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**

```
RPTSMRY
 1 /
```

## RPTSOL

## Controls on output from SOLUTION section

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

Mnemonic	Output	E100	E300
AIM	Output AIM status array (1 for implicit cells; 2 for explicit cells).		x
ALSURF	Output alkaline concentration when surfactant and alkaline models are activated.		x
ALSTML	Output alkaline surface tension multiplier when surfactant and alkaline models are activated.		x
AMF	Output of aqueous mole fractions ( <a href="#">CO2SOL</a> , <a href="#">GASSOL</a> and <a href="#">THERMAL</a> , <a href="#">KWTABTn</a> / <a href="#">CO2STORE</a> and <a href="#">COMPW</a> options).		x
ANIONS	Output of anion concentration.	x	
APIVD	Output of tables of initial API versus depth.	x	

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 <a href="#">AQUFET</a> , <a href="#">AQUFETP</a> , <a href="#">AQUCT</a> , <a href="#">AQUCHWAT</a> , <a href="#">AQUCHGAS</a> and <a href="#">AQUFLUX</a> ).  ECLIPSE 100: if set to 2: in addition, data for each aquifer-grid block connection will be printed (input using <a href="#">AQUANCON</a> ).  ECLIPSE 300: data for each aquifer-grid block connection will be printed (input using <a href="#">AQUANCON</a> ).  Note: <a href="#">AQUFET</a> 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 <a href="#">Construction of flows</a> in the <i>ECLIPSE Technical Description</i> ). This is only output if <a href="#">VDFLOW</a> or <a href="#">VDFLOWR</a> (or <a href="#">VELDEP</a> 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 <a href="#">Construction of flows</a> in the <i>ECLIPSE Technical Description</i> ). This is only output if <a href="#">VDFLOW</a> , <a href="#">VDFLOWR</a> or <a href="#">VELDEP</a> 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 ( <a href="#">SOLID</a> 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 ( <a href="#">SOR</a> 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 ( <a href="#">SOLID</a> 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 ( <a href="#">THERMAL</a> option).		x
EQUIL	Output of equilibration data.	x	

Mnemonic	Output	E100	E300
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
ESALSUR	Output of effective salinity for phase behavior and surfactant properties.	x	
FFORG	Forchheimer Non-Darcy flow factor for Gas ( <b>VELDEP</b> option). ECLIPSE 100: maximum deviation from Darcy Flow.	x	x
FFORO	Forchheimer Non-Darcy flow factor for Oil ( <b>VELDEP</b> option). ECLIPSE 100: not applicable in the current version.		x
FIP	Output of fluid in place reports:  If set to 1: initial fluids in place are reported for the whole field,  If set to 2: in addition, initial fluids in place are reported for each fluid in place region defined with the <b>FIPNUM</b> keyword,  If set to 3: in addition, initial fluids in place are reported for all sets of fluid in place regions defined with the <b>FIP</b> keyword.  ECLIPSE 300 only: FIP=1 or 2 produces no additional output as these reports are produced by default.	x	x
FIPFOAM	Output of foam fluid in place reports (for the Foam model):  1: gives a report for the whole field  2: in addition, gives a report for each fluid-in-place region	x	
FIPPLY	Output of polymer/salt fluid in place reports (for Brine option and salt-sensitive Polymer Flood Model):  1: gives a report for the whole field  2: in addition, gives a report for each fluid-in-place region	x	
FIPRESV	Output of region reservoir volume reports.	x	
FIPSOL	Output of the solvent fluid in place reports:  1: gives a report for the whole field  2: in addition, gives a report for each fluid-in-place region	x	
FIPSURF	Output of surfactant fluid in place reports (for the Surfactant Model):  1. gives a report for the whole field  2. in addition, gives a report for each fluid-in-place region	x	
FIPTEMP or FIPHEAT	Output of energy in place reports (for the Temperature option):  1. gives a report for the whole field  2. in addition, gives a report for each fluid-in-place region	x	

Mnemonic	Output	E100	E300
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	
FLOCn	Output of inter-block component flows (for component n).		x
FLOE	Output of inter-block energy flow ( <a href="#">THERMAL</a> 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	
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 <a href="#">SOLID</a> option).		x
HWAT	Output water enthalpy values (ECLIPSE 300 Thermal).		x

Mnemonic	Output	E100	E300
JV	Output of ln(K-Value) values (compositional cases only).		x
KRG	Output of grid block gas relative permeabilities.	x	x
KRN	Output of grid block solvent relative permeabilities (for the Solvent Model)	x	
KRO	Output of grid block oil relative permeabilities.	x	x
KRW	Output of grid block water relative permeabilities.	x	x
MLSC	Output of total molar density for hydrocarbon components (moles per reservoir volume).		x
MWAT	Output of total molar density for water components (moles per reservoir volume).		x
NCNG	Output of capillary numbers for gas ( <a href="#">VELDEP</a> option). The normalized capillary number is output when used with keyword <a href="#">VDKRG</a> .		x
NCNO	Output of capillary numbers for oil ( <a href="#">VELDEP</a> option). The normalized capillary number is output when used with keyword <a href="#">VDKRO</a> .		x
OILAPI	Output of initial grid block API values (for API tracking).	x	
OILPOT	Output of oil potentials.		x
PART	Output of all possible domain partitions for the Parallel option, that is the <a href="#">PSPLITX</a> and <a href="#">PSPLITY</a> keywords. This is always enabled in parallel.		x
PERM_MDX PERM_MDY PERM_MDZ	Output of directional permeability modifiers (Rock Compaction option) when <a href="#">RKTRMDIR</a> or <a href="#">ROCKTRMX</a> , <a href="#">ROCKTRMY</a> and <a href="#">ROCKTRMZ</a> are defined in ECLIPSE 300.  This information can be output in ECLIPSE 100 via the <a href="#">ROCKC</a> mnemonic (described below).		x
PERM_MOD	Output of permeability modifiers for Rock Compaction option ( <a href="#">ROCKCOMP</a> or <a href="#">ROCKPAME</a> ).  This information can be output in ECLIPSE 100 via the <a href="#">ROCKC</a> mnemonic (described below).		x
PORV_MOD	Output of pore volume modifiers for rock compaction ( <a href="#">ROCKCOMP</a> case or <a href="#">ROCKPAME</a> ).  This information can be output in ECLIPSE 100 via the <a href="#">ROCKC</a> mnemonic (described below).		x
PB or PBUB RSSAT	Output of grid block bubble point pressures:  1: outputs bubble point pressures only  2: in addition, outputs the saturated dissolved gas-oil ratios	x	x
PBLK or POLYMER	Output of initial grid block polymer concentration values (for Polymer Flood Model).	x	
PCGW	Output of gas-water capillary pressures ( <a href="#">GASWAT</a> cases).		x
PCOG	Output of gas-oil capillary pressures.		x

Mnemonic	Output	E100	E300
PCOW	Output of oil-water capillary pressures.		x
PD or PDEW RVSAT	Output of grid block dew point pressures: 1: outputs dew point pressures only 2: in addition, outputs the saturated vaporized oil-gas ratios	x	
PERMSTAB	Geomechanics: outputs permeability multipliers (stress dependence).		x
PGAS	Output of initial grid block gas phase pressures.	x	
PGASD or POTG	Output of initial grid block gas phase pressures corrected to datum depth) (See also the <a href="#">DATUM</a> keyword).	x	
PGOSTRES	Geomechanics: outputs initial gravity induced principal stress.		x
PKRG	Output of processed max. oil relative permeability and the gas relative permeability at residual oil if the end point scaling is activated.	x	
PKRO	Output of processed max. oil relative permeability and the oil relative permeability at residual gas and water if end-point scaling is activated.	x	
PKRW	Output of processed max. water relative permeability and the water relative permeability at residual oil if the end-point scaling is activated.	x	
PKRG	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRO	Output of the processed scaled oil relative permeability at the maximum oil saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRORG	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 <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRORW	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 via the use of the <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRW	Output of the processed scaled water relative permeability at the maximum water saturation for each cell if composition or temperature-dependent end-point scaling is selected via the use of the <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x
PKRWR	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 via the use of the <a href="#">ENKRVC</a> or <a href="#">ENKRVT</a> keywords respectively.		x

Mnemonic	Output	E100	E300
PLYADS	Output of polymer concentration currently adsorbed by the rock formation (for Polymer Flood Model).	x	
POIL	Output of initial grid block pressures.	x	x
POILD or POTO	Output of initial grid block oil phase pressures (corrected to datum depth).	x	
POLY	Output of polymer concentration (Polymer option)		x
POLYVM	Output of polymer viscosity multiplier (Polymer option)		x
POTC	Output of initial contact corrected potential.	x	
PPCG	Output of the processed maximum gas-oil capillary pressures (End-Point Scaling).	x	
PPCW	Output of the processed maximum oil-water capillary pressures (End-Point Scaling).	x	
PPCG	Output of the processed maximum gas-oil capillary pressures if composition or temperature-dependent end-point scaling selected via the use of the <a href="#">ENPCVC</a> or <a href="#">ENPCVT</a> keywords respectively.		x
PPCW	Output of the processed maximum oil-water capillary pressures if composition or temperature-dependent end-point scaling selected via the use of the <a href="#">ENPCVC</a> or <a href="#">ENPCVT</a> keywords respectively.		x
PRES or PRESSURE	Output of initial oil pressures.	x	x
PRESMIN	Output of minimum pressures (irreversible Rock Compaction option).		x
PRSTRESS	Geomechanics: outputs initial principal in-situ stress.		x
PSAT	Output of saturation pressures (Compositional mode).  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.		x
PSGCR	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively or the mobile fluid correction options or <a href="#">TZONE</a> are activated.		x
PSGL	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 <a href="#">ENPTVC</a> or <a href="#">ENPTVT</a> keywords respectively.		x
PSGU	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 <a href="#">ENPTVT</a> or <a href="#">ENPTVT</a> keywords respectively.		x

Mnemonic	Output	E100	E300
PSOGCR	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 ENPTVT or ENPTVT keywords respectively or the mobile fluid correction options or TZONE are activated.		x
PSOWCR	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.		x
PSWCR	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.		x
PSWL	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.		x
PSWU	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.		x
PTCSTRES	Geomechanics: outputs initial tectonic induced principal stress.		x
PTHSTRES	Geomechanics: outputs initial thermal induced principal stress.		x
PVDPH	Derivative of pore volume with respect to porosity.		x
PWAT	Output of initial grid block water phase pressures.	x	
PWATD or POTW	Output of initial grid block water phase pressures (corrected to datum depth).	x	
QUIESC	Output phase pressure modifications.	x	
RECOV	Output of recovery mechanisms.	x	
RESTART	Output of Restart files:  If set > 1: an initial Restart file is created.  If set > 2: the inter block flows of each phase are written to the Restart files.  If set > 3: the current fluids in place and phase potentials in each grid cell are written to the Restart files.  (There must be at least one <a href="#">TIME</a> , <a href="#">TSTEP</a> or <a href="#">DATES</a> keyword if an initial Restart file is required; but the simulation mode can be turned off by the keyword <a href="#">NOSIM</a> in the RUNSPEC section)  In ECLIPSE 300 the keyword <a href="#">RPTRST</a> controls output to the Restart file.	x	
RK	Output of permeability reduction factor currently being applied to the water phase (for Polymer Flood Model).	x	

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 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
ROCKCON	Output of rock connections to the local grid refinements and to the global grid ( <b>THERMAL</b> option only).		x
ROCKMASS	Output rock mass (Polymer, Surfactant and Environmental Tracer options).	x	
ROMLS	Residual oil composition ( <b>SOR</b> option).		x
RPORV	Output of reservoir pore volumes.		x
PORV	Output of reservoir pore volumes at reference pressure.		x
RS	Output of initial 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
RSVD or PBVD	Output of tables of either initial $R_s$ versus depth or initial $P_b$ versus depth.	x	
RSW	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 AMF, by its associated index.		x
RV	Output of initial 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.	x	x
RVVD	Output of tables of initial $R_v$ versus depth.	x	
SALT	Output of initial grid block salt concentration values (for Brine option and salt-sensitive Polymer Flood Model).	x	x
SALTVD	Output of tables of initial salt concentration versus depth (entered using <b>SALTVD</b> keyword)  For the Multi-component Brine Model (ECLIPSE 100 only) SALTVD outputs tables of initial anion and divalent cation equivalent molar concentrations versus depth.	x	x
SFOIL	Output of initial grid block foamy oil saturation.		x
SFSOL	Output of initial grid block foamy solid saturation.		x
SGAS	Output of initial grid block gas saturations.	x	x

Mnemonic	Output	E100	E300
SGCO or SGCR or SGL or SGU	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.	x	
SIGM_MOD	Output of dual porosity sigma multipliers (Rock Compaction option)	x	x
SMMULT	Output of solid mobility multiplier ( <b>SOLID</b> option).		x
SOCRS	Output of critical oil saturations following a surfactant flood (Surfactant option with end point scaling active).	x	
SOGCR	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.	x	
SOIL	Output of initial grid block oil saturations.	x	x
SOILM	Mobile oil saturation ( <b>SOR</b> option).		x
SOILR	Residual oil saturation ( <b>SOR</b> option).		x
SOLADS	Adsorbed solid concentration ( <b>Solid</b> option).		x
SOLADW	Adsorbed wetting-solid concentration ( <b>Solid</b> option).		x
SOLVD	Composition with depth.		x
SOLWET	Solid wettability interpolation parameter ( <b>Solid</b> option).		x
SOWCR	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.	x	
SSFRAC	Output of suspended solids fraction ( <b>SOLID</b> option).		
SSOL	Output of the initial solvent saturation.	x	
SSOLID	Output of initial grid block solid saturations ( <b>SOLID</b> option).		x
STATE	Output states present.		x
STEN	Output of oil/gas surface tensions ( <b>Miscible</b> option).		x
SURF	Surfactant concentration - Surfactant option.		x
SURFADS	Output of current adsorbed surfactant concentration on the formation rock (for the Surfactant Model).	x	
SURFBLK	Output of the Surfactant concentrations (for the Surfactant Model) and water-oil surface tensions.	x	
SURFCNM	log (capillary number) - Surfactant option.		x
SURFKR	Relative permeability interpolation parameter - Surfactant option.		x
SURFCP	Capillary pressure multiplier - Surfactant option.		x
SURFST	Oil-water surface tension - Surfactant option.		x
SWAT	Output of grid block water saturations.	x	x
SWATINIT	Output of scaled maximum water-oil capillary pressures generated from initial water distribution entered with keyword <b>SWATINIT</b> .	x	

Mnemonic	Output	E100	E300
SWCO or SWL	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 P <sub>SWL</sub> .	x	
SWCR	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 P <sub>SWCR</sub> .	x	
SWU	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 P <sub>SWU</sub> .	x	
TBLK or TRACER	Output of initial grid block tracer concentrations (for Tracer Tracking).	x	
TCMULT	Saturation dependent thermal transmissibility multiplier $\lambda$ ( <a href="#">THERMAL</a> option).		x
TCSTRES	Geomechanics: outputs initial tectonic induced effective stress.		x
TEMP	Output of the grid block temperature (for the Temperature option in ECLIPSE 100 or the <a href="#">THERMAL</a> option in ECLIPSE 300).	x	x
THPRES	Output of threshold pressures (see keywords <a href="#">EQLOPTS</a> and <a href="#">THPRES</a> )	x	x
THSTRES	Geomechanics: outputs initial thermal induced effective stress.		x
TRADS or TRDCY	Output of tracer adsorption and decay (for Environmental Tracers).	x	
TVDP	Output of initial tracer concentration versus depth tables for the Tracer Tracking option (input using <a href="#">TVDP</a> ).	x	
VGAS	Output of grid block gas viscosity.	x	x
VMF	Output of vapor mole fractions (fraction of total hydrocarbon moles in gas).		x
VOIL	Output of grid block oil viscosity.	x	x
VWAT	Output of grid block water viscosity.	x	x
WATPOT	Output of water potentials		x
WOC	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	x	
XFW	Output of liquid component mass fractions (weight fraction of each component in liquid).		x
XGAS	Output of gas fractional saturations (Sudation option).		x
XMF	Output of liquid component mole fractions (fraction of each component in liquid).		x
XWAT	Output of water fractional saturations (Sudation option).		x
YFW	Output of vapor component mass fractions (weight fraction of each component in gas).		x

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.

Mnemonic	Output	E100	E300
I'FTGO	Output of gas-oil interfacial tension		x
I'FTOW	Output of oil-water interfacial tension		x
I'FTGW	Output of gas-water interfacial tension		x
KRGOE	Output of equivalent relative permeabilities to gas for gas-oil system		x
KRGWE	Output of equivalent relative permeabilities to gas for gas-water system		x
KROGE	Output of equivalent relative permeabilities to oil for oil-gas system		x
KROWE	Output of equivalent relative permeabilities to oil for oil-water system		x
KRWGE	Output of equivalent relative permeabilities to water for water-gas system		x
KRWOE	Output of equivalent relative permeabilities to water for water-oil system		x
KRGOH	Output of turning point relative permeabilities to gas for gas-oil system		x
KRGWH	Output of turning point relative permeabilities to gas for gas-water system		x
KROGH	Output of turning point relative permeabilities to oil for oil-gas system		x
KROWH	Output of turning point relative permeabilities to oil for oil-water system		x
KRWGH	Output of turning point relative permeabilities to water for water-gas system		x
KRWOH	Output of turning point relative permeabilities to water for water-oil system		x
KRGOR	Output of representative gas-oil relative permeabilities		x
KRGWR	Output of representative gas-water relative permeabilities		x
KROGR	Output of representative oil-gas relative permeabilities		x
KROWR	Output of representative oil-water relative permeabilities		x
KRWGR	Output of representative water-gas relative permeabilities		x
KRWOR	Output of representative water-oil relative permeabilities		x
KRGOT	Output of opposite saturation direction turning point relative permeabilities to gas for gas-oil system		x
KRGWT	Output of opposite saturation direction turning point relative permeabilities to gas for gas-water system		x
KROGT	Output of opposite saturation direction turning point relative permeabilities to oil for oil-gas system		x
KROWT	Output of opposite saturation direction turning point relative permeabilities to oil for oil-water system		x

<b>Mnemonic</b>	<b>Output</b>	<b>E100</b>	<b>E300</b>
KRWGT	Output of opposite saturation direction turning point relative permeabilities to water for water-gas system		x
KRWOT	Output of opposite saturation direction turning point relative permeabilities to water for water-oil system		x
PCOWR	Representative oil-water capillary pressure		x
PCWOR	Representative water-oil capillary pressure		x
PCOGR	Representative oil-gas capillary pressure		x
PCGOR	Representative gas-oil capillary pressure		x
PCGWR	Representative gas-water capillary pressure		x
PCWGR	Representative water-gas capillary pressure		x
SONRM	Output of normalized oil saturations		x
SGNRM	Output of normalized gas saturations		x
SWNRM	Output of normalized water saturations		x
SDRGO	Output of gas phase saturation direction indicators for gas-oil system		x
SDRGW	Output of gas phase saturation direction indicators for gas-water system		x
SDROG	Output of oil phase saturation direction indicators for oil-gas system		x
SDROW	Output of oil phase saturation direction indicators for oil-water system		x
SDRWG	Output of water phase saturation direction indicators for water-gas system		x
SDRWO	Output of water phase saturation direction indicators for water-oil system		x
SGONE	Output of normalized gas equivalent opposite direction saturations for gas-oil system		x
SGWNE	Output of normalized gas equivalent opposite direction saturations for gas-water system		x
SOGNE	Output of normalized oil equivalent opposite direction saturations for oil-gas system		x
SOWNE	Output of normalized oil equivalent opposite direction saturations for oil-water system		x
SWGNE	Output of normalized water equivalent opposite direction saturations for water-gas system		x
SWONE	Output of normalized water equivalent opposite direction saturations for water-oil system		x
SGONH	Output of normalized gas hysteresis saturations for gas-oil system		x
SGWNH	Output of normalized gas hysteresis saturations for gas-water system		x
SOGNH	Output of normalized oil hysteresis saturations for oil-gas system		x
SOWNH	Normalized oil hysteresis saturations for oil-water system		x
SWGNH	Output of normalized water hysteresis saturations for water-gas system		x
SWONH	Output of normalized oil hysteresis saturations for water-oil system		x
SGROB	Output of residual gas saturation in gas-oil system used to calculate normalized saturations		x

Mnemonic	Output	E100	E300
SGRWB	Output of residual gas saturation in gas-water system used to calculate normalized saturations		x
SORGB	Output of residual oil saturation in oil-gas system used to calculate normalized saturations		x
SORWB	Output of residual oil saturation in oil-water system used to calculate normalized saturations		x
SWRGB	Output of residual water saturation in water-gas system used to calculate normalized saturations		x
SWROB	Output of residual water saturation in water-oil system used to calculate normalized saturations		x
SGROP	Output of process dependent residual gas saturation in gas-oil system		x
SGRWP	Output of process dependent residual gas saturation in gas-water system		x
SORGP	Output of process dependent residual oil saturation in oil-gas system		x
SORWP	Output of process dependent residual oil saturation in oil-water system		x
SWRGP	Output of process dependent residual water saturation in water-gas system		x
SWROP	Output of process dependent residual water saturation in water-oil system		x
SGONT	Output of normalized gas turning point saturations for gas-oil system		x
SGWNT	Output of normalized gas turning point saturations for gas-water system		x
SOGNT	Output of normalized oil turning point saturations for oil-gas system		x
SOWNT	Output of normalized oil turning point saturations for oil-water system.		x
SWGNT	Output of normalized water turning point saturations for water-gas system		x
SWONT	Output of normalized water turning point saturations for water-oil system		x

Table 3.47: ODD3P RPTSOL output controls

The following table lists the deprecated mnemonics.

Mnemonic	Output	E100	E300
SGCR	This mnemonic has been superseded by PSGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SGL	This mnemonic has been superseded by PSGL (ENKRVC and ENKRVT cases).		x
SGU	This mnemonic has been superseded by PSGU (ENKRVC and ENKRVT cases).		x
SOGCR	This mnemonic has been superseded by PSOGCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SOWCR	This mnemonic has been superseded by PSOWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWCR	This mnemonic has been superseded by PSWCR (TZONE, mobile fluid correction options, ENKRVC and ENKRVT cases).		x
SWL	This mnemonic has been superseded by PSWL (ENKRVC and ENKRVT cases).		x
SWU	This mnemonic has been superseded by PSWU (ENKRVC and ENKRVT cases).		x

Mnemonic	Output	E100	E300
WMF	The mnemonic has been superseded by AMF.		x

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (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 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](#):

```
RS
48*1.29      192*1.42      /
```

## RSCONST Sets a constant $R_s$ value for dead oil

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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), atma (LAB), atma (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 [RSCONSTT](#).

### Example

```
RSCONST
0.37 1015 /
```

## RSCONSTT Sets a constant $R_s$ value for each dead oil PVT table

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [RSCONST](#) is that [RSCONSTT](#) allows the oil in different PVT table regions to have different  $R_s$  values. There must be no transmissibility between PVT table regions with different  $R_s$  values, to prevent oils with different  $R_s$  values from mixing. (If oil is allowed to flow between the regions, their  $R_s$  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  $R_s$  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 [PVTNUM](#).

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 ( $R_s$ ).

UNITS:  $\text{sm}^3/\text{sm}^3$  ([METRIC](#)),  $\text{Mscf}/\text{stb}$  ([FIELD](#)),  $\text{scc}/\text{scc}$  ([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  $R_s$  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 /
```

## RSGI

## Variation of saturated GOR with pressure and Gi

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
RSGI
100.0  1.0
        0.9  /
200.0  1.0
        0.85 /
/
```

## RSM

### Output extra data to RSM file

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## RSVD

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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  $R_s$ , the dissolved 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)

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](#).

ECLIPSE 100

In API tracking runs, however, the PBVD keyword cannot be used, and thus the RSVD keyword should be used.

ECLIPSE 300

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  $\geq$ 2:

```

RSVD
7000 1.4
8000 1.4 /

```

## RSW

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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,  $\text{CO}_2$  and/or  $\text{H}_2\text{S}$ ) in the aqueous phase. This is expressed as the volume of dissolved gases (for example,  $\text{CO}_2$  or  $\text{H}_2\text{S}$ ) at surface conditions per volume of water at surface conditions. If `H2SSOL` is used the grid values correspond to the  $\text{H}_2\text{S}$  component. If both `CO2SOL` and `H2SSOL` are present all grid values for  $\text{CO}_2$  are entered first followed by the values for  $\text{H}_2\text{S}$ . If `GASSOL` is used the grid values correspond to the `NCMPHW` components.

The data field should be terminated by a slash (/).

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (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 `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---- --- h2s --- ---c3
147*0.135 147*0.174 147*0.184 /
```

## RSWVD

## Aqueous concentration of dissolved gases versus depth

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises a table of concentrations of dissolved gases (for example, CO<sub>2</sub> and or H<sub>2</sub>S) versus depth. Each table consists of number columns of data (for example, 2 columns for CO<sub>2</sub>SOL or H<sub>2</sub>SSOL, 3 columns for CO<sub>2</sub>SOL and H<sub>2</sub>SSOL, and NCMPHW +1 columns for the GASSOL case), and must be terminated by a slash (/).

Column:

- Depth values.  
Should increase monotonically down column.  
UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)
- Concentration of first dissolved gas (for example, CO<sub>2</sub> in the CO<sub>2</sub>SOL case or the concentration of H<sub>2</sub>S for the H<sub>2</sub>SSOL case) in the aqueous phase.  
UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)
- Concentration of second dissolved gas (for example, H<sub>2</sub>S if both CO<sub>2</sub>SOL and H<sub>2</sub>SSOL are present) in the aqueous phase.  
UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)
- to NCMPHW+1.  
Concentration of last dissolved gas, for example concentration of dissolved gas number NCMPHW specified by the GASSOL keyword.  
UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (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<sub>2</sub> and or H<sub>2</sub>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<sub>2</sub> or H<sub>2</sub>S) and hydrocarbon liquid. The concentration reported at initialization is the concentration after flashing the system, and hence dissolved gases (for example, CO<sub>2</sub> or H<sub>2</sub>S) may be redistributed between the phases in each grid block.

See also "CO<sub>2</sub>/H<sub>2</sub>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<sub>2</sub>)

```
RSWVD
100 0.001
10000 0.002 /
```

### Example 2

Definition of the user defined concentrations of dissolved gases (for example, for CO<sub>2</sub> and H<sub>2</sub>S)

```
RSWVD
100 0.001 0.003
1000 0.002 0.004 /
```

### Example 3

Definition of the user defined concentrations of dissolved gases in the [GASSOL](#) case.

```
RSWVD
100 0.001 0.003 0.25 0.01
1000 0.002 0.004 0.27 0.03/
```

## RTEMP

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 9<sup>th</sup> 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

```
RTEMP
200 /
```

## RTEMPA

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## RUNSUM

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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), stb/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 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\*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 NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword [DIMENS](#):

```
RV
48*0.00725 48*0.00326 144*0 /
```

## RVCONST Sets a constant $R_v$ value for dry gas

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ( $R_v$ ).

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

2. The dew point pressure.

DEFAULT: 0.0

UNITS: barsa (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  $R_v$  \* the gas flow rate.

See also keyword [RVCONSTT](#).

### Example

```
RVCONST
0.00047 330 /
```

## RVCONSTT Sets a constant $R_v$ value for each dry gas PVT table

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $R_v$  values. There must be no transmissibility between PVT table regions with different  $R_v$  values, to prevent gases with different  $R_v$  values from mixing. (If gas is allowed to flow between the regions, their  $R_v$  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  $R_v$  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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

2. The dew point pressure.

DEFAULT: 0.0

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (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 /
```

## RVGI

## Variation of saturated OGR with pressure and Gi

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
/
```

## RVVD

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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  $\geq$ 2:

```
RVVD
7000 0.00725
8000 0.00725 /
```

## **S**

This section contains an alphabetic listing of keywords starting with the letter S.

## SALINTR

## Regional reservoir salinity

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 CO<sub>2</sub> solubility data in [CO2SOL](#) cases. The units are molality (gm-M/kg).

UNITS: 10<sup>-3</sup> kg-M/kg (METRIC), 10<sup>-3</sup> lb-M/lb (FIELD), 10<sup>-3</sup> gm-M/gm (LAB), 10<sup>-3</sup> 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.

```
SALINTR
0.51 /
0.30 /
```

## SALINITY

## Reservoir salinity

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 CO<sub>2</sub> solubility data in [CO2SOL](#) cases.

The units are molality (gm-M/kg).

UNITS: 10<sup>-3</sup> kg-M/kg (METRIC), 10<sup>-3</sup> lb-M/lb (FIELD), 10<sup>-3</sup> gm-M/gm (LAB), 10<sup>-3</sup> 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 /
```

# SALT

## Initial salt concentrations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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 [SALTVTD](#) keyword should be used instead of SALT.

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<sup>3</sup> (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

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
/
```

ECLIPSE 100 only

ECLIPSE 100 only

## SALTEFF

## Effective salinity coefficients for surfactant and polymer models

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword may only be used when salinity dependent surfactant and/or polymer models are active (when keywords `SURFSTES` with `ESSNODE` or `PLYVISCS` with `ESPNOE` 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 [PLYESAL](#) can be recovered with Formula 1 above. Similarly, the expression for effective salinity input with the ECLIPSE 100 keyword [SURFESAL](#) can be recovered with Formula 2 above.

There are currently no restrictions on the choice of coefficients  $a_i$  and  $b_i$ ,  $i = 1, \dots, N_w$ . However, if item 2 is set to 2, it is not recommended to choose coefficients  $a_i$  and  $b_i$ ,  $i = 0, \dots, N_w$  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 *ECLIPSE Technical Description* and Chapter 3 "Salinity Effect and Ion Exchange" in [\[Ref. 137\]](#).

## Example

With  $N_w = 3$  and NTPVT=1:

```

SALTEFF
-- model    effective      a_0   a_1   a_2   a_3   b_0   b_1   b_2   b_3
-- type     salinity type
  POLY      1              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
  SURF      2              0.0   2.1   1.4   0.0   0.1   2.1   0.8   0.0 /
    
```

## SALTNODE Salt concentration nodes for polymer solution viscosity

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (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 [PLYVISC](#) 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 /
```

## SALTVD

## Salt concentration versus depth for equilibration

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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:

```
SALTVD
7000 1.4
8000 1.4 /
```

### Example 2

Multi-component brine model with two salts specified in the [BRINE](#) keyword and with NTEQUL=2 and NDRXVD ≥2:

```
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/
```

ECLIPSE 100 only

ECLIPSE 100 only

## SAMG

## Algebraic multigrid solver

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 300

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [KRUNUMX](#), [KRUNUMY](#) and [KRUNUMZ](#) 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 [KRUNUMX](#), [KRUNUMX-](#), [KRUNUMY](#), [KRUNUMY-](#), [KRUNUMZ](#) and [KRUNUMZ-](#) in the **REGIONS** section.

If **DIRECT** is specified but **IRREVERS** is not, then only 3 sets of tables are needed ([KRUNUMX](#), [KRUNUMY](#) and [KRUNUMZ](#)). 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-](#).

*ECLIPSE 100 only*

**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 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:

SATOPTS  
DIRECT IRREVERS /

## SAVE

## Requests output of SAVE file for fast restarts

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [FMPSAVE](#) 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

```
-- ECLIPSE 100
RUNSPEC
SAVE
  FORMATTED /
```

## Example 3

Save the current solution after 1 year

```
-- 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## SCALECRS Controls the end point scaling method

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#) and [SWU](#)

$K_{rg}$ , [SGCR](#) and [SGU](#)

$K_{row}$ , [SOWCR](#) and (1.0- [SWL](#) - [SGL](#))

$K_{rog}$ , [SOGCR](#) 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) and [SWU](#)

$K_{rg}$ , [SGCR](#), (1.0-SOGCR-SWL) and [SGU](#)

$K_{row}$ , [SOWCR](#), (1.0-SWCR-SGL) and (1.0-SWL-SGL)

$K_{rog}$ , [SOGCR](#), (1.0-SGCR-SWL) 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) and [SWU](#)

$K_{rg}$ , [SGCR](#), (1.0-SWCR) 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.

---

ECLIPSE 300

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

```
SCALECRS  
YES /
```

## SCALELIM Sets saturation table scaling limits

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [ENDNUM](#) (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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $\geq 5$ :

```

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\geq 5$ :

```
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

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

- Kinetic first order rate constant for laminar flow

UNITS: kg/m<sup>2</sup>/day (METRIC), lb/ft<sup>2</sup>/day (FIELD), gm/cm<sup>2</sup>/hr (LAB)

A value of 0, or a negative value, means dissolution turned off for laminar flow

DEFAULT: 0
- Kinetic first order rate constant for turbulent flow

UNITS: kg/m<sup>2</sup>/day (METRIC), lb/ft<sup>2</sup>/day (FIELD), gm/cm<sup>2</sup>/hr (LAB)

A value of 0, or a negative value, means dissolution turned off for turbulent flow

DEFAULT: 0
- Kinetic fourth order rate constant

UNITS: kg/m<sup>2</sup>/day (METRIC), lb/ft<sup>2</sup>/day (FIELD), gm/cm<sup>2</sup>/hr (LAB)

A value of 0, or a negative value, means dissolution turned off for turbulent flow

DEFAULT: 0
- Diffusion coefficient of calcite in conduit water flow

UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/hr (LAB)

This input cannot be defaulted to zero, rather a positive small value should be given.

DEFAULT: 0.00001 (all unit sets)
- Equilibrium concentration beyond which dissolution of karst conduit will stop.

UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)

DEFAULT: no default
- Relative equilibrium concentration fraction threshold beyond which dissolution kinetics changes from first order to higher order

UNITS: dimensionless

DEFAULT: 1.0
- Maximum diameter size beyond which dissolution stops.

UNITS: m (METRIC), ft (FIELD), cm (LAB)

DEFAULT: no default

8. Density of carbonate rock

UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#), [SCDATAB](#) 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 [SCDATABG](#) 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 [SCDATAB](#) 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 [SCDATABG](#) 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 [SCDATABG](#) keyword.  
DEFAULT: 0
7. ECLIPSE 100 only for Karst aquifer [NTSCDE](#) The number of Karst Aquifer dissolution tables to be entered in the [SCDETAB](#) keyword.  
DEFAULT: 0

### Example

```
SCDPDIMS
2 20 3 25 1 15 10 /
```

## SCDPTAB Scale deposition tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WSCTAB](#) 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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), gm/sm<sup>3</sup> (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:

```

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  /

```

## SCDPTRAC Sets a tracer to be associated with the sea water fraction for scale deposition

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
SCDPTRAC
SWA /
```

## SCFDIMS

## Conductive fracture dimensions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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. MSCBNP The maximum number of grouping bins due to the value of cell permeability.  
DEFAULT: 3

## Example

```
SCFDIMS
1* 5 1 /
```

## SCONINJE Control data for secondary injection wells

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: sm<sup>3</sup>/day (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB), sm<sup>3</sup>/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.

Thermal

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:  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{stb}/\text{day}/\text{psi}$  or  $\text{Mscf}/\text{day}/\text{psi}$  (FIELD),  $\text{scc}/\text{hr}/\text{atm}$  (LAB),  $\text{sm}^3/\text{day}/\text{atm}$  (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

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: No target or limit.

6. Water rate target or upper limit.

UNITS: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: No target or limit.

7. 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),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: No target or limit.

8. Liquid rate target or upper limit.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: No target or limit.

9. 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),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{rm}^3/\text{day}$  (PVT-M)

DEFAULT: No target or limit.

10. Segment pressure lower limit.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: Atmospheric pressure.

If the secondary well (segment) pressure falls below this value, the secondary well will be shut.

11. The maximum production rate per unit pressure difference between the current segment pressure and the lower pressure limit defined by item 10.

UNITS:  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{stb}/\text{day}/\text{psi}$  or  $\text{Mscf}/\text{day}/\text{psi}$  (FIELD),  $\text{scc}/\text{hr}/\text{atm}$  (LAB),  $\text{sm}^3/\text{day}/\text{atm}$  (PVT-M)

DEFAULT: No upper limit.

12. 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.

13. The minimum pressure limit to be applied to the alternative segment number designated for minimum pressure control.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), 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 /
/
```

## SCREF

## 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:

$$\rho_c = \frac{\rho_{ref,c}}{\left[1 - c_{P_c}(P - P_{ref,c})\right] \left[1 + c_{T1_c}(T - T_{ref,c})\right]} \quad \text{Eq. 3.166}$$

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  /
```

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## SCVD

## Initial coal solvent concentration versus depth tables

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (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  $\geq$  2

```
SCVD
 7000 94.0
10000 94.0 /
```

## SDENSITY

## Miscible gas density at surface conditions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (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 /
```

## SDREF

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.167}$$

where:

$\rho_{ref,c}$  is the component reference solid density, specified with keyword [SDREF](#)

$c_{P_c}$  is the component solid compressibility, specified with keyword [SCREF](#)

$P_{ref_c}$  is the component reference pressure, specified with keyword [SPREF](#)

$T_{ref_c}$  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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

DEFAULT: 799.2112 kg/m<sup>3</sup> (METRIC) for each component, except for the [CO2STORE](#) options where components NaCl, CaCl<sub>2</sub> and CaCO<sub>3</sub> are assigned default values of 2170 kg/m<sup>3</sup>, 2150 kg/m<sup>3</sup> and 2709.89 kg/m<sup>3</sup>, 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:

SDREF  
2\* 43.887 /

## SDRGO

## Initial gas to oil saturation direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword specifies the initial gas 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 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.
3. Secondary hysteresis saturation direction.
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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
3. Secondary hysteresis saturation direction.
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/
```

## SDROG

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 [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 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.
3. Secondary hysteresis saturation direction.
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/
```

## SDROW

## Initial oil to water saturation direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword specifies the initial oil 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 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.
3. Secondary hysteresis saturation direction.
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/
```

## SDRWG

## Initial water to gas saturation direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
3. Secondary hysteresis saturation direction.
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/
```

## SDRWO

## Initial water to oil saturation direction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.
3. Secondary hysteresis saturation direction.
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:

```
SDRWO
12*1 12*2 12*3 24*4/
```

## SECTBC

## Sector boundary conditions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```

-- Specify sector
SECTOR
  30 40 30 40 1 30 /
-- No flow boundary conditions
SECTBC
  NOFLOW /

```

## Example creating the FLUX file

If a full field model (BASE.DATA) 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

## Sector model

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$1 \leq IX1 \leq IX2 \leq \text{NDIVIX}$$

$$1 \leq JY1 \leq JY2 \leq \text{NDIVIY} \text{ (where NDIVIX and NDIVIY are specified in RUNSPEC keyword [DIMENS](#))}$$

The data should be terminated with a slash (/).

See also the keywords [GDFILE](#) and [IMPFIL](#) 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

```

RUNSPEC
-- IX1-IX2 JY1-JY2 KZ1-KZ2
SECTOR
  10 19 21 40 1 20 /
GRID
-- Read grid
GDFILE
'../FULLFIELD' /
-- Read grid arrays
IMPFIL
'../FULLFIELD.INIT' /
PERMX PERMY PERMZ PORO /

```

## SEPARATE Requests run summary output to separate RSM file

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

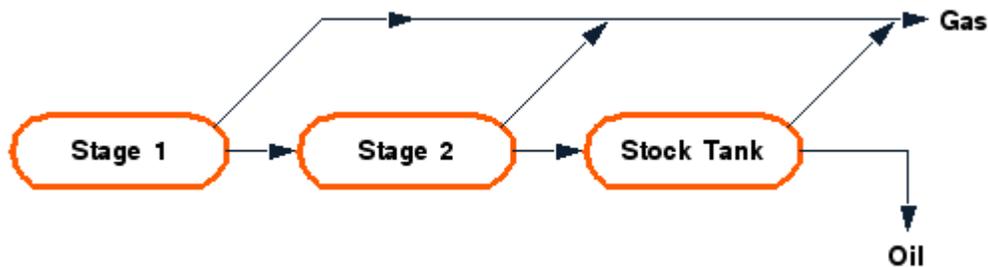


Figure 3.15. A three stage separator, with the stock tank as the third stage

### 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 /
/
```

## SEPVALS

## Bo and Rs values from separator test

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [NGMAXZ](#) separators, where [NGMAXZ](#) 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 [COMPFLSH](#).

## 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
```

```

    PLAT-A      SEP1      /
    PLAT-B      SEP2      /
  /

```

Simulate:

```

TIME or TSTEP or DATES
.
.

```

Change conditions of SEP1:

```

SEPVALS
-- separator name  Bob    Rsi
   SEP1           1.276  0.526  /
/

```

Continue simulation:

```

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,  $P_{bub}$ .
- A differential liberation, to determine the change in fluid properties and the amount of gas liberated as the reservoir pressure declines from  $P_{bub}$ .
- A separator test, in which the fluid sample is flashed from bubble point conditions ( $P_{bub}$ ,  $T_{res}$ ) 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 ( $B_{ob}$ ) 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 ( $R_{si}$ ) 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 [PVTO](#) keyword is prepared by a combined PVT analysis of the differential liberation and separator test results. Thus the values in the PVTO tables are specific to a particular set of separator conditions. ECLIPSE assumes that the PVTO tables were prepared using the original separator conditions, and it is up to you to ensure that the first entry of SEPVALS contains the values of  $B_{ob}$  and  $R_{si}$  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 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  $R_s$  values by a certain amount.

In fact, ECLIPSE does not transform the PVTO tables. All calculations of fluid flow and material balance within the reservoir are performed using the 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:

$$Q_o^* = T_{oil-oil} Q_o \quad \text{Eq. 3.168}$$

$$Q_g^* = T_{gas-oil} Q_o + Q_g \quad \text{Eq. 3.169}$$

Where the transformation coefficients are:

$$T_{oil-oil} = \frac{B_{ob}}{B_{ob}^*} \quad \text{Eq. 3.170}$$

$$T_{gas-oil} = T_{oil-oil} R_{si}^* - R_{si} \quad \text{Eq. 3.171}$$

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.

## SFOAM

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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<sup>3</sup> (METRIC), lb/Mscf (FIELD), gm/scc (LAB)

If item 1 of [FOAMOPTS](#) is set to 'WATER':

UNITS: kg/sm<sup>3</sup> (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 [NDIVY](#)), and finally the Z axis index (from 1 to [NDIVZ](#)). [NDIVIX](#), [NDIVY](#) and [NDIVZ](#) 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 adsorped 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, [NDIVY](#)=1 and [NDIVZ](#)=2 specified in [RUNSPEC](#) keyword [DIMENS](#)

```
SFOAM
 10*0.0
  9*0.0 0.5 /
```

## SGAS

## Initial gas saturations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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
48*0 /

```

## SGCR

## Scaled critical gas saturations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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 ENDSALE, 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 ENDSALE 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.

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 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

```

----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 /

```

## SGCWMIS

**Miscible critical gas saturation tables**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
SGCWMIS
 .00 .00
 1.00 .00 /
 .00 .00
 .25 .05
 1.00 .05 /
```

## SGF3

## Three-phase gas saturation functions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $1.0 \times 10^{-6}$  and  $1.0 \times 10^{-20}$  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

```

.60	.30	.50	3.0
.78	1.00	1.00	3.9 /

## SGF32D

## 2D table of gas relative permeability

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

The gas/oil capillary pressure must be provided using the [PCG32D](#) keyword.

ECLIPSE 300

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 ≥ 11:

```

SGF32D
-----SOIL-----
--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 /

```

```
0.40 0.100 0.065 0.031 0.015 0.002 0.002 0.001 0.000 /
0.50 0.060 0.025 0.014 0.002 0.001 0.001 0.000 /
0.65 0.020 0.012 0.001 0.001 0.001 0.000 /
0.80 0.010 0.001 0.001 0.001 0.000 /
1.00 0.000 0.000 0.000 0.000 /
/
```

## SGFN

## Gas saturation functions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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).

ECLIPSE 100

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:

```
SGFN
.0000 .0000 .0000
.0400 .0000 .2000
.1000 .0220 .5000
.2000 .1000 1.0000
.3000 .2400 1*
.5000 .4200 1*
.6000 .5000 3.0000
.7000 .8125 3.5000
.7800 1.0000 3.9000
/
.00 .0000 0
.12 .0220 0
.22 .1000 0
.32 .2400 0
.52 .4200 0
.62 .5000 0
.72 .8125 0
.80 1.0000 0
/
```

## SGL,ISGL Scaled connate gas saturations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----RUNSPEC section
DIMENS
 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 /

```

## SGLPC

## Scaled connate gas saturations, for Pc curves only

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The SGLPC and ISGLPC keywords scales 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 [GAS](#) in the RUNSPEC section). The keyword [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 [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 [DIMENS](#):

```
SGLPC
7*0.0 7*0.02 7*0.0 /
```

## SGOF

## Gas/oil saturation functions versus gas saturation

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ( $k_{rog}$  at  $S_g = 0$ ) must be the same as the first value in the  $k_{row}$  column in keyword SWOF (that is  $k_{row}$  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 SWF32D.

**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 ≥ 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
.00        .0000      1.00      0
.05        .0220      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

```

## SGU,ISGU

**Scaled saturation table maximum gas saturations**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

----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 /

```

## SGWCR

## Scaled critical gas saturations in water

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword specifies the critical gas saturation in water 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 gas-water relative permeabilities are specified using the [SGF3](#) keyword. The critical gas saturation in oil is obtained from [SGCR](#).

Seven keywords are provided to allow the critical gas saturation in water for each grid cell to be scaled independently. These keywords are:

**SGWCR** Critical gas saturation in water for the non-directional saturation function tables

**SGWCRX** Critical gas saturation in water for the +X face of each grid cell

**SGWCRX-** Critical gas saturation in water for the -X face of each grid cell

**SGWCY** Critical gas saturation in water for the +Y face of each grid cell

**SGWCY-** Critical gas saturation in water for the -Y face of each grid cell

**SGWCRZ** Critical gas saturation in water for the +Z face of each grid cell

**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 SGWCR 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 'IRREVERS' in the ENDSCALE keyword in RUNSPEC, the critical gas saturations in water for flow across the +X and -X faces of a grid cell are set using the keywords SGWCRX and SGWCRX-. Similarly the keywords SGWCY and SGWCY- are used for the +Y and -Y faces and the keywords SGWCRZ and 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 SGWCR keyword.

When the end-point scaling is requested to be directional and reversible, using the switch 'DIRECT' but not 'IRREVERS' in ENDSCALE, the keyword SGWCRX defines the critical gas saturations in water for the +X and -X faces of each grid cell. Keywords SGWCY and 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 SGWCR keyword.

If ENDSCALE is specified in RUNSPEC without the switch 'DIRECT', none of the directional forms of the SGWCR keyword are allowed. The 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 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
```

## SGWFN

## Gas-water saturation functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
SGWFN
.0000 .0000 1.0000 .0000
.0400 .0000 0.9000 .2000
.1000 .0220 0.8000 .5000
.2000 .1000 0.5000 1.0000
.3000 .2400 1* 1*
.5000 .4200 1* 1*
.6000 .5000 0.1125 3.0000
.7000 .8125 0.0000 3.5000
.7800 1.0000 0.0000 3.9000
/
.00 .0000 1.0 0
.12 .0220 0.85 0
.22 .1000 0.6 0
.32 .2400 0.45 0
.52 .4200 0.25 0
.62 .5000 0.05 0
.72 .8125 0.0 0
.80 1.0000 0.0 0
/
```

**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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$C_\gamma = C \left( \frac{3n+1}{4n} \right)^{\frac{n}{n-1}}$$

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 /
```

## SIGMA

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

ECLIPSE 100

ECLIPSE 100

## 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:

$$\sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 3.172}$$

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: m<sup>-2</sup> (METRIC), ft<sup>-2</sup> (FIELD), cm<sup>-2</sup> (LAB), m<sup>-2</sup> (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.

## SIGMAGD

## Matrix-fracture coupling for oil-gas gravity drainage

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 3.173}$$

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 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.

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 will drain through a transmissibility based on [SIGMAGD](#) and regions swept by water will be produced through a transmissibility based on [SIGMA](#).

The [SIGMAGDV](#) 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:

$$\Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \quad \text{Eq. 3.174}$$

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.

UNITS: m<sup>-2</sup> (METRIC), ft<sup>-2</sup> (FIELD), cm<sup>-2</sup> (LAB), m<sup>-2</sup> (PVT-M)

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 [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 will be based on the sigma value supplied by the `SIGMA` or `SIGMAV` keywords, unless `LTOSIGMA` is specified.

When the `SIGMAGD` values are output (mnemonic `SIGMAGDV` in the `RPTGRID` keyword) the first `NDIVIZ / 2` layers are copied into the lower half of the grid.

Refer to "Gravity imbibition/drainage" in the *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 `SIGMAGDV` to `SIGMAV`

```
COPY
SIGMAV SIGMAGDV 1 8 1 1 1 2 /
/
```

### Example 3

Divide the `SIGMAGDV` by 4.

```
MULTIPLY
SIGMAGDV 0.25 1 8 1 1 1 2 /
/
```

## SIGMATH

## Dual porosity matrix-fracture conductivity coupling

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

# SIGMAV

## Dual porosity matrix-fracture coupling

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\Sigma = 4 \left( \frac{1}{l_x^2} + \frac{1}{l_y^2} + \frac{1}{l_z^2} \right) \tag{Eq. 3.175}$$

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<sup>-2</sup> (METRIC), ft<sup>-2</sup> (FIELD), cm<sup>-2</sup> (LAB), m<sup>-2</sup> (PVT-M)

### ECLIPSE 100

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.

### ECLIPSE 300

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 0.12
1.0  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**

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## 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

```

-----GRID section
SKIP
CARFIN
'HUGE' 1 20 1 20 1 20 100 100 100 /
ENDFIN
ENDSKIP

```

## SKIP100

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## Skip keywords for ECLIPSE 100

This keyword causes ECLIPSE 100 to ignore all subsequent keywords, up to the next ENDSKIP keyword. This keyword is ignored by ECLIPSE 300.

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

```

----Props section
SKIP100
INCLUDE
'COMPOSITIONAL.PVO' /
ENDSKIP

```

## SKIP300

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## Skip keywords for ECLIPSE 300

This keyword causes ECLIPSE 300 to ignore all subsequent keywords, up to the next ENDSKIP keyword. This keyword is ignored by ECLIPSE 100.

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

```
----Props section
SKIP300
INCLUDE
'BLACK.PVO' /
ENDSKIP
```

## SKIPREST Skip subsequent keywords until the restart time

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [VFPPROD](#), [VFPINJ](#))
- Reporting instructions  
(Keywords [RPTSCHED](#), [RPTRST](#) 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 [WSEGDFIN](#)).
- 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
```

## SKIPSTAB

**Request the fast flash option**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
SLAVES
--  slave  datafile  machine  directory
--  name   root      hostname  of data file
--
--  'upper' 'base'   '*'      '/usr/models/upper' /
--  'lower' 'base'   '*'      '/usr/models/lower' /
/
```

## SLGOF

## Gas/oil saturation functions versus liquid saturation

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $S_1 = S_{wco} + S_{or}$  ( $S_{wco}$  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 ( $k_{rog}$  at  $S_g=0$ ) must be the same as the first value in the  $k_{row}$  column in keyword SWOF (that is  $k_{row}$  at  $S_o = 1 - S_{wco}$ ). 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 ≥ 10:

```

SLGOF
-- S1      Krg      Krog      Pcog
.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 [SOLID](#) option. If this keyword is used then keyword [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

```
SMF
300*0.00
300*0.00
300*1.00
/
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## SMRYDIMS **Maximum number of summary quantities**

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## SOCRS

## Scaled critical oil-in-water saturations at miscible conditions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#) tables.

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
18*0.05 /

```

## SOF2

## Oil saturation functions (two-phase)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $k_{rp} = k_m$ .

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 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 ≥12:

```
SOF2
0      0
.2     0
.38    0
.4     .004
.48    .02
.5     .036
.58    .1
.6     .146
.68    .33
.7     .42

.74    .60
.78    1.00 /
.0000  .0000
.2000  .0000
.3800  .0043
.4000  .0048
.4800  .0529
.5000  .0649
.5800  .1130
.6000  .1250
.6800  .3450
.7000  .4000
.7400  .7000
.7800  1.0000 /
```

## SOF3

## Oil saturation functions (three-phase)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

- 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  $1 - S_{wco}$ , where  $S_{wco}$  is the connate water saturation.
- 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.
- 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

```
SOF3
0      0      0
0.2    0      1*
1      1      0.6 /
```

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 ( $S_o = 1 - S_{wco}$ ) 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 SOF32D 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 ≥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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 >NSSFUN ≥11:

```

SOF32D
-----SWAT-----
--SGAS
0.00  0.22  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
0.05  1.000 0.625 0.345 0.207 0.113 0.083 0.053 0.023 0.002 0.001 0.000/
0.10  0.555 0.337 0.210 0.110 0.078 0.047 0.021 0.004 0.001 0.000 /
0.15  0.330 0.212 0.106 0.074 0.042 0.019 0.003 0.002 0.000 /
0.20  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 /
0.35 0.010 0.001 0.001 0.001 0.000 /
0.40 0.000 0.000 0.000 0.000 /
/
```

## SOGCR

## Scaled critical oil-in-gas saturations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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 ENDSALE, 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 ENDSALE 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, 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, ISOGCRX-, ...) 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

```

----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 /

```

## SOIL

## Specifies cell initial oil saturation values

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## SOILR

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## SOLID

### 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*.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## SOLIDADS Solid adsorption function

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (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 ≥6

```

SOLIDADS
.0000 .00000
.0000 .00001
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0010 .00010 /

```

## SOLIDMMC

## Mobility multiplier as a function of adsorbed solid concentration

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises NTSFUN (see keyword [TABDIMS](#)) 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 [TABDIMS](#)).

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
.0000 1.00
.0001 0.99
.0003 0.81
.0005 0.75
.0008 0.36
.0010 0.10 /

.0000 1.00
.0010 0.10 /
```

## SOLIDMMS

## Mobility multiplier as a function of adsorbed solid saturation

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

[SOLIDMMC](#) 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 /
```

## SOLUAQA

## Specify properties of water multi-component gas system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub>, 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`), atma (`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<sup>3</sup>/sm<sup>3</sup> (`METRIC`), Mscf/stb (`FIELD`), scc/scc (`LAB`), sm<sup>3</sup>/sm<sup>3</sup> (`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
SOLUAQA
  2/
  1.0132    0.3038
  533.7694  36.5522
  1013.2500 43.4860
/
  4/
  1.0132    0.2947
  533.7694  35.4555

```

```
1013.2500 42.1814
/
/ null record terminates table
```

## Recommended CO<sub>2</sub> solubility data (metric units)

```
--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
```

## SOLUBILI

## Specify properties of water-gas system

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 CO<sub>2</sub> ([CO2SOL](#)) or H<sub>2</sub>S ([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: sm<sup>3</sup>/sm<sup>3</sup> ([METRIC](#)), Mscf/stb ([FIELD](#)), scc/scc ([LAB](#)), sm<sup>3</sup>/sm<sup>3</sup> ([PVT-M](#))
3. The formation volume of water saturated with gas at the specified pressure.  
UNITS: rm<sup>3</sup>/sm<sup>3</sup> ([METRIC](#)), rb/stb ([FIELD](#)), rcc/scc ([LAB](#)), rm<sup>3</sup>/sm<sup>3</sup> ([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 CO<sub>2</sub>

---

**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 CO<sub>2</sub>, [[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 "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

## Example

```

SOLUBILI
--Pres-----Rs-----FVF-----Visc-----Compress
-[barsa]---[sm3/sm3]--[rm3/sm3]--[cP]-----[1/bars]
  1.013      0.7537      1.0057      .30      .4E-4
  54.29      23.09      1.0037      .30      .4E-4
 107.6      30.05      1.0015      .30      .4E-4
 160.8      32.65      .99923      .30      .4E-4
 214.1      33.94      .99701      .30      .4E-4
 267.4      34.88      .99482      .30      .4E-4
 320.7      35.78      .99266      .30      .4E-4
 373.9      36.68      .99052      .30      .4E-4
 427.2      37.59      .98842      .30      .4E-4
 480.5      38.49      .98635      .30      .4E-4
 533.8      39.39      .98431      .30      .4E-4
 587.0      40.29      .98230      .30      .4E-4
 640.3      41.20      .98032      .30      .4E-4
 693.6      42.10      .97837      .30      .4E-4
 746.9      43.00      .97644      .30      .4E-4
 800.1      43.91      .97455      .30      .4E-4
 853.4      44.81      .97268      .30      .4E-4
 906.7      45.71      .97084      .30      .4E-4
 960.0      46.61      .96903      .30      .4E-4
1013.      47.52      .96725      .30      .4E-4
/

```

## SOLUBILS

Specify properties of water-CO<sub>2</sub> system with salt dependencies

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub> is at its minimum for a given salinity. The simulator does not assume that the minimum CO<sub>2</sub> in water is zero; if you want to specify that the amount of CO<sub>2</sub> 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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (PVT-M)

2. Brine surface density.

UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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<sub>2</sub> solution ratio at the specified pressure and salt concentration.

This is expressed as the volume of CO<sub>2</sub> at surface conditions per volume of water at surface conditions for a salt-saturated solution.

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

- The formation volume factor of water saturated with  $\text{CO}_2$  at the specified pressure and salt concentration.

UNITS:  $\text{rm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{rm}^3/\text{sm}^3$  (PVT-M)

- The viscosity of water saturated with  $\text{CO}_2$  at the specified pressure and salt concentration.

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

- The compressibility of water saturated with  $\text{CO}_2$  at the specified pressure and salt concentration.

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

## Example

```

--Metric units
SOLUBILS
  0.0 999.0 /
  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
/
  20.0 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-CO<sub>2</sub> system with temperature dependencies

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub> is at its minimum for a given temperature. The simulator does not assume that the minimum CO<sub>2</sub> in water is zero; if you want to specify that the amount of CO<sub>2</sub> 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<sub>2</sub> solution ratio at the specified pressure and temperature.

This is expressed as the volume of CO<sub>2</sub> at surface conditions per volume of water at surface conditions.

UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

3. The formation volume factor of water saturated with CO<sub>2</sub> at the specified pressure and temperature.

UNITS: rm<sup>3</sup>/sm<sup>3</sup> (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

4. The viscosity of water saturated with CO<sub>2</sub> at the specified pressure and temperature.

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

5. The compressibility of water saturated with CO<sub>2</sub> at the specified pressure and temperature.

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

## Example

```
--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-CO<sub>2</sub> system with H<sub>2</sub>S dependencies

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub> is at its minimum for a given H<sub>2</sub>S aqueous mole fraction. The simulator does not assume that the minimum CO<sub>2</sub> in water is zero; if you want to specify that the amount of CO<sub>2</sub> 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<sub>2</sub>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), atma (LAB), atma (PVT-M)

2. The dissolved CO<sub>2</sub> solution ratio at the specified pressure and H<sub>2</sub>S concentration.

This is expressed as the volume of CO<sub>2</sub> at surface conditions per volume of water at surface conditions for a salt-saturated solution.

UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

3. The formation volume factor of water saturated with CO<sub>2</sub> at the specified pressure and H<sub>2</sub>S concentration.

UNITS: rm<sup>3</sup>/sm<sup>3</sup> (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

4. The viscosity of water saturated with CO<sub>2</sub> at the specified pressure and H<sub>2</sub>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<sub>2</sub> at the specified pressure and H<sub>2</sub>S concentration.

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

### Example

```

--Metric units
SOLUCO2H

```

```
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  
/  
0.005/  
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  
/  
/ null record terminates table
```

## SOLUH2SC

## Specify properties of water-H<sub>2</sub>S system with CO<sub>2</sub> dependencies

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>2</sub>S is at its minimum for a given CO<sub>2</sub> aqueous mole fraction. The simulator does not assume that the minimum H<sub>2</sub>S in water is zero; if you want to specify that the amount of H<sub>2</sub>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<sub>2</sub> 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<sub>2</sub>S solution ratio at the specified pressure and CO<sub>2</sub> concentration.

This is expressed as the volume of H<sub>2</sub>S at surface conditions per volume of water at surface conditions for a salt-saturated solution.

UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

3. The formation volume factor of water saturated with H<sub>2</sub>S at the specified pressure and CO<sub>2</sub> concentration.

UNITS: rm<sup>3</sup>/sm<sup>3</sup> (METRIC), rb/stb (FIELD), rcc/scc (LAB), rm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

4. The viscosity of water saturated with H<sub>2</sub>S at the specified pressure and CO<sub>2</sub> 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<sub>2</sub>S at the specified pressure and CO<sub>2</sub> concentration.

UNITS: 1/bars (METRIC), 1/psi (FIELD), 1/atm (LAB), 1/atm (PVT-M)

See also "[CO<sub>2</sub>/H<sub>2</sub>S solution in the aqueous phase](#)" in the *ECLIPSE Technical Description*.

### Example

```

--Metric units
SOLUH2SC

```

```
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  
/  
0.005/  
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  
/  
/ null record terminates table
```

## SOLVCONC Initial coal solvent concentration

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB),  $\text{sm}^3/\text{m}^3$  (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](#)

```
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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $NSOLX \times NSOLY \times NSOLZ \ll NX \times NY \times NZ$ .

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

```

RUNSPEC

DIMENS
 1000 1 1 /

SOLVDIMS
 10 10 10 /...

GRID

SOLVNUM
... /

```

## SOLVDIRS Overrides the solver principal directions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 diskling 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $[1, NSOLX]$  by  $[1, NSOLY]$  by  $[1, NSOLZ]$ .

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 REFINE and ENDFIN pair.

ECLIPSE 300

**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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## SOLWTAB Solid wettability table

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## SOLWW

## Solid wettability weighting factor

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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* 1* SOL SOL SOL /
MW
18.0 32.0 45.6 78.0 88.0 98.0 /
SOLWW
1* 1* 1* 1* 1.0 0.0 /

```

## SOMGAS

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

```
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  /
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-SOGR (1 - 0.25 - 0.1)

## SOMWAT

## Minimum oil saturation for the STONE I model

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $1 - S_{gco} - S_{owcr}$  where:

$S_{gco}$  is the connate gas saturation

$S_{owcr}$  is 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.)

2. The corresponding minimum oil saturation.

The minimum oil saturation at connate water (first point) should not exceed  $S_{ogcr}$ , the residual oil-to-gas saturation in the corresponding [SOF3](#) table. ( $S_{ogcr}$  is the largest oil saturation with  $k_{rog}$  of zero, in column 3 of the [SOF3](#) keyword.)

The minimum oil saturation at  $1 - S_{gco} - S_{owcr}$  (last point) should not exceed  $S_{owcr}$ , 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  $S_{wco}$  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.

---

See "[Saturation functions](#)" in the *ECLIPSE Technical Description* for more information.

## Example

With NTSFUN=1 and NSSFUN ≥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
/
```

## SOR

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 `RPTSOL`, `RPTSCHED`, the restart file `RPTRST`, and the summary file, using the following mnemonics and keywords:

	RPTSOL/RPTSCHED/RPTRST	SUMMARY vector
Residual oil composition	ROMLS	BROMLS
Residual oil saturation	SOILR	BSOILR
Residual oil compressibility (see <code>SOROPTS</code> )	COILR	BCOILR
Mobile oil saturation	SOILM	BSOILM

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This keyword may be used with the coal bed methane model (see "[Coal bed methane model](#)" in the *ECLIPSE Technical Description*).

The keyword specifies 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.

## SORMW1

## Residual oil saturation for compositional

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#)

## SOROPTS Residual oil model options

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\frac{1}{SOILR} \cdot \frac{dSOILR}{dP} = -COILR - \frac{1}{PORV} \cdot \frac{dPORV}{dP}$$

where  $COILR$  is the residual oil compressibility:

$$COILR = -\frac{1}{VOILR} \cdot \frac{dVOILR}{dP}$$

Three compressibility models are available:

### Constant SOR model

$$\text{'ZERO'} \quad \frac{1}{SOILR} \cdot \frac{dSOILR}{dP} = 0$$

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

$$\text{'CONSTANT'} \quad COILR = CONSTANT$$

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## SOWCR

## Scaled critical oil-in-water saturations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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 ENDSALE, 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 ENDSALE 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 /

```

## SPECGRID Specification of grid characteristics

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword repeats the specification of dimensions and other properties describing the simulation grid, as defined in the RUNSPEC section.

The SPECGRID keyword 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

*ECLIPSE 100 only*

4. The number of reservoirs (NUMRES)

Each reservoir has its own coordinate system.

DEFAULT: 1

*ECLIPSE 100 only*

5. Type of coordinates

T Cylindrical (radial) coordinates.

F Cartesian coordinates.

DEFAULT: F

*ECLIPSE 300*

**Note:** Only the first 3 items have any use in this simulator. The 4<sup>th</sup> and 5<sup>th</sup> items can be read in, but are ignored.

See also keywords [DIMENS](#), [NUMRES](#), [COORDSYS](#) and [COORD](#).

### Example

```
SPECGRID
5 3 4 2 T /
```

## SPECHA

## Oil component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_o = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.176}$$

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

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.177}$$

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:

- 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

```
SPECHA  
0.55 0.57 0.58 0.61 /
```

## SPECHB

## Oil component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_o = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.178}$$

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

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.179}$$

where

$C_{p1c}$  is the first coefficient of the component liquid specific heat entered with the [SPECHA](#) 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](#).

---

**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:

- liquid hydrocarbon component specific heats,  $N_c$  values, one for each component

UNITS: kJ/kg/K/K (METRIC), Btu/lb/°R/°R (FIELD), J/gm/K/K (LAB), kJ/kg/K/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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
/
```

## SPECHG

## Gas component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_g = \sum_{c=1}^{N_c} y_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.180}$$

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

$$h_c = h_{cs} + C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.181}$$

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 [TABDIMS](#). 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 /
```

## SPECHH

## Gas component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

This keyword can only be used with the [THERMAL](#) option.

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:

$$h_g = \sum_{c=1}^{N_c} y_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.182}$$

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 [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 the [SPECHG](#) keyword, and

$C_{p2c}$  is the second coefficient of the component gaseous specific heat entered with this 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.

---

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/K (METRIC), Btu/lb/°R/°R (FIELD), J/gm/K/K (LAB), kJ/kg/K/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

```
SPECHH  
0.5E-4 0.5E-4 0.5E-4 0.6E-4 /
```

## SPECHS

## Solid component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_s = \sum_{c=1}^{N_c} s_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.183}$$

where

$s_c$  is the mole fraction of component  $c$  in the oil phase, and

$mw_c$  is the component molecular weight.

The component solid enthalpy  $h_c$  has the form

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.184}$$

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:

- 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_s = \sum_{c=1}^{N_c} x_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.185}$$

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  $h_c$  has the form

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.186}$$

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:

- liquid hydrocarbon component specific heats,  $N_c$  values, one for each component

UNITS: kJ/kg/K/K (METRIC), Btu/lb/°R/°R (FIELD), J/gm/K/K (LAB), kJ/kg/K/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

```
SPECHT
0.005 0.007 0.008 0.006 /
```

## SPECHW1 Water component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_w = \sum_{c=1}^{N_c} a_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.187}$$

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

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.188}$$

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 H<sub>2</sub>O component enthalpy is computed internally. In a [CO2STORE](#) run with the H<sub>2</sub>O component specified in [CNAMES](#), defaulting the value for H<sub>2</sub>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 [KWTABTn](#), 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*.

## Example

For four components

```
SPECHW1  
0.55 0.57 0.58 0.61 /
```

## SPECHW2 Water component specific heat

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$h_w = \sum_{c=1}^{N_c} a_c \cdot h_c \cdot mw_c \quad \text{Eq. 3.189}$$

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

$$h_c = C_{p1c} \cdot (T - T_{st}) + \frac{1}{2} \cdot C_{p2c} \cdot (T - T_{st})^2 \quad \text{Eq. 3.190}$$

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<sub>2</sub>O component enthalpy is computed internally. In a [CO2STORE](#) run with the H<sub>2</sub>O component specified in [CNAMES](#), defaulting the value for H<sub>2</sub>O means that the enthalpy is computed similarly to a standard case.

---

The data record contains one value for each component:

- Liquid hydrocarbon component specific heats,  $N_c$  values, one for each component.

UNITS: kJ/kg/K/K (METRIC), Btu/lb/°R/°R (FIELD), J/gm/K/K (LAB), kJ/kg/K/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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/m<sup>3</sup>/K ([METRIC](#)), Btu/ft<sup>3</sup>/°R ([FIELD](#)), J/cc/K ([LAB](#)), kJ/m<sup>3</sup>/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
/

```

## SPOLY

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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<sup>3</sup> (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 "[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 adsorped 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 NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword [DIMENS](#):

```
SPOLY
 10*0.0
  9*0.0 25.0 /
```

## SPREF

## Reference pressures for solid phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [SOLID](#) in the [RUNSPEC](#) section).

The expression used to calculate component solid densities is the following:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.191}$$

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 this keyword

$T_{ref_c}$  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: barsa ([METRIC](#)), psia ([FIELD](#)), atma ([LAB](#)), atma ([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

```
SPREF
1527.4
1527.4 /
```

## SSFN

## Gas/solvent saturation functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ( $K_{rg}^*$  and  $K_{rs}^*$ ).

The relative permeabilities of the 2 components are given by:

$$K_{rg} = K_{rgt}(S_{gas} + S_{solvent})K_{rg}^* \quad \text{Eq. 3.192}$$

$$K_{rs} = K_{rgt}(S_{gas} + S_{solvent})K_{rs}^*$$

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 >NSSFUN ≥6

```

SSFN
--      Krg*      Krs*
0.0      0.0      0.0
1.0      1.0      1.0
/
0.0      0.0      0.0
0.2      0.1      0.2
0.4      0.2      0.4
0.6      0.4      0.5
0.8      0.7      0.8
1.0      1.0      1.0
/

```

## SSHIFT

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

```
-- 3-Parameter EoS Shift Coefficients
SSHIFT
-0.07361 -0.15427 -0.05559 -0.02573 -0.08493 -0.16602 /
```

## Example 2

```
--With two reservoir EoS regions
RTEMP
212.0 /
223.0 /
```

## SSHIFTS

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## SSOL

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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 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 "[Solvent model](#)" in the *ECLIPSE Technical Description* for more information.

## Example

With NDIVIX=10, NDIVIY=1 and NDIVIZ=2 specified in RUNSPEC keyword [DIMENS](#):

```
SSOL
 10*0.0
  9*0.0 0.5 /
```

## SSOLID

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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)

DEFAULT: 1 Jan 1983 00:00:00

DEFAULT: 1 Jan 1990 00:00:00

*ECLIPSE 100*

*ECLIPSE 300*

### Example

```
START
12 FEB 1987 /
```

## STCOND

## Specify standard conditions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 blank record containing just a slash.

### Example

```
STEST
'PROD-S*' 5 /
/
```

## STHERMX1 Thermal expansion coefficient for solid phase

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.193}$$

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](#),

$c_{T1_c}$  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* /
```

## STOG

## Oil-gas surface tension versus pressure

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 WELSPPCS 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:

$$P_{cog} \text{ (oil pressure)} = P_{cog} \text{ (input by user)} \frac{ST \text{ (oil pressure)}}{ST \text{ (reference pressure)}} \quad \text{Eq. 3.194}$$

**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 ≥ 11:

```
STOG
--
-- REFERENCE PRESSURE
--
--      5545 /
--
-- 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      /
```

## 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](#).

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## STONE1

## Request Stone three-phase oil relative permeability model

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\text{MIN}(S_{\text{owcr}}, S_{\text{ogcr}})$ , 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
```

## STONE1EX Stone's first three-phase oil relative permeability model exponent term

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## STONE2

## Request Stone three-phase oil relative permeability model

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## STONEPAR **Modify the Stone I three-phase oil relative permeability**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$S_{om} = S_{ogcr} X_g^A + S_{owcr} (1 - X_g)^A + B S_{gm}^2 X_g (1 - X_g) \quad \text{Eq. 3.195}$$

where:

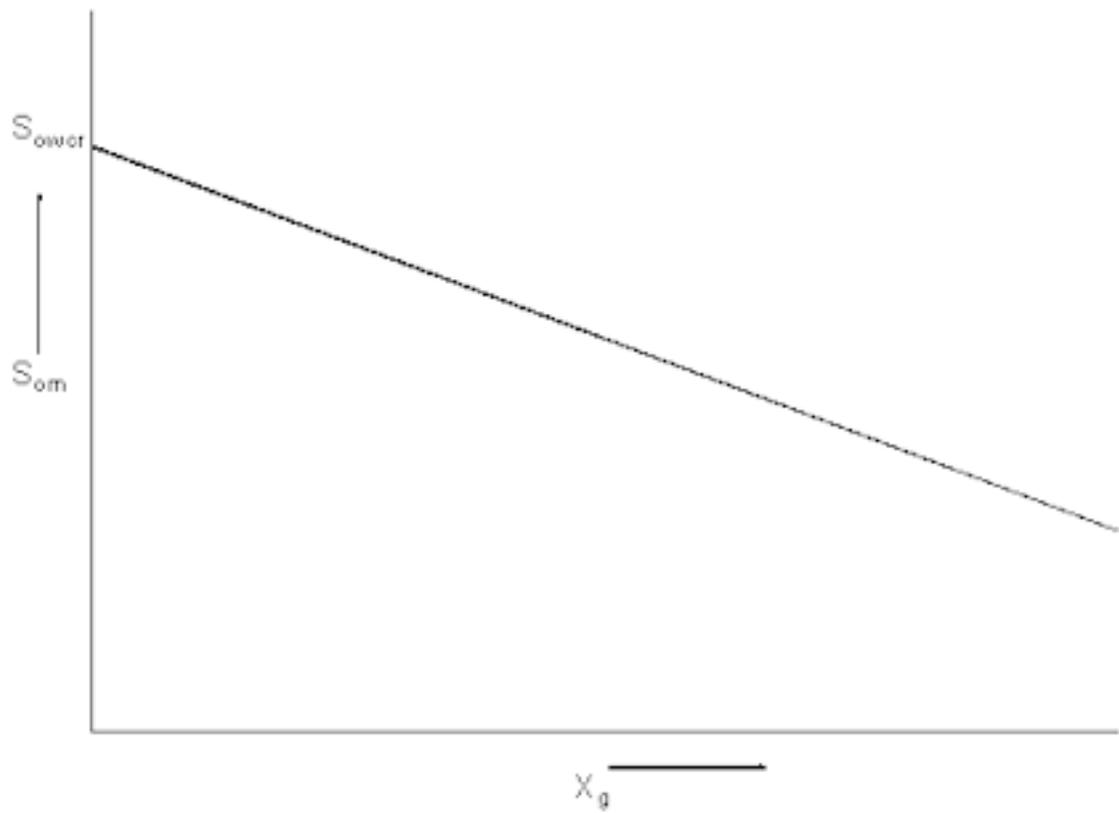
$$S_{gm} = 1 - S_{wco} - S_{ogcr} \quad \text{Eq. 3.196}$$

$$X_g = \frac{S_g}{S_{gm}} \quad \text{Eq. 3.197}$$

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:

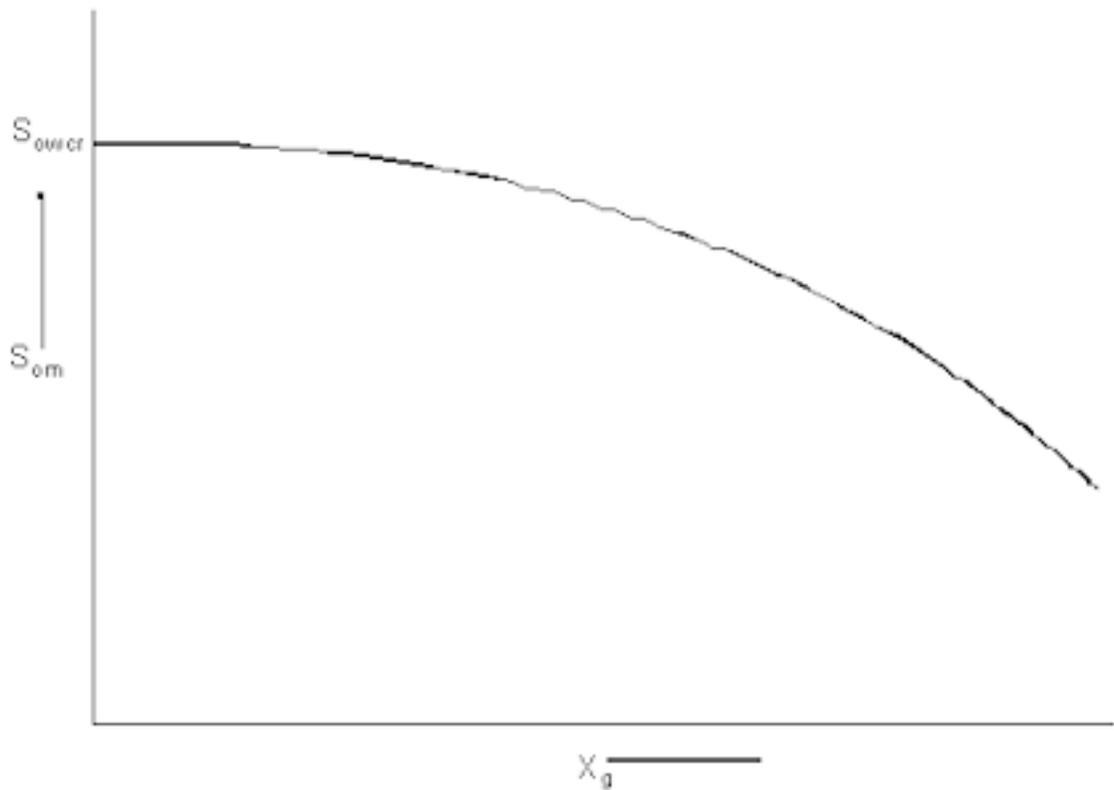


Figure 3.17. Curved interpolation for  $S_{om}$  between  $S_{ogcr}$  and  $S_{owcr}$

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 /
```

## STOPROD

## Stoichiometric coefficients for products

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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).

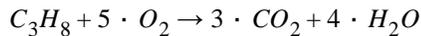
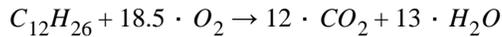
DEFAULT: 0

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



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 /

```

## STOREAC Stoichiometric coefficients for reactants

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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).

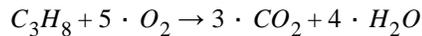
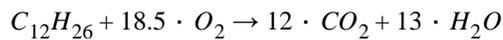
DEFAULT: 0

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  $C_{12}H_{26}$ ,  $C_3H_8$ ,  $O_2$ ,  $CO_2$  and water, the two reactions



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 /

```

## STOW

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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  /

```

## STREF

## 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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.199}$$

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:

```
STREF
527.4 527.4 /
```

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## STRESBC

## Stress boundary conditions for coupled stress and fluid flow

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

DEFAULT: 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 or 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
-- 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*      9*      0  6*  'X' /
  'I-'  3  0      9*      0  6*  'Y' /
  'I-'  3  0      9*      0  6*  'Z' /
  'I+'  4  3*      9*      0  6*  'X' /
  'I+'  3  0      9*      0  6*  'Y' /
  'I+'  3  0      9*      0  6*  'Z' /
  'J-'  4  3*      9*      0  6*  'Y' /
  'J-'  3  0      9*      0  6*  'Z' /
  'J-'  3  0      9*      0  6*  'X' /
  'J+'  4  3*      9*      0  6*  'Y' /
  'J+'  3  0      9*      0  6*  'Z' /
  'J+'  3  0      9*      0  6*  'X' /
  'K-'  4  3*      9*      0  6*  'Z' /
  'K-'  3  0      9*      0  6*  'X' /
  'K-'  3  0      9*      0  6*  'Y' /
  'K+'  4  3*      9*      0  6*  'Z' /
  'K+'  3  0      9*      0  6*  'X' /
  'K+'  3  0      9*      0  6*  'Y' /
/

```

## STRESBCL Stress boundary conditions for coupled stress and fluid flow with an LGR

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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.

DEFAULT: 4 for K+ face, and 2 for I-/I+, J-/J+ and K- faces.

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 or 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
-- set a 3000 psi overburden pressure (compressive)
-- 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* 0 6* 'Z' /
'K+' 3 'LGR1' 0 9* 'X' /
'K+' 3 'LGR1' 0 9* 'Y' /
/
```

## STVP

## Surface tension with respect to pressure

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 **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 **MISCSTR**, **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 (**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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
x	SUMMARY
x	SCHEDULE

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.

```
SUMTHIN
50.0 /
```

## SURF

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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/sm<sup>3</sup> (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 adsorped 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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## SURFACTW Enables modeling of changes of wettability in the surfactant model

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [SURFNUM](#) 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 [SURFNUM](#) 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises NTSFUN (item 1 of keyword [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 [SURFACTW](#) in the 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.

UNITS: kg/kg (METRIC), lb/lb (FIELD), gm/gm (LAB), kg/kg (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 [SATNUM](#)) and a value of 0.0 implies purely water-wet saturation functions (as defined by [SURFNUM](#)).

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 NSSFUN (item 3 of keyword [TABDIMS](#)).

---

**Note:** NTSFUN tables must be supplied; however only the SURFNUM table numbers are used. If any table after the first is defaulted, then that table is copied from the previous table.

---

### Example

With NTSFUN=4 and NSSFUN ≥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

```

## SURFADS

## Surfactant adsorption functions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (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 [ADSORP](#) keyword.

### Example

With NTSFUN=2 and NSSFUN ≥5

```

SURFADS
.0000 .00000
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0010 .00010 /

```

## SURFCAPD Surfactant capillary de-saturation functions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

$$C_{SE} = \frac{C_-}{1 - \alpha^s X^s} \quad \text{Eq. 3.200}$$

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 (/).

UNITS: dimensionless

DEFAULT: 0.0

### Example

For NTSFUN=2:

```
SURFESAL
-- alpha
  0.80 /
  0.65 /
```

## SURFNUM

## Surfactant miscible region numbers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [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 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. A non-zero capillary pressure entered in the [SWFN](#) or [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 SURFNUM keyword should be specified when the [SURFACT](#) option has been specified. If this keyword is undefined or only partially specified when the [SURFACT](#) option has been specified, the unspecified miscible saturation region numbers will default to 1.

---

**Note:** When end-point scaling has been specified via the [ENDSCALE](#) option, no end-point scaling will be applied to the saturation table used for the miscible saturation functions.

---

### 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 /
SURFNUM
 50*3 50*4 /
```

## SURFOPTS **Surfactant options**

	ECLIPSE 100
X	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $1 \text{ e}^{-6}$

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**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/rm<sup>3</sup> (METRIC), lb/rb (FIELD), gm/rcc (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 adsorped 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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 /

```

## SURFSTES Water/oil surface tension versus surfactant and salt concentrations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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
--
-- Concentration      Surface Tension
--
      0                2.00E-2
                        1.99E-2
                        1.98E-2 /
      5                2.60E-5
                        2.60E-5
                        2.60E-5 /
     10                8.00E-6
                        7.00E-6
                        6.00E-6 /
      /
      0                2.00E-2
                        1.99E-2
                        1.98E-2 /
      6                2.60E-5
                        2.60E-5
                        2.60E-5 /
     12                8.00E-6
                        7.00E-6
                        6.00E-6 /
      /

```

## SURFVISC Surfactant solution viscosity function

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

- The surfactant concentration in the solution.  
The values should increase monotonically down the column.  
UNITS: kg/sm<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB)
- 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 /
```

## SURFNUM Water-wet saturation function region and oil/water fraction table numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (SURFNUM 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 SURFNUM 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 SURFNUM 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 NTSFUN=4 specified in keyword [TABDIMS](#); NDIVIX=8, NDIVIY=6 and NDIVIZ=5 specified in RUNSPEC keyword [DIMENS](#); and no input [BOX](#) set:

```
SURFNUM
144*3 96*4 /
```

# SWAT

## Initial water saturations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 NNDIVIX), followed by the Y-axis index (from 1 to NNDIVIY), and finally the Z-axis index (from 1 to NNDIVIZ). NNDIVIX, NNDIVIY and NNDIVIZ 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 NNDIVIX=8, NNDIVIY=6, NNDIVIZ=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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$P_c = (PCW - P_{cmin}) \left( \frac{P_{ct} - P_{cmin}}{P_{cmax} - P_{cmin}} \right) + P_{cmin} \quad \text{Eq. 3.201}$$

for switch equal 5.

and

$$P_c = (P_{cmax} - PCW) \left( \frac{P_{ct} - P_{cmin}}{P_{cmax} - P_{cmin}} \right) + PCW \quad \text{Eq. 3.202}$$

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 `NDIVIX=20`, `NDIVIY=6` and `NDIVIZ=5` specified in `RUNSPEC` keyword [DIMENS](#):

```
SWATINIT
120*0
120*0.1
120*0.5
120*0.7
120*1 /
```

## SWCR

## Scaled critical water saturations

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [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, 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 ENDSCALE, 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 ENDSCALE 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, 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.

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 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

```

----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

```

## SWF3

## Three phase water saturation functions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

# SWF32D

## 2D table of water relative permeability

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $S_{oil} + S_{gas} \geq 1.0$ .
- 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.

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-----
--SGAS  0.22  0.27  0.32  0.37  0.42  0.47  0.52  0.57  0.72  0.77  0.78 /
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  0.002  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 /
    
```

```
0.35 0.010 0.001 0.001 0.001 0.000 /  
0.40 0.000 0.000 0.000 0.000 /  
/
```

## SWFN

## Water saturation functions

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, of if you are using the ECLIPSE 100 Miscible Flood or Solvent options, the saturation functions must be input with keywords from family (ii).

ECLIPSE 100

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 NTSFUN=2 and NSSFUN ≥8:

```
SWFN
.2200 .0000 7.0000
.3000 .0700 4.0000
.4000 .1500 3.0000
.5000 .2400 1*
.6000 .3300 2.0000
.8000 .6500 1.0000
.9000 .8300 1*
1.0000 1.0000 .0000 /
.18 .00 0
.32 .07 0
.50 .31 0
.60 .38 0
.80 .57 0
1.00 1.00 0 /
```

**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.

## SWGCR

## Scaled critical water saturations in gas

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 'IRREVERS' 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 'IRREVERS' 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

```
-----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
```

# SWINGFAC Swing and profile factors for FIELD gas contract

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 × 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 × 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 /
    
```

## SWL,ISWL Scaled connate water saturations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-----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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The SWLPC and ISWLPC keywords scales 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

```

----RUNSPEC section
DIMENS
 1 7 3 /
ENDSCALE
/
----PROPS section
SWLPC
7*0.22 7*0.20 7*0.18 /

```

## SWOF

## Water / oil saturation functions versus water saturation

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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      Pcow
.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        .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.

## SWU,ISWU

## Scaled saturation table maximum water saturations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 ENDSALE 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 ENDSALE, 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 ENDSALE 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

```

----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 /

```

## **T**

This section contains an alphabetic listing of keywords starting with the letter T.

## TABDIMS

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.)
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 [PVTNUM](#) 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.

DEFAULT: 20

*ECLIPSE 100*

DEFAULT: 50

*ECLIPSE 300*

4. NPPVT The maximum number of pressure nodes in any PVT table ([PVTG](#), [PVTO](#), [PVDG](#) and [PVDO](#) for example), or in any rock compaction table ([ROCKTAB](#)).

DEFAULT: 20

*ECLIPSE 100*

DEFAULT: 50

*ECLIPSE 300*

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.

*ECLIPSE 100*

6. NRPVT

The maximum number of  $R_s$  nodes in a live oil PVT table ([PVTO](#) or [PVCO](#)) or  $R_v$  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  $R_s$  nodes across all the PVT tables.

*ECLIPSE 300*

The maximum number of  $R_s$  nodes in a live oil PVT table ([PVTO](#) or [PVCO](#)).

DEFAULT: 20

*ECLIPSE 300 only*

7. The maximum number of  $R_v$  nodes in a wet gas PVT table ([PVTG](#)).

DEFAULT: 20

8. 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.

- DEFAULT: 1
- ECLIPSE 300 only* 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
- ECLIPSE 300 only* 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.
- ECLIPSE 300 only* 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
- ECLIPSE 300 only* 12. The maximum number of thermal regions.
- DEFAULT: 1
13. NTROCC The number of rock tables (see keyword [ROCK](#)).
- 
- ECLIPSE 100 only*
- 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.
- 
- ECLIPSE 300 only*
- This value also sets the number of rock tables for keywords [ROCKTAB](#), [ROCKTABH](#) and [ROCKPAMA](#).
- DEFAULT: number of pressure tables
- ECLIPSE 300 only* 14. The maximum number of pressure maintenance regions (see keyword [GPMAINT](#)).
- DEFAULT: 0
- ECLIPSE 300 only* 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.
- ECLIPSE 300 only* 16. NTALPHA The number of transport coefficient table ([ALPHA](#), [ALPHAD](#), [ALPHAI](#)) regions
- DEFAULT: NTPVT
- ECLIPSE 300 only* 17. Asphaltene: the maximum number of rows in the [ASPKDAM](#) keyword table when permeability damage is not correlated to porosity change.
- DEFAULT: 10
- ECLIPSE 300 only* 18. Asphaltene: the maximum number of rows in the [ASPREWG](#) keyword table.
- DEFAULT: 10
- ECLIPSE 300 only* 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

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $K_i$  is defined in the Cartesian  $x$ ,  $y$  and  $z$  axes.
- 2  $K_i$  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  $K_i$  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 /
```

## TBLK

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## 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/sm<sup>3</sup> ([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 /
TBLKSSGS
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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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
/
```

## TBOILS

**Boiling points for surface EoS**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

## TCBDIMS

## Temperature cutback table dimensions

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#).

## TCRIT

## Critical temperatures

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

Thermal option

## TCRITDET

**Critical temperatures for detailed components**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## TCRITS

**Critical temperatures for surface EoS**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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  
TECBDI  
X+ -3000 /  
X- -3000 /  
Y+ -3000 /  
Y- -3000 /  
/
```

## TECBDIL

**Initial tectonic boundary conditions in an LGR**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

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  
  
TECBDIL  
X+ 'LGR1' -3000 /  
X- 'LGR1' -3000 /  
Y+ 'LGR1' -3000 /  
Y- 'LGR1' -3000 /  
/
```

## TEMP

### 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.

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## TEMPI

## Cell initial temperatures

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

This keyword specifies the initial temperature values for each cell. It can be used with an explicit initialization (see "Specifying the initial state of a run in ECLIPSE 300" in the *ECLIPSE Technical Description*) in simulations where temperature variation is allowed - for example, with the Thermal option (keyword [THERMAL](#) in the RUNSPEC section) - or with a Peng-Robinson or Soave-Redlich-Kwong Equation of State.

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  $N_x N_y N_z$  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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## TEMPVD

## Temperature versus depth data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

*Thermal 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## THANALV **Use analytic oil viscosities**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\mu_o = \mu(T) \frac{\mu(P)}{\mu(P_{\text{ref}})} \quad \text{Eq. 3.203}$$

where

$\mu(T)$  Viscosity from the THANALV keyword.

$\mu(P)$  Viscosity from the PVCO keyword at the prevailing pressure.

$\mu(P_{\text{ref}})$  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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

$$THCAVE = \varphi \frac{(THCOIL + THCGAS + THCWATER + THCSOLID)}{NPHAS} + (1-\varphi)THCROCK$$

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](#), [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

```
THCGAS
1200*25.0 /
```

## THCOIL

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

$$\text{THCAVE} = \varphi \frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\varphi)\text{THCROCK}$$

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 [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 /
```

## THCONMF

**Matrix-fracture conductivity**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## THCONSF

## Saturation-dependence of conductivity

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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,  $A$ , 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:

$$A = A(S_g), \Lambda = (1 - \alpha S_g) \quad \text{Eq. 3.204}$$

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,  $A$ , as described above.

The keyword should be followed by  $N_x N_y N_z$  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 /
```

## THCROCK Thermal conductivity of rock

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

$$\text{THCAVE} = \varphi \frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\varphi)\text{THCROCK}$$

In the above expression is the  $\varphi$  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 /
```

## THCSOLID Solid phase thermal conductivity

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

$$\text{THCAVE} = \varphi \frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\varphi)\text{THCROCK}$$

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [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.

$$\text{THCAVE} = \varphi \frac{(\text{THCOIL} + \text{THCGAS} + \text{THCWATER} + \text{THCSOLID})}{\text{NPHAS}} + (1-\varphi)\text{THCROCK}$$

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 [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 /
```

## THERMAL

**Request thermal option**

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

---

## THERMEX1 Thermal expansion coefficient

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.205}$$

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 keyword [TREF](#),

and  $c_{T1_c}$  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

```

THERMEX1
-- thermal expansion coefficients for a system with 3 hydrocarbon
-- components
3.5E-06 3.5E-06 4.5E-06 /

```

## THERMNUM Thermal region numbers

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## THFLXTF

## Heat transmissibility factor for flux boundary option

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\Psi_{ni} = \alpha \times \gamma_{ni}$$

where

$\Psi_{ni}$  is the heat transmissibility between cells  $n$  and  $i$

$\gamma_{ni}$  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 /
```

## THPRES

## Threshold pressures

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 /
.....

```

```
3 1 7.0 /
2 3 /
```

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 THPRES in keyword [RPTSOL](#).

## THPRESFT Sets up a threshold pressure for a fault

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
--
'zigzag'      10.0      /
'block'       1.2      /
/

```

## THSTT97

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## THSVC

## Steam viscosity coefficients

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
 3.8217E-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 [THANALE](#)).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## THWVC

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Water viscosity coefficients

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:  $C_w = 5.1547 \times 10^{-6}$

### 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**

```
TIGHTEN
0.01 /
```

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## TIGHTENP **Tightens simulator control parameters**

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 [RPTSCHEM](#) 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 /
```

## TITLE

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```

-- RUNSPEC section:
MISCIBLE
 5 /
--PROPS section:
TLMIXPAR
0.35 /
0.57 /
0.19 /
0.0 /
1.0 0.7 /

```

## TNUM

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

ECLIPSE 100

These keywords can now be used in conjunction with [ADD](#), [ADDREG](#), [MULTIPLY](#) or [MULTIREG](#), in addition to [EQUALS](#), [EQUALREG](#) and [COPYBOX](#).

## Examples

### Example 1

ECLIPSE 100

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:

```
TNUMFSGS
48*1
48*2
48*2 /
TNUMSSGS
48*1
48*1
48*1 /
```

### Example 2

Where  $NDIVIX=2$ ,  $NDIVIY=3$  and  $NDIVIZ=3$  are specified in RUNSPEC keyword [DIMENS](#); the water phase tracer is 'WT1'; and  $NTTRVD \geq 4$  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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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   1  13   1   1   /
TOPS
6222 6232 6242 6250 6256 6260 6262 6259
6202 6212 6222 6230 6236 6241 6242 6239
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 /
    
```

# TOPT

## Preset tuning options

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 5<sup>th</sup> 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* 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:

```
TSCRIT
-- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--11--12--13-- .....
  1* 1* 1* 1.5 .125 1* 1* 1* 1* .1 1* 1* 1* /
AIMCON
-- 1-- 2-- 3-- 4-- 5-- 6-- 7-- 8-- 9--10--
  2 1* 1* 1 .5 5 .25 5 5 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 .5 100 .25 100 5 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{m}^3$  (METRIC),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

2. The corresponding volumetrics strain term (eps)

Values should be level or increasing down the column. This look up value replaces the term

$$\varepsilon_i \frac{\beta P}{1 + \beta P}$$

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](#):

```

TPAMEPS
--GC EPS1
0.000000000 0.000000000
0.000600529 0.000201575
0.001175503 0.000396899
0.005022606 0.001765517
0.008499794 0.003103030

```

0.011049732	0.004151351
0.014539122	0.005688889
0.019051263	0.007876923
0.020090423	0.008416438
0.021249485	0.009035294
0.022550474	0.009752381
0.02341045	0.010240000
0.026434767	0.012047059
0.027500578	0.012720497/

## TPAMEPSS Tables of volumetric strain values for solvent

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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),  $\text{Mscf}/\text{ft}^3$  (FIELD),  $\text{scc}/\text{cc}$  (LAB)

2. The corresponding volumetrics strain term (eps)

Values should be level or increasing down the column. The look up value will replace the solvent strain term

$$\varepsilon_s \frac{\beta_s P}{1 + \beta_s P}$$

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≥14 specified in keyword [TABDIMS](#):

```

TPAMEPSS
--SC-----EPS-----
0.0          0.0
0.002282519 0.000201575
0.004382436 0.000396899
0.016600137 0.001765517
0.02547928  0.00310303
0.031007804 0.004151351
0.037520858 0.005688889
0.044536954 0.007876923
0.045969611 0.008416438
0.047497503 0.009035294
0.049130451 0.009752381
0.05016525  0.01024
0.05354883   0.012047059
0.054654747 0.012720497
/

```

## TRACER

## Set up tracers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

Stock tank phase name: OIL, WAT or GAS.

Component name, defined by [CNAMEs](#) for a hydrocarbon component or [WNAMEs](#) for a water component.

If the tracer is associated with a water component and WNAMEs, 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), 'scf' (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](#).

ECLIPSE 100

ECLIPSE 300

ECLIPSE 100

ECLIPSE 100

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 species 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 /
SFR 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

```
A O G /
-- PRES K1(O/A) K2(G/A)
    10    1.1    5.7
    50    1.5    6.8
    90    1.7    9.1
/
-- PRES K1(O/A) K2(G/A)
    10    1.2    5.0
    50    1.7    5.9
    90    1.9    6.4
/
/ -- Default third table to second table
```

## TRACERKP Specifies the K(P) tables for the partitioned tracer option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
/
```

## TRACERS

## Dimensions and options for tracers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

DIFF 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 `DIFF` 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 `DIFF` option is not available in combination with `PARALLEL`.

NODIFF or STANDARD

No diffusion control scheme is used, that is ECLIPSE's standard fully implicit solver is used.

DEFAULT: NODIFF

---

**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 /
```

## TRACITVD

## Requests implicit flux limited transport for tracers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 **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.

---

**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.

---

## Examples

### Example 1

The TRACITVD keyword can, with strong recommendation, be used with no associated data:

```
TRACITVD  
/
```

### Example 2

The TRACITVD keyword can also be used with a chosen implicit flux limiter:

```
TRACITVD  
2 YES/
```

## TRACK

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

- 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
- Specify the required depth if DEPTH is selected in item 1.  
 UNITS: m (METRIC), ft (FIELD), cm (LAB), m (PVT-M)  
 DEFAULT: 0.0
- Request normalization of the recovered volumes. Specify N for normalization.  
 DEFAULT: unnormalized.
- Specify a list of components to be tracked, or ALL for all the components.  
 DEFAULT: ALL

### Example

```
TRACK
DEPTH 9640 N ALL /
```

## TRACKREG Tracer tracking regions

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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, NDIVI 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

-- 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 /
/

```

## TRACTBCL Traction boundary conditions in a local grid refinement

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

-- 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 /
/

```

## TRACTVD

**Requests explicit flux limited transport for tracers**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
```

## TRADS

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:  $\text{sm}^3/\text{sm}^3$  (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/ $\text{sm}^3$  (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:  $\text{sm}^3/\text{kg}$  (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 adsorped 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 .00000
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0010 .00010 /
TRADSWT2
.0000 .00000
.0003 .00005
.0005 .00007
.0008 .00008
.0010 .00008 /
.0000 .00000
.0010 .00010 /
```

## TRANGE

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/day/atm (PVT-M)

DEFAULT: 0.0

### Example

```
TRANGL
1 2 3 5 3 5 2.0 /
3 2 3 5 6 7 2.0 /
/
```

## TRANR

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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

```

-- IR1-IR2 JT1-JT2 KZ1-KZ2
BOX
  5  5  3  8  4  6  /
TRANR
18*0 /

```

## TRANHT

## Transmissibilities in the azimuthal direction

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: cP.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 TRANHT 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   /
TRANHT
12*0 /

```

## TRANX

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 /

```

## TRANY

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Y-direction transmissibility values

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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 /

```

## TRANZ

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
x	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.rm<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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 /

```

## TRCOEF

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Enable use of transport coefficients

This keyword enables use of transport coefficients. These are entered using the [ALPHA](#), [ALPHAD](#) and [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 *ECLIPSE Technical Description*).

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 /
```

## TRDCY

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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
/
...
TRDCYWT1
  10.0 /
  10.0 /
TRDCYWT2
  40.0 /
  40.0 /
TRDCYPT1      -- decay parameters apply to solution phase
  30 /
  30 /
TRDCYPT2      -- separate decay parameters for each phase
  30 10 /      -- no decay of adsorped tracer
  30 10 /
TRDCYPT3      -- separate decay parameters for each phase
  30 10 5 /    -- separate decay parameter for adsorped tracer
  30 10 5 /

```

## TRDIF

## Tracer diffusion data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [PVTNUM](#) (if NTPVT > 1).

Each table consists of one diffusion coefficient for each phase that the tracer exists in.

UNITS: m<sup>2</sup>/day (METRIC), ft<sup>2</sup>/day (FIELD), cm<sup>2</sup>/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:

```

TRACER
  WT1  WAT  /
  WT2  WAT  /
  PT1  OIL  1*  WAT  /
  PT2  OIL  1*  MULT /
/
...
TRDIFWT1
  3E-5 /
  3E-5 /
TRDIFWT2
  0.05 /
  0.05 /
TRDIFPT1  -- diffusion coefficients apply to solution phase
  4E-5 /
  4E-5 /
TRDIFPT2  -- separate diffusion coefficients for each phase
  5E-4  9E-4/
  5E-4  9E-4/

```

## TRDIS

## Tracer dispersion table numbers

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 TRDISBRI.

---

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 [PVTNUM](#) (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  $D_5$ .
6. The tracer dispersion table number for coefficient  $D_6$ .
7. The tracer dispersion table number for coefficient  $D_7$ .
8. The tracer dispersion table number for coefficient  $D_8$ .
9. The tracer dispersion table number for coefficient  $D_9$ .

---

**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:

$$\left[ \frac{\partial}{\partial x} \frac{\partial}{\partial y} \frac{\partial}{\partial z} \right] \cdot \begin{bmatrix} D_1(|v_x|, c) & D_2(|v_y|, c) & D_3(|v_z|, c) \\ D_4(|v_x|, c) & D_5(|v_y|, c) & D_6(|v_z|, c) \\ D_7(|v_x|, c) & D_8(|v_y|, c) & D_9(|v_z|, c) \end{bmatrix} \cdot \begin{bmatrix} \partial / \partial x \\ \partial / \partial y \\ \partial / \partial z \end{bmatrix} \cdot S_p c \quad \text{Eq. 3.206}$$

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/
```

## TREF

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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:

$$\rho_c = \frac{\rho_{ref_c}}{\left[1 - c_{P_c}(P - P_{ref_c})\right] \left[1 + c_{T1_c}(T - T_{ref_c})\right]} \quad \text{Eq. 3.207}$$

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

$c_{T1_c}$  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

```
TREF
527.4
527.4 /
```

## TREFS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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*.

## TREFT

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## TRKPF

## Defines tracer partitioning regions

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

```
TRKPFTRA
144*1 96*2 /
```

## TRNHD

## Enables speed-up of dispersion calculation for non-homogeneous diffusion

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

---

**Note:** This keyword cannot be used in parallel runs (that is if the [PARALLEL](#) keyword is present).

---

### Example

```
TRPLPORO
3 /
```

## TRROCK

## Specifies the tracer-rock properties

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup> (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:

$$\text{Rock mass} = V\rho_r \frac{(1-\varphi)}{\varphi} \quad \text{Eq. 3.208}$$

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 /
```

# TSCRIT

## Timestepping criteria

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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)
3. Maximum timestep.  
 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
9. Maximum worst throughput ratio.

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

13. Maximum normalized solution change.

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 /
```

## TSTEP

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 31 30 31 /
```

## TUNING

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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. TRGTTE Target time truncation error  
DEFAULT: 0.1 (Implicit), 1.0 (IMPES)
2. TRGCNV Target non-linear convergence error  
DEFAULT: 0.001 (Implicit) 0.5 (IMPES)
3. TRGMBE Target material balance error  
DEFAULT: 1.0E-7
4. TRGLCV Target linear convergence error  
DEFAULT: 0.0001 (Implicit), 0.00001 (IMPES)
5. XXXTTE Maximum time truncation error  
DEFAULT: 10.0
6. XXXCNV Maximum non-linear convergence error  
DEFAULT: 0.01 (Implicit), 0.75 (IMPES)
7. XXXMBE Maximum material balance error  
DEFAULT: 1.0E-6
8. XXXLCV Maximum linear convergence error  
DEFAULT: 0.001 (Implicit), 0.0001 (IMPES)
9. XXXWFL 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
```

```

10      0.01      1E-6      0.001      0.001      /
12  1  25  1  8  8  4*1E6      /

```

### 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      DDPLIM      DDSLIM
12  1  25  3  8  8  0.0001  0.0001  /

```

### Example 5

To run the Chappelaar 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

2. Maximum non-linear convergence error.

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<sup>2</sup>.**

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

13. Maximum normalized solution change.

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [IMPLICIT](#).

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.

- TRGLCV Target linear convergence error  
 DEFAULT: 0.00001 (if TUNINGDP is present)  
 DEFAULT: 0.0001 (if TUNINGDP is not present)
- XXXLCV Maximum linear convergence error  
 DEFAULT: 0.0001 (if TUNINGDP is present)  
 DEFAULT: 0.001 (if TUNINGDP is not present)
- 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)
- 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:

```
TUNINGDP  
/
```

## TUNINGH

## Tuning parameters for gradient calculations

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $p$ ,

$$\frac{\|R_p\|_2}{\|R_p^o\|_2} < GMSLCV \quad \text{Eq. 3.209}$$

where

$$\|R_p\|_2 = \sqrt{\frac{\sum_{i=1}^N \left(\frac{R_{i,p}}{PV_i}\right)^2}{N}} \cdot \Delta t \cdot \bar{B}_p \quad \text{Eq. 3.210}$$

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

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## TUNINGS

## Simulator control parameters for individual local grids

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
/
```

## TVDP

## Initial tracer concentration versus depth tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

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.

ECLIPSE 300

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<sup>3</sup> (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**.

ECLIPSE 100 only

---

**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:

```
TVDPFVOL
2000.0 0.0
6000.0 0.0 /
TVDPSVOL
2000.0 1.0
6000.0 1.0 /
```

### Example 2

Where VAPOIL not specified, NTTRVD=2, NSTRVD=2, oil phase tracer OR1:

```
TVDPFOR1
3000.0 1.0
4000.0 1.0 /
4000.0 0.0
5000.0 0.0 /
```

### Example 3

ECLIPSE 300

Where NTTRVD=1, NSTRVD=3, Tracer TRA

```
TVDPTRA
1000.0 0.34
4000.0 0.35
6000.0 0.39 /
```

## TZONE

## Controls the transition zone option

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [RPTSOL](#) 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 /
```

## **U**

This section contains an alphabetic listing of keywords starting with the letter U.

## UDADIMS

## Dimensions for the user defined arguments facility

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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](#), [WELDRAW](#), [WTADD](#), [WTMULT](#), [WINJEDET](#), [WECONCMF](#), [WINJTEMP](#)

and group keywords:

- [GCONPRI](#), [GCONPROD](#), [GCONINJE](#), [GECON](#), [GCONSALE](#), [GCONSUMP](#), [GRUPFUEL](#), [GRUPSALE](#), [GTADD](#), [GTMULT](#).

and segment keyword:

- [WSEGTABL](#)

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.

---

## 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 /
/
```

## UDQ

## User defined quantities

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 be also 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, WTMULT, WINJEDET, WECONCMF

and group keywords:

- GCONPRI, GCONPROD, GCONINJE, GECON, GCONSALE, GCONSUMP, GRUPFUEL, 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.

ECLIPSE 100

---

**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 <a href="#">FIP</a> sets, including the <a href="#">FIPNUM</a> 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.

Function	Precedence and Type		Description
(			Opening bracket.
)			Closing bracket.
[			Start of table argument list.
]			End of table argument list.
,			Separator for table arguments.
-	6	unary elemental	Sign change of defined element.
ABS ( )	6	unary elemental	Absolute value of defined elements.
AVEA ( )	6	unary scalar	Arithmetic average of defined elements.
AVEG ( )	6	unary scalar	Geometric average of defined elements. The result will be undefined if the input vector contains zero or negative elements.
AVEH ( )	6	unary scalar	Harmonic average of defined elements. The result will be undefined if the input vector contains zero elements.
DEF ( )	6	unary elemental	1 if the element is defined, otherwise the result is undefined.
EXP ( )	6	unary elemental	Exponential of defined elements.
IDV ( )	6	unary elemental (all elements)	1 if the element is defined, zero if the element is undefined.
LN ( )	6	unary elemental	Natural logarithm of defined elements.
LOG ( )	6	unary elemental	Base 10 logarithm of defined elements.
MAX ( )	6	unary scalar	Maximum of defined elements.
MIN ( )	6	unary scalar	Minimum of defined elements.
NORM1 ( )	6	unary scalar	1 norm of defined elements.
NORM2 ( )	6	unary scalar	2 norm of defined elements.
NORMI ( )	6	unary scalar	Infinity norm of defined elements.
NINT ( )	6	unary elemental	Nearest integer to defined elements.
PROD ( )	6	unary scalar	Product of defined elements.

Function	Precedence and Type	Description
RANDN ( )	6 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 <a href="#">UDQPARAM</a> keyword. The input argument is used purely to determine the size of the vector and the defined/undefined elements.
RANDU ( )	6 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 <a href="#">UDQPARAM</a> keyword. The input argument is used purely to determine the size of the vector and the defined/undefined elements.
RRNDN ( )	6 unary elemental	<p>“Really random” normally distributed random number with zero mean and unit variance. This is the same as the 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.</p> <p>A base simulation case seed value will be used in restart simulations unless <a href="#">UDQDIMS</a> item 11 is set to ‘Y’.</p>
RRNDU ( )	6 unary elemental	<p>“Really random” uniformly distributed random number between -1 and 1. This is the same as the 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.</p> <p>A base simulation case seed value will be used in restart simulations unless <a href="#">UDQDIMS</a> item 11 is set to ‘Y’.</p>
SORTA ( )	6 unary elemental	Position of the element in an ascending sort of defined elements.
SORTD ( )	6 unary elemental	Position of the element in a descending sort of defined elements.
SUM ( )	6 unary scalar	Sum of defined elements.
UNDEF ( )	6 unary elemental	1 if the element is undefined, otherwise the result is undefined.
TU* [ ]	6 unary elemental	<p>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 <a href="#">UDT</a> keyword, and the interpolation method used for each argument is specified in the table.</p> <p>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.</p> <p>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 <a href="#">UDQ</a> s if a more complicated expression is required.</p>
<=	5 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 <a href="#">UDQPARAM</a> keyword.

Function	Precedence and Type		Description
>=	5	binary intersection	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.
<	5	binary intersection	1 if the left argument is less than the right argument, else 0.
>	5	binary intersection	1 if the left argument is greater than the right argument, else 0.
==	5	binary intersection	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.
!=	5	binary intersection	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.
^	4	binary intersection	Exponentiation.
*	3	binary intersection	Multiplication.
/	3	binary intersection	Division.
+	2	binary intersection	Addition.
-	2	binary intersection	Subtraction.
UADD	1	binary union	Union of the two vectors with intersecting elements added together.
UMAX	1	binary union	Union of the two vectors with the maximum of intersecting elements taken.
UMIN	1	binary union	Union of the two vectors with the minimum of intersecting elements taken.
UMUL	1	binary union	Union of the two vectors with intersecting elements multiplied together.

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:

```
ASSIGN WUMW1 'P*1*' 1.0 /
ASSIGN WUMW1 P12 2.0 /
```

would result in the user defined well vector:

	P11	P12	P32	P33
WUMW1	1.0	2.0	UNDEFINED	UNDEFINED

---

**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:

```
DEFINE WUMW1 WOPR 'P*1*' + WOPR 'P*2*' /
```

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:

	P11	P12	P32	P33
WOPR 'P*1*'	WOPR P11	WOPR P12	UNDEFINED	UNDEFINED
+				
WOPR 'P*2*'	UNDEFINED	WOPR P12	WOPR P32	UNDEFINED
=				

WUMW1      UNDEFINED 2 x WOPR P12 UNDEFINED UNDEFINED

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:

```
ASSIGN WUMW1 P1 1 /
DEFINE WUMW1 WUMW1 * WOPR /
```

for which the calculation would be:

	P11	P12	P32	P33
WUMW1	1	UNDEFINED	UNDEFINED	UNDEFINED
*				
WOPR	WOPR P11	WOPR P12	WOPR P32	WOPR P33
=				
WUMW1	WOPR P11	UNDEFINED	UNDEFINED	UNDEFINED

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:

```
DEFINE FUMW1 WOPR P11 + WOPR P12 /
```

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:

```
DEFINE RUMW1 ROIP_NUM UMAX RGIP_ONE UMIN RWIP_TWO /
```

which would generate the vectors:

	NUM set			ONE set			TWO set		
ROIP_NUM	X	X	X	U	U	U	U	U	U
UMAX									
RGIP_ONE	U	U	U	X	X	X	U	U	U
UMIN									
RWIP_TWO	U	U	U	U	U	U	X	X	X
=									
RUMW1	ROIP_NUM			RGIP_ONE			RWIP_TWO		

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

```
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

```
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 /
```

```

-- A COUPLE OF GROUP QUANTITIES
DEFINE GUMW1 SORTA(GOPR) /
UNITS GUMW1 GRPS /
ASSIGN GUMW2 G2 99.99 /
UNITS GUMW2 GRPS /

-- A COUPLE OF REGION QUANTITIES
DEFINE RUMW1 ROIP /
UNITS RUMW1 REGS /
ASSIGN RUMW2 2.0E-2 /
ASSIGN RUMW2ONE 3 3.0 /
ASSIGN RUMW2TWO 4 4.0 /
UNITS RUMW2 REGS /

-- A COUPLE OF WELL QUANTITIES
DEFINE WUMW1 WBHP 'P*1*' UMAX WBHP 'P*4*' /
UNITS WUMW1 WELLS /
UPDATE WUMW1 NEXT /
DEFINE WUMW2 RANDN(WBHP 'P1*') /
UNITS WUMW2 WELLS /
-- A COUPLE OF AQUIFER QUANTITIES
DEFINE AUMW1 SORTA(AAQR) /
UNITS AUMW1 AQUIFER /
UPDATE AUMW1 NEXT /
DEFINE AUMW2 RANDN(AAQR) /
UNITS AUMW2 AQUIFER/

-- A COUPLE OF BLOCK QUANTITIES
DEFINE BUMW1 MAX(BOSAT) /
UNITS BUMW1 BLOCK /
UPDATE BUMW1 NEXT /
DEFINE BUMW2 RANDN(BOSAT) /
UNITS BUMW2 BLOCK/
-- A WELL QUANTITY CALCULATED FROM A 3-DIMENSIONAL UDT
DEFINE WUPRICE TUPRICE[FGPR,FOPR,FWPR]-TUPRICE[WGPR,WOPR,WWPR] /
/
/

```

## 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_CAPXT 0.0 /
DEFINE FU_CAPXT FU_CAPXT + FU_CAPEX /
DEFINE FU_CAPXQ FU_CAPXQ + FU_CAPEX /
--
/

```

**Note:** Both FU\_CAPXT and FU\_CAPXQ are defined in (almost) exactly the same way but only FU\_CAPXT will give the desired behavior.

The only difference is that FU\_CAPXQ is initialized at the beginning of the UDQ definitions, whereas FU\_CAPXT 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\_CAPXT 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

```
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.

```
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.

```
UDQ
ASSIGN WUBHP1 0.0 /
DEFINE WUBHP2 WBHP P /
DEFINE WUBHPDT ((WUBHP2- WUBHP1) / Timestep) /
DEFINE WUBHP1 WUBHP2 /
UNITS WUBHP1 PSIA /
UNITS WUBHP2 PSIA /
/
```

## UDQDIMS

## Dimensions for the user defined quantity facility

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 RRNDU. 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 2 Y /
```

## UDQPARAM Parameters for the user defined quantity facility

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- Integer seed value for the RANDN and RANDU random number functions. This item should be a integer greater than 0.  
DEFAULT: 1
- 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
- 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
- The fractional equality tolerance used in the ==, !=, <= and >= functions.  
DEFAULT: 1.0E-4

### Example

```
UDQPARAM
100 1* -1.0 /
```

## UDT

## User defined tables

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

```

RUNSPEC
UTDDIMS
(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
-- the next 1..n2 rows contain n1 values each, and represent a table
  in dimensions IA1 and IA2, for IA3 = 2 and IA4 = 1.

-- 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

```

```

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.
-- Next 1 to j rows corresponding to the results table for in the
dimensions of IA1 and IA2, for IA3 value k
-- {IA1 value 1} {IA1 value 2} ... {IA1 value i}
{Result11k} {Result21k} ...{Resultilk} / -- for IA2 Value 1
.../
{Result1jk} {Result2jk} ...{Resultijk} / -- for IA2 Value j
/ -- Third dimension (IA3) value k table termination
--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

UTDDIMS
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.

```

RUNSPEC

UDTDIMS
1 20 4 3/
...
SCHEDULE

UDT
'TUTESTB' 3 /
'LL' 1 2 3 /
'NV' 2 4 /
'ID' 'W2' '*' /
-- the next two rows are both for the first 'ID' dimension'
1 2 3 / -- values corresponding to the first 'NV' dimension
2 4 6 / -- values corresponding to the second 'NV' dimension
/ -- this slash terminates the 'NV' dimensions
-- the next two rows are both for the second 'ID' dimension'
10 11 12 / -- values corresponding to the first 'NV' dimension
20 22 24 / -- values corresponding to the second 'NV' dimension
/ -- this slash terminates the 'NV' dimensions
/ -- this slash terminates the 'ID' dimensions
/ -- this slash terminates the UDT keyword

```

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.

WOPR would be evaluated for W1, and then linear interpolation/extrapolation used.

WGPR would be evaluated for W1, and then the "nearest value" method of interpolation used.

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 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 WOPR lies with relation to the interpolation points for the first dimension, which are 1 2 3.

So, if WGPR for W1 is 2, and 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 WGPR and WOPR, then this time the result will be 1.5.

We now have well vector WUTESTB as our result, with values 10.5 and 1.5. Units can be assigned to it within the UDQ keyword as required.

## UDTDIMS

## Dimensions for the user defined tables facility

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

ECLIPSE 300

This keyword indicates that gradient, or sensitivity, information produced by the simulator for history matching purposes is to be unencoded.

---

**Note:** The “ unencodedhmd ” 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.

## UNIFIN

### 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.

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## UNIFOUT

### 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.

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## UNIFOUTS

**Indicates that summary output files are unified**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## USECUPL

## Tells ECLIPSE to read a reservoir coupling file

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

## USEFLUX

## Use a flux file

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

ECLIPSE 100

ECLIPSE 300

ECLIPSE 300

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.

ECLIPSE 300

**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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

## VAPOIL

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 is present.

## VAPPARS Oil vaporization controls

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$R_v = R_{v \text{ sat}} \left( \frac{S_o}{S_{o \text{ max}}} \right)^{\text{vap1}} \quad \text{Eq. 3.211}$$

$$R_s = R_{s \text{ sat}} \left( \frac{S_o}{S_{o \text{ max}}} \right)^{\text{vap2}} \quad \text{Eq. 3.212}$$

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/kg-M (METRIC), ft<sup>3</sup>/lb-M (FIELD), cc/gm-M (LAB), m<sup>3</sup>/kg-M (PVT-M)

### Example

In a run with 2 components and one EoS region:

```
VCOMPACT
1.473 5.529 /
```

## VCRIT

## Critical volumes

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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: m<sup>3</sup>/kg-M (METRIC), ft<sup>3</sup>/lb-M (FIELD), cc/gm-M (LAB), m<sup>3</sup>/kg-M (PVT-M)

## Example

```
VCRIT
1.473 5.529 /
```

## VCRITDET Critical volumes for detailed components

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/kg-M (METRIC), ft<sup>3</sup>/lb-M (FIELD), cc/gm-M (LAB), m<sup>3</sup>/kg-M (PVT-M)

### Example

```
VCRITDET
1.473 5.529 /
```

## VCRITS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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*.

## VCRITVIS

**Critical volumes for viscosity calculations**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/kg-M (METRIC), ft<sup>3</sup>/lb-M (FIELD), cc/gm-M (LAB), m<sup>3</sup>/kg-M (PVT-M)

## VDFLOW

## Specify velocity-dependent flow coefficient

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\text{atma.s}^2.\text{g}^{-1}$  (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

```
VDFLOW
10.0 /
```

## VDFLOWR

## Specify velocity-dependent flow coefficient

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $\text{atma.s}^2.\text{g}^{-1}$  (the Forchheimer) in all unit sets. Dake [[Ref. 30](#)] in chapter 8 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
```

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 /
```

## VDKRG

## Gas velocity dependent relative permeability data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sub>1</sub> Forchheimer model.

6. The gas Forchheimer  $b_g$  parameter.

This value must be specified for the B<sub>1</sub> 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<sub>2</sub> 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  $9.86 \text{ Forch} = 10^7 \text{ cm}^{-1}$ ), and the permeability unit (mD):

- item 5 has units of Forch(mD)<sup>d</sup>
- item 9 has units of Forch/(mD)<sup>d</sup>

where  $d$  is the  $d_g$  specified in item 8.

See also the [VDKRO](#) 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*  /  
.....
```

## VDKRGC

## Gas condensate velocity dependent relative permeability data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  /
```

## VDKRO

## Oil velocity dependent relative permeability data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

The data comprises of 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<sub>1</sub> Forchheimer model.

6. The oil Forchheimer  $b_o$  parameter.

This value must be specified for the B<sub>1</sub> 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<sub>2</sub> 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  $9.86 \text{ Forch} = 10^7 \text{ cm}^{-1}$ ), and the permeability unit (mD):

- item 5 has units of Forch(mD)<sup>d</sup>

- item 9 has units of Forch/(mD)<sup>d</sup>

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.

```
VDKRO
-- 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*  /
```

## VE

## 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 /
```

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## VECTABLE **Size of vector property table**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
VECTABLE
2000 /
```

## VEDEBUG

## Controls debug for the VE and compressed VE options

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 7<sup>th</sup> 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:

$$1 \leq IX1 \leq IX2 \leq NDIVIX$$

$$1 \leq JY1 \leq JY2 \leq NDIVIY$$

$$1 \leq KZ1 \leq KZ2 \leq NDIVIZ \text{ (where NDIVIX, NDIVIY and NDIVIZ are specified in RUNSPEC keyword [DIMENS](#))}$$

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

## Controls vertical equilibrium model

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

VEFIN may be used in LGR runs with vertical equilibrium in the global grid (keyword **VE** in the RUNSPEC section), to control the vertical equilibrium model in the local grid systems. VEFIN should be placed after the keyword introducing the local grid (**CARFIN** or **RADFIN**) and before the local grid data is terminated with **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 **CARFIN** or **RADFIN** keyword:

1. Vertical Equilibrium flag:

YES If vertical equilibrium is required in the local grid.

NO To use dispersed flow in the local grid.

DEFAULT: NO

2. 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 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 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. (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 NVEPT is set less than the number of distinct local grid depths within a global cell, then only NVEPT equally spaced distinct depths will be used. Note that if NVEPT is quite small (10 to 20), this is still a better approximation of the local system than the original (NVEPT = 0) description of the global cell.

In general it is unlikely to be worth setting NVEPT to a value greater than 50.

DEFAULT: 0 (which applies the original description of the global cell.)

---

**Note:** The equilibration option selected by the item 9 in the **EQUIL** keyword is **not** affected by the VEFIN setting. If VE equilibration is requested (item 9 of **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 **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.

```
VEFIN  
YES /
```

### Example 2

Increase the number of distinct depths used to represent the global host cell (with no VE in the local grid)

```
VEFIN  
--      NVEPT  
NO      40  /
```

## VEFRAC

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 VEFAC 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 VEFACV keyword. The corresponding keywords VEFACP and VEFACPV 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 [SWFN](#).

By setting VEFAC to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction (1.0 - VEFAC) of the rock curve and VEFAC of the VE curve is taken.

See also keywords [VEFACV](#), [VEFACP](#), [VEFACPV](#).

### Example

```
VEFRAC
0.9 /
```

## VEFRACP

## Fraction of VE pseudo capillary pressure to be used

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 `VEFRACPV` parameter on a cell-by-cell basis.

DEFAULT: 1.0 (Throughout the entire grid.)

When `VEFRACPV` 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 `VEFRACPV` to a value less than 1.0, it is possible to mix the VE and rock curves; a fraction (1.0 - `VEFRACPV`) of the rock curve and `VEFRACPV` 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   1   2   /
VEFRACPV
8*0.5
8*1.0 /

```

## VEFRACV

## Fraction of VE relative permeability curves used

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /

```

## VELDEP

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## Velocity dependent relative permeability settings

The data consists of the following items, describing the options available within the VDRP Model.

- 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)
- 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)
- 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)
- 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)
- 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 /
```

## VFPCHECK VFP table consistency check data

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword is generated by VFP *i* when it is working in its compositional mode. It contains data that was used by VFP *i* 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 *i*. 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 (/).

- Water density at standard conditions  
UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)
- Molecular weight of water  
UNITS: kg/kg-M (METRIC), lb/lb-M (FIELD), gm/gm-M (LAB), kg/kg-M (PVT-M)
- Standard pressure  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
- Standard temperature  
UNITS: °C (METRIC), °F (FIELD), °C (LAB), °C (PVT-M)
- Units system
- Number of components

### Example

```
VFPCHECK
1022.0 18.02 1.0132 15.5556 METRIC 15 /
```

## VFPCHK

## Sets threshold BHP for VFP table checks

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

ECLIPSE 300

## VFPIDIMS

## Injection well VFP table dimensions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

---

**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).

---

### 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

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 VFPIIDIMS.

Volume rate: UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  for oil or liquid (FIELD),  $\text{Mscf}/\text{day}$  for gas (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

Molar rate: UNITS:  $\text{kg-M}/\text{day}$  (METRIC),  $\text{lb-M}/\text{day}$  (FIELD),  $\text{gm-M}/\text{hr}$  (LAB),  $\text{kg-M}/\text{day}$  (PVT-M)

ECLIPSE 300

## 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 VFPIIDIMS. 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<sup>st</sup> FLO value and NT<sup>th</sup> THP value.
3. BHP at 2<sup>nd</sup> FLO value and NT<sup>th</sup> THP value.
- .... ..

NFLO +1 BHP at last FLO value and NT<sup>th</sup> THP value.

All combinations of  $\text{NF} = 1$  to NFLO and  $\text{NT} = 1$  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:

```
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:

```
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
   /
```

# VFPPDIMS

## Production well VFP table dimensions

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

---

### ECLIPSE 100

To help select the most appropriate values, if the mnemonic 'VFPPROD=2', or higher, is entered in the RPTSCHEM 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 5<sup>th</sup> 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`, `WVFPDP`, `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  $f_w^{\text{vol}} / f_h^{\text{vol}}$ , 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  $f_w^m / (f_h^m + f_w^m)$ , where  $f_w^m$  is the molar flow rate of water at surface conditions, and  $f_h^m$  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 <a href="#">WALQCALC</a> )
ECLIPSE 100 only	DENG	Surface density of gas (See keyword <a href="#">WALQCALC</a> )
	BEAN	Choke diameter (multisegment wells)

' ' Undefined.

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 [WALQCALC](#) 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 MXMFLO, which is item 1 in VFPPDIMS.

Volume rate: UNITS:  $\text{sm}^3/\text{day}$  (METRIC), stb/day for oil or liquid (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

ECLIPSE 300

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 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), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

For WGR or WWR: UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

ECLIPSE 300

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 the 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), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

For OGR: UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

ECLIPSE 300

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

When ALQ = IGLR or TGLR: UNITS: sm<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

ECLIPSE 100 only

When ALQ = DENO or DENG: UNITS: kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

When ALQ = BEAN:                    UNITS: mm (METRIC), 1/64<sup>th</sup> 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<sup>th</sup> THP value
    - NW<sup>th</sup> WFR value
    - NG<sup>th</sup> GFR value
    - NA<sup>th</sup> ALQ value
  6. BHP or THT at
    - 2<sup>nd</sup> 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
- .....
- 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 *i* 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 *i* to use another pressure value to indicate supersonic flow (see the “VFPi User Guide”, and keyword SONICPRS in the VFP *i* 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 [VFPCHK](#) 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 [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, NFLO=5, NTHP=2, NWFR=1, NGFR=3, NALQ=1:

```

VFPPROD
 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+03 / 5 flow vals
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 1.93199E+03 1.36585E+03 6.77031E+02 7.15261E+02
8.62436E+02 /
2 1 1 1 2.73663E+03 2.73303E+03 2.75085E+03 2.77323E+03
2.90209E+03 /
1 1 2 1 1.77471E+03 4.33035E+02 5.38422E+02 6.30479E+02
9.39472E+02 /
2 1 2 1 2.51228E+03 2.38072E+03 2.35995E+03 2.26536E+03
2.28849E+03 /
1 1 3 1 1.64735E+03 4.41989E+02 6.95286E+02 8.81634E+02
1.41797E+03 /
2 1 3 1 2.46600E+03 1.78161E+03 1.80525E+03 1.85156E+03
2.04484E+03 /

```

## VFPTABL

## Set method of VFP table interpolation for ALQ

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## VISCAQA

## Specify coefficients of Ezrokhi's method for aqua viscosity

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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
/
```

## VISCD

### Activates the viscous displacement option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [LX](#), [LY](#) and [LZ](#) keywords in the `GRID` section. `VISCD` is not available with the [PARALLEL](#) keyword.

See "[Viscous displacement \(ECLIPSE 100\)](#)" in the *ECLIPSE Technical Description* for further information.

The keyword has no associated data.

## VISCREF

## Reference conditions for the viscosity / temperature tables

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (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:

```
VISCREF
3000 1.24 /
3000 1.24 /
```

## VISDATES

## VISAGE geomechanics stress step dates

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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:

- The day of the month  
An integer between 1 and 31.
- The name of the month abbreviated to three characters  
JAN, FEB, MAR, APR, MAY, JUN, JULY, AUG, SEP, OCT, NOV or DEC.  
JUL is an acceptable alternative to JULY.
- The year  
A positive 4 digit integer.
- 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 /
/

```

## VISOPTS

## VISAGE geomechanics workflow options

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- 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

- 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

- 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

- 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

- Not used in this release

- Whether the existing restart frequency is to be maintained.

YES If there are [RPTRST](#) 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\mu_w = \mu_{ref_w} / (1 + Y + Y^2/2)$$

$$Y = -c_{V_w} \cdot (P - P_{ref_w})$$

where

$\mu_{ref_w}$  is the reference viscosity, specified with this keyword

$c_{V_w}$  is the viscosibility, specified with this keyword

$P_{ref_w}$  is the reference pressure, specified with keyword [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:

UNITS: cP (METRIC), cP (FIELD), cP (LAB), cP (PVT-M)

DEFAULT: 0.3 cP.

2. The second  $N_w$  values represent the viscosibility at the reference pressure for each water 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 in 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

```
VREFW
0.31 0.32
0.0 0.0 /
```

## W

This section contains an alphabetic listing of keywords starting with the letter W.

## WADVANCE Assign advance import gas to a well

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$Sg_{trap} = Sg_{cr} + \frac{(Sg_m - Sg_{cr})}{(1 + C \cdot (Sg_m - Sg_{cr}))}$$

where

$Sg_{trap}$  is the trapped gas saturation

$Sg_m$  is the maximum gas saturation attained

$Sg_{cr}$  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  $Sg_m$  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

## 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  $S_{g_m}$  and  $S_{g_{trap}}$  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:

```
WAGHYSTR
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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- 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 (/).

## ECLIPSE 300

**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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WLIST](#).

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. 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 [WLIST](#).

DENO The well's ALQ is equated to its oil surface density, in kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

DENG The well's ALQ is equated to its gas surface density, in kg/m<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (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 [GLIFTLIM](#)).

---

## Example

```
WALQCALC
' * '      DENO      / equates ALQ to oil surface density in all wells
PROD3     NONE      / except for PROD3 which has no special ALQ definition
/
```

## WAPI

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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.

x	<b>ECLIPSE 100</b>
	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
x	GRID
x	EDIT
x	PROPS
x	REGIONS
x	SOLUTION
x	SUMMARY
x	SCHEDULE

## WARP

### 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.

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## WATDENT

## Water density as a function of temperature

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $3.0e-4 \text{ K}^{-1} = 1.67 \text{ °R}^{-1}$
3. The second thermal expansion coefficient  $c_{T2}$   
 UNITS: 1/K<sup>2</sup> (METRIC), 1/°R<sup>2</sup> (FIELD), 1/K<sup>2</sup> (LAB), 1/K<sup>2</sup> (PVT-M)  
 DEFAULT:  $3.0e-6 \text{ K}^{-2} = 9.26e-7 \text{ °R}^{-2}$

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

$$X = C \cdot (P - P_{ref}) \text{ and}$$

$P_{ref}$ ,  $B_w(P_{ref})$  and  $C$  are defined by the [PVTW](#) keyword.

## Example

```
WATDENT
/
```

## WATER

### Run contains water

This indicates that the run contains water as an active phase, whose saturation can vary.

The keyword has no associated data.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## WATERTAB **Water pressure tables**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{mm}^3/\text{sm}^3$  (METRIC), rb/stb (FIELD), rcc/scc (LAB),  $\text{mm}^3/\text{sm}^3$  (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

```

WATERTAB
3000 1.00341 0.52341
3900 1      0.56341 /

```

## WATVISCT Water viscosity as a function of temperature

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:

$$\mu_w(T, P) = \mu_T(T) \frac{\mu_p(P)}{\mu_p(P_{ref})} \quad \text{Eq. 3.213}$$

or, if the brine option is active, as

$$\mu_w(T, P, c_s) = \mu_T(T) \frac{\mu_p(P, c_s)}{\mu_p(P_{ref}, c_{s,ref})} \quad \text{Eq. 3.214}$$

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  $NPPVT \geq 3$ :

```
WATVISCT
  10  1
 100 0.6
 150 0.4  /
```

## WAVAILIM

## Requests availability limiting of injection

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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*.

## WBHGLR

## Well limits on bottom hole gas liquid ratio

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELSPEC](#)). 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 (/).

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. Upper limit bottom hole GLR for well rate cutback

A value of 0.0 or > 1.0E19 switches off this check.

UNITS:  $\text{rm}^3/\text{rm}^3$  (METRIC),  $\text{rb}/\text{rb}$  (FIELD),  $\text{rcc}/\text{rcc}$  (LAB),  $\text{rm}^3/\text{rm}^3$  (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:  $\text{rm}^3/\text{rm}^3$  (METRIC),  $\text{rb}/\text{rb}$  (FIELD),  $\text{rcc}/\text{rcc}$  (LAB),  $\text{rm}^3/\text{rm}^3$  (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.

UNITS: Fraction

DEFAULT: same as item 6 in the WCUTBACK keyword, if that has been entered previously, otherwise 1.0

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:  $\text{rm}^3/\text{rm}^3$  (METRIC),  $\text{rb}/\text{rb}$  (FIELD),  $\text{rcc}/\text{rcc}$  (LAB),  $\text{rm}^3/\text{rm}^3$  (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.

NO 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*
/
```

## WBOREVOL Sets volume for wellbore storage

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>). 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<sup>3</sup> (METRIC), ft<sup>3</sup> (FIELD), cc (LAB), m<sup>3</sup> (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
```

PRODUCER	17600	/
/		

## 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:

$$C_D = \frac{C}{2\pi\phi c_t h r_w^2} \quad \text{Eq. 3.215}$$

where

$C$  is the wellbore storage coefficient

$\pi$  is 3.14159

$\phi$  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,

$$V = \frac{dV}{dP} \quad \text{Eq. 3.216}$$

For storage due to fluid expansion in the wellbore,

$$C = c_w V_w \quad \text{Eq. 3.217}$$

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:

$$V_w = \frac{2\pi\phi c_t h r_w^2 C_D}{c_w} \quad \text{Eq. 3.218}$$

## WCALVAL Sets the calorific value of gas at a well

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), Btu/Mscf (FIELD), J/scc (LAB), kJ/sm<sup>3</sup> (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 /
/
```

## WCOL

## Coloring for WARP linear solver

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

*ECLIPSE 300 only*

**WGRA** The well produces the observed wet gas rate. The other phases will be produced according to their mobility ratios.

*ECLIPSE 300 only*

**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

ORAT Controlled by the observed oil rate (item 4)

WRAT Controlled by the observed water rate (item 5)

GRAT Controlled by the observed gas rate (item 6)

LRAT Controlled by the observed liquid rate (oil + water)

CRAT Controlled by the observed linearly combined rate (see keyword [LINCOM](#))

RESV Controlled by the reservoir fluid volume rate calculated from the observed phase flow rates

BHP Controlled by the well bottom hole pressure (item 10)

*ECLIPSE 300 only*

WGRA Controlled by the observed wet gas rate (item 11)

*ECLIPSE 300 only*

NGL Controlled by the observed NGL rate (item 12)

The control mode may be overridden by earlier use of the [WHISTCTL](#) keyword.

### 4. Observed oil production rate

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M )

DEFAULT: 0.0

### 5. Observed water production rate

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: 0.0

### 6. Observed gas production rate

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: 0.0

### 7. Production well VFP table number

(see keyword [VFPPROD](#))

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 [VFPPROD](#))

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 WOPTH).

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 WOPTH 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  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*.

---

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## WCONINJE Control data for injection wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

*ECLIPSE 300* 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 300* 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](#)"

*ECLIPSE 100* OIL Oil injector

*ECLIPSE 300* An alternative oil injector type, HCOIL must be used.

MULTI Multiphase injector

*ECLIPSE 100* Water and/or gas and/or oil injection.

*ECLIPSE 300* 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.

*ECLIPSE 300* HCGAS 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](#)"

*ECLIPSE 300* HCOIL Oil injector

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*            STEAM-    Steam and gas injector  
                         GAS

*Thermal*            STEAM-    Steam and oil injector  
                         OIL

*Thermal*            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](#).

For a multiphase injector, this is the surface flow rate of the preferred phase of the well.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{rm}^3/\text{day}$  (METRIC), rb/day (FIELD), rcc/hr (LAB),  $\text{rm}^3/\text{day}$  (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), stb/Mscf for gas injectors (FIELD), Mscf/stb for oil injectors (FIELD), scc/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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

DEFAULT: 0.0

*ECLIPSE 100 only*

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

13. 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

14. 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

*Thermal*

15. Ratio of oil volume to steam volume (C.W.E.) for a STEAM-OIL injector (for use with steam-solvent injection).

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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 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.
- 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.

*Thermal*

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.

*ECLIPSE 100 only*

## 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.

*ECLIPSE 100 only*

```
WCONINJE
MI1 MULTI OPEN RATE 16000 1* 500 4* 0.05 0.15 0.80 /
/
```

In this example, a gas rate target of 16,000 sm<sup>3</sup>/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<sup>3</sup>/day of gas, (16,000\*0.05/0.8 =) 1,000 sm<sup>3</sup>/day of oil and (16,000\*0.15/0.8 =) 3,000 sm<sup>3</sup>/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.

```
WCONINJE
MI2 MULTI OPEN RATE 2000 1* 5000 5* 0.8 0.2 /
/
```

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WEL SPECS](#)) 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 [WEL SPECS](#)). 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:  $\text{sm}^3/\text{day}$  (METRIC), stb/day for oil or water (FIELD), Mscf/day for gas (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 WBHPH. 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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf for gas injectors (FIELD), Mscf/stb for oil injectors (FIELD), scc/scc (LAB)

DEFAULT: 0.0

*ECLIPSE 100 only* 9. 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

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 /
/

```

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

*ECLIPSE 300*

When running in compositional mode, the nature of the injected gas must also be specified with keyword [WINJGAS](#).

*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

4. BHP upper limit

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:  $\text{sm}^3/\text{sm}^3$  (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*    9. 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

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 `WEL SPECS`) 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*

- 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.

## 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.

```
WCONINJP
INJW WAT 1* 5000 /
PRD1 /
PRD2 /
PRD3 /
PRD4 /
/
```

## 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 (WCONINJP record 1, items 9-11).

## WCONPROD Control data for production wells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

*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 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 <b>CALVAL</b> 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 <b>GCONPROD</b> .
	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 <b>UDQ</b> and <b>UDADIMS</b> .  UNITS: sm <sup>3</sup> /day (METRIC), stb/day (FIELD), scc/hr (LAB), sm <sup>3</sup> /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 <b>UDQ</b> and <b>UDADIMS</b> .  UNITS: sm <sup>3</sup> /day (METRIC), stb/day (FIELD), scc/hr (LAB), sm <sup>3</sup> /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 <b>UDQ</b> and <b>UDADIMS</b> .
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 <sup>3</sup> /day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm <sup>3</sup> /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 <b>UDQ</b> and <b>UDADIMS</b> .  UNITS: sm <sup>3</sup> /day (METRIC), stb/day (FIELD), scc/hr (LAB), sm <sup>3</sup> /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:  $\text{rm}^3/\text{day}$  (METRIC),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{rm}^3/\text{day}$  (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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{Mscf}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{kg-M}/\text{day}$  (METRIC),  $\text{lb-M}/\text{day}$  (FIELD),  $\text{gm-M}/\text{hr}$  (LAB),  $\text{kg-M}/\text{day}$  (PVT-M)

- DEFAULT: No target or limit
- ECLIPSE 300 only* 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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
- DEFAULT: No target or limit
- ECLIPSE 300 only* 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  $BHP \geq P_{sat} + \Delta P$  where  $P_{sat}$  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)
- ECLIPSE 300 only* 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  $BHP \geq P_{sat}(T + \Delta T)$  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)
- ECLIPSE 300 only* 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
- ECLIPSE 300 only* 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<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 /
.....

```

```
PROD4 OPEN ORAT 1000 2000 3*          2000 /  
/
```

## WCUTBACK Well cutback limits (for producers and injectors)

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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

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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/Mscf (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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

- 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.
- 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.
- 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 [WLIMTOL](#).

## Example

```
WCUTBACK  
'P*' 0.6 3* 0.9 LIQ 3000 /  
WELL2 1* 3.0 2* 0.9 RESV 3* 2.0 /  
/
```

## WCUTBACK Tracer cutback limits for production wells

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

WCUTBACK 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*/
/
```

## WCYCLE

## Automatic cycling of wells on and off

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$(T - T_o) / \text{Start-up Time} \quad \text{Eq. 3.219}$$

where

T is the time at the end of the timestep

T<sub>o</sub> 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:

```
WELOPEN
      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 /
WELOPEN
      INJG   OPEN  /
/
```

## WDFAC

## Sets well D-factor (flow-dependent skin for gas)

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/sm<sup>3</sup> (METRIC), day/Mscf (FIELD), hr/scc (LAB), day/sm<sup>3</sup> (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:

$$S \rightarrow S + D_w |Q_{fg}| \quad \text{Eq. 3.220}$$

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:

$$S \rightarrow S + D(j) |q_{fg}(j)| \quad \text{Eq. 3.221}$$

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:

$$D(j) = D_w \frac{\sum T_w(i)}{T_w(j)} \quad \text{Eq. 3.222}$$

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 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 `COMPDAT` or item 13 of keyword `COMPDATL`.

See also "[The flow-dependent skin factor](#)" in the *ECLIPSE Technical Description* for further discussion on the treatment of flow-dependent skin.

ECLIPSE 300

---

**Note:** If non-Darcy Forchheimer effects for gas flow have been enabled via 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 those set in this keyword. See "[Forchheimer parameter based well D-factors](#)" in the *ECLIPSE Technical Description* for further details.

---

## Example

```

WDFAC
-- well    well
-- name    D-factor
   PROD1  1.5E-3  /
   PROD2  1.2E-3  /
/

```

## WDFACCOR D-factor correlation (flow-dependent skin factor for gas)

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- 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](#).
- Coefficient A in the D-factor correlation  
DEFAULT: 0.0
- Power B of a connected grid block permeability in the D-factor correlation  
DEFAULT: 0.0
- 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 [COMPDAT](#) or [COMPDATL](#) or [WDFAC](#) or WDFACCOR is found in the schedule.

The calculated D-factor may be written to the summary file using the keyword [CDFAC](#).

---

**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 [WDFACORR](#) keyword.

---

### The D-factor correlation

Following [Dake \[Ref. 30\]](#), the expression for the non-Darcy flow D-factor used here is:

$$D = A \cdot K_e^B \cdot \varphi^C \cdot \frac{K_e}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \quad \text{Eq. 3.223}$$

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,  $K_e = (k_x k_y)^{1/2}$ .

$\varphi$  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

$$D = \alpha \cdot \beta \cdot \frac{Ke}{h} \cdot \frac{1}{r_w} \cdot \frac{\gamma_g}{\mu_g} \tag{Eq. 3.224}$$

where  $\beta$  is the coefficient of inertial resistance, given by

$$\beta = const \cdot K_e^B \cdot \phi^C \tag{Eq. 3.225}$$

$\beta$  has Forchheimer units. A laboratory determined relationship for  $\beta$  can be obtained by using a  $const = 883.90$  (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.

	Metric	Field	Lab	PVT-Metric
Unit	day / sm <sup>3</sup>	day / Mscf	hr / scf	day / sm <sup>3</sup>
$\alpha$	2.24460 x 10 <sup>-10</sup>	6.83352x10 <sup>-8</sup>	5.41375 x 10 <sup>-11</sup>	2.25533 x 10 <sup>-10</sup>

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  $A = 6.04e - 5$ ,  $B = -1.1045$  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)
/
    
```

## WDRILPRI Puts wells into the prioritized drilling queue

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

### ECLIPSE 100

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' are opened as soon as allowed by constraints on drilling rate and number of open wells per group, irrespective of group flow rates.

The WDRILPRI 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. 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 /
/
```

## WDRILRES Prevents two wells being drilled in the same block

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## WDRILTIM

## Controls on the automatic drilling of new wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

*ECLIPSE 100 only*

- 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

*ECLIPSE 100 only*

4. Compartment number for the well (a positive integer or zero).

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.

ECLIPSE 100

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 /
```

QDRILL /  
/

## WECON

## Economic limit data for production wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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.

ECLIPSE 100

ECLIPSE 100

ECLIPSE 300

If, in compositional mode, a well is controlled by gas volume rate, surface volumes are obtained by  $(Q_g^m RT_{\text{ref}}) / P_{\text{ref}}$ , 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.

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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.

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), stb/Mscf (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 WEL SPECS)

ECLIPSE 100

PLUG 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.

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

11. Secondary maximum water cut limit.

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:  $\text{sm}^3/\text{sm}^3$  (`METRIC`), `stb/stb` (`FIELD`), `scc/scc` (`LAB`),  $\text{sm}^3/\text{sm}^3$  (`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:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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:  $\text{sm}^3/\text{day}$  (METRIC), stb/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{m}^3/\text{day}$  (METRIC),  $\text{rb}/\text{day}$  (FIELD),  $\text{rcc}/\text{hr}$  (LAB),  $\text{m}^3/\text{day}$  (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](#), [WORKLIM](#), [GRUPRIG](#) and [PRORDER](#).

## Example

```
WECON
PROD1 500 1* 0.8 10.0 1* CON N INJ1 /
PROD2 100 5*                Y /
PROD3 100 /
/
```

## WECONCMF Economic limit data on component mole fractions for production wells

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
WECONCMF
PROD1 1 0.05 +CON /
/
```

## WECONINJ Economic limit data for injection wells

x	ECLIPSE 100
	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#).

2. Minimum economic injection rate.

If the injection rate falls below the specified minimum, the well is shut or stopped (see item 9 in keyword [WEL SPECS](#)). 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 [WLIMTOL](#).

### Example

```
WECONINJ
INJ1 500 RATE /
```

INJ2 800 POTN /  
/

## WECONT

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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 /
/
/
```

## WEFAC

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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.

#### ECLIPSE 100

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

```

WEFAC
' P * '    0.9 /
PROD2     0.8 /
/

```

## WELCNTL

**Resets a well control mode and operating target**

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WCONPROD](#) and [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 [WELTARG](#).

---

**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 [WCONHIST](#) or [WCONINJH](#), then the old historical **target** (for example, oil rate) will still be present as a **limit**. This limit may interfere with the new control mode specified with the [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 [WELTARG](#) keyword to set this limit to a large value (1.0E+8 for example).

---

## Example

```
WELCNTL
  PROD1  ORAT  1500  /
  PROD2  BHP   /
  /
```

## WELDEBUG Controls on debug output for individual wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

This is a 4-figure integer:

The 1<sup>st</sup> figure controls debug from well flow and property calculations.

The 2<sup>nd</sup> figure controls debug from connection flow calculations.

The 3<sup>rd</sup> figure controls debug from calculations involving VFP tables.

The 4<sup>th</sup> figure controls debug from well terms in the solver.

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 100*

*ECLIPSE 300*

*ECLIPSE 300*

*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.

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.

### Example

```
WELDEBUG
  PROD1 2120 /
  PROD2 1001 /
/
```

## WELDRAW

## Sets maximum allowable drawdown for production wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword can be used to set the maximum drawdown allowed for each production well. At each timestep, the maximum drawdown  $D_{max}$  is converted into a maximum production rate  $Q_{max}$  of a selected phase, either liquid (oil + water) or gas, using the relationship

$$Q_{max} = D_{max} \sum_j [T_{wj} M_j] \quad \text{Eq. 3.226}$$

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

$$D = P_{av} - P_w \quad \text{Eq. 3.227}$$

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

$$P_{av} = \frac{\sum_j \{T_{wj} M_j [P_j - H_{wj}]\}}{\sum_j [T_{wj} M_{wj}]} \quad \text{Eq. 3.228}$$

$H_{wj}$  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

$$P_{w,\min} = \text{Max}_j (P_j - H_{wj} - D_{max}) \quad \text{Eq. 3.229}$$

which it converts into a maximum production rate using the current fluid mobilities,

$$Q_{max} = \sum_j \{T_{wj} M_j [P_j - H_{wj} - P_{w,\min}]\} \quad \text{Eq. 3.230}$$

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  $Q_{max}$  given above. If you have set a

specific rate limit for the selected phase (using keyword `WCONPROD` or `WELTARG`), the flow rate is the minimum of  $Q_{\max}$  and the user-specified limit.

Since  $Q_{\max}$  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  $Q_{\max}$  is updated only during the first `NUPCOL` iterations of each timestep, just like the target rates of wells under group control (see keyword `NUPCOL`). For any remaining iterations,  $Q_{\max}$  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 `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 *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 allowable drawdown for the well(s). This is a user defined argument that can be set with keywords `UDQ` and `UDADIMS`.

UNITS: bars (`METRIC`), psi (`FIELD`), atm (`LAB`), atm (`PVT-M`)

3. Phase selected for rate limit.

LIQ For oil wells (unless fully gassed out)

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.

DEFAULT: GAS if the well's preferred phase (`WELSPECS` item 6) is gas, LIQ in all other cases.

4. Do you wish the drawdown limit to be taken into account when calculating the well's production potential?

YES The well's production potential depends on its BHP, THP and drawdown limits

NO The well's production potential depends only on its BHP and THP limits.

Production potentials may be output in the well reports (`WELSPECS` item 7) and the `SUMMARY` file (`WOPP` for example). ECLIPSE uses them to calculate well guide rates or priorities for group control. See "[Well potentials](#)" in the *ECLIPSE Technical Description*.

DEFAULT: 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 /  
/
```

## WELEVNT

## Sets summary mnemonic WPWEM for individual wells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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* /
/
```

## WELLCOMP Specifies a well completion

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day/bars (METRIC), cP.rb/day/psi (FIELD), cP.rcc/hr/atm (LAB), cP.rm<sup>3</sup>/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  $Kh_{net}$  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 /
/
```

## WELLDIMS Well dimension data

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 (/).

- The maximum number of wells in the model.  
 DEFAULT in ECLIPSE 100: 0  
 DEFAULT in ECLIPSE 300: 10
- 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
- The maximum number of groups in the model.  
 DEFAULT in ECLIPSE 100: 0  
 DEFAULT in ECLIPSE 300: 5
- The maximum number of wells in any one group (or child groups in a group; keyword [GRUPTREE](#)).

*ECLIPSE 100 only* This is also the maximum number of wells in a particular local grid refinement (keyword [WELSPECL](#)).

*ECLIPSE 300 only* This is taken as the maximum of the value specified in item 4 and the value specified in item 1.

*ECLIPSE 100* DEFAULT: 0

*ECLIPSE 300* DEFAULT: 10

*ECLIPSE 300 only* 5. The maximum number of stages per separator, including the last stage, which represents the stock tank (see keywords [SEPCOND](#) and [FIELDSEP](#)).  
 DEFAULT: 5

*ECLIPSE 300 only* 6. The maximum number of well streams (see keyword [WELLSTRE](#)).  
 DEFAULT: 10

*ECLIPSE 300 only* 7. The maximum number of mixtures (see keywords [WINJMIX](#) and [WINJORD](#)).  
 DEFAULT: 5

*ECLIPSE 300 only* 8. The maximum number of separators, including the field separator (see keywords [SEPCOND](#) and [FIELDSEP](#)).  
 DEFAULT: 4

*ECLIPSE 300 only* 9. The maximum number of mixture items in a mixture (see keywords [WINJMIX](#) and [WINJORD](#)).  
 DEFAULT: 3

*ECLIPSE 300 only* 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

11. The maximum number of well lists that a well may belong to at any one time (see keywords [WLIST](#) and (ECLIPSE 300 only) [WLISTDYN](#)).

DEFAULT: 1

- ECLIPSE 300 only*
12. The maximum number of dynamic well lists in the simulation (see keyword [WLISTDYN](#)).

DEFAULT: 1

- ECLIPSE 300 only*
13. The maximum number of secondary wells in the simulation.

DEFAULT: 10

## Example

```
WELLDIMS  
20 4 4 5 10 2 4 /
```

## WELLGR

## Specifies well guide rates

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

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)
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
Z45 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**

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC), stb/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

6. The target/limit water volume rate.

UNITS:  $\text{sm}^3/\text{day}$  (METRIC), stb/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

7. The target/limit gas volume rate.

UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

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<sup>3</sup>/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm<sup>3</sup>/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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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 m<sup>3</sup>/day:

```
WELLINJE
I Stream SOLVENT Gas 2* 150000 /
/
```

## WELLKBHP Specify K-index of cell used to define BHP reference depth

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## WELLIM

# Specifies well economic limits

This is an alias for [WECON](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## WELLOPEN Opens a well in the SCHEDULE section

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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   /
/
```

## WELLOPTS **Select well solution options**

WELLOPTS is an alias for [NUPCOL](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## WELLPROD Specifies well production targets

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 <a href="#">LINCOM</a> )
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: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
4. The target water volume rate.  
UNITS: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
5. The target gas volume rate.  
UNITS: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
6. The target liquid (oil+water) volume rate.  
UNITS: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)
7. The target or limiting bottom hole pressure.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)
8. The target tubing head pressure.  
UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (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 [WEL SPECS](#).  
UNITS: rm<sup>3</sup>/day (METRIC), rb/day (FIELD), rcc/hr (LAB), rm<sup>3</sup>/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  $V = nRT / P$ , 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: sm<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/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  $BHP \geq P_{sat} + \Delta P$  where  $P_{sat}$  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  $BHP \geq P_{sat}(T + \Delta T)$  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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
WELLSHUT
WellA1 /
PR14   /
/
```

## WELLSMA

## Calculate connection critical drawdowns for sanding and take action

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELLSPECS](#) 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $N_c$ <sup>th</sup> 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 /
/
```

## WELLSTRW Set composition of injection water stream

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  $N_w^{\text{th}}$  component.

$N_w$  mole fractions should be specified, where  $N_w$  is the number of components in the run.

---

**Note:** If [GASSOL](#) is present, NCMPHW components and water are present in the aqueous phase; hence  $N_w$  is the number of components in the aqueous phase NCMPHW 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](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## WELLTCB **Well temperature cutback**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## WELLTCBC Well temperature cutback critical temperature

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

-- 2TCB tables
WELLTCBT
126 1
140 .95
210 .85
500 .4
/
135 1
300 .6
600 .3
/

```

## WELLWAG Specifies WAG well injection targets

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

(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 /
```

```
/  
WELLWAG  
I T W 91.25 G 91.25 /  
/
```

## 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## WELOPEN Shuts or reopens wells or well connections

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.)

*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).

POPEN 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

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

```

WELOPEN
' * '          / opens all shut or stopped wells in the field
PROD3  SHUT / shuts well PROD3, if it is open
' * '          OPEN 0 0 3 / opens all well connections in layer 3
PROD4  OPEN 0 0 0 / opens all connections of well PROD4
PROD5  SHUT 0 0 4 / shuts connection(s) to layer 4 in well PROD5
PROD6  SHUT 0 0 0 4 6/ shuts connection in completions 4, 5 and 6 in PROD6
/

```

## WELOPENL Shuts or reopens wells or well connections within local grids

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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).

*ECLIPSE 100*

If items 4 - 8 are all defaulted but a local grid name is specified here, the action in item 3 will be performed on any wells named in item 1 that have connections in the named local grid (or any other local grids amalgamated with it).

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*). 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.

ECLIPSE 100

**Note:** The local grid name in item 2 is required only when the keyword has to operate on connections within amalgamated local grids. In all other cases it can be defaulted. But if it is specified in these other cases, it restricts the keyword's operation to wells having connections in the named local grid (or any other local grid amalgamated with it). Whenever item 2 is defaulted, keyword **WELOPEN** may be used instead of **WELOPENL** with the same effect.

## 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
-- (and any other local grids amalgamated with it)
'*' CARF1 SHUT /
-- Open all well connections in layer 3 of local grid CARF2
'*' CARF2 OPEN 0 0 3 /
```

```
-- 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 0 4 6/
/
```

## WELPI

## Sets well productivity/injectivity index values

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## ECLIPSE 300

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:  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC),  $\text{stb}/\text{day}/\text{psi}$  or  $\text{Mscf}/\text{day}/\text{psi}$  (FIELD),  $\text{scc}/\text{hr}/\text{atm}$  (LAB),  $\text{sm}^3/\text{day}/\text{atm}$  (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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 multibranch) 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 defined 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<sup>3</sup> (METRIC), ft<sup>3</sup> (FIELD), cc (LAB), m<sup>3</sup> (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.

---

**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).

- Thermal Option only* 10. Cross-sectional area of the pipe wall used in thermal conductivity calculation

(see note 3).

UNITS: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (PVT-M)

DEFAULT: 0.0

- Thermal Option only* 11. Volumetric heat capacity of the pipe wall.

UNITS: kJ/m<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/K (PVT-M)

DEFAULT: 0.0

- Thermal Option only* 12. 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 NSEGMX (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)

- 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

UNITS: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (PVT-M)

DEFAULT:  $\pi(D^2/4)$

10. Volume of the segment

UNITS: m<sup>3</sup> (METRIC), ft<sup>3</sup> (FIELD), cc (LAB), m<sup>3</sup> (PVT-M)

DEFAULT:  $A \cdot L$

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  $A = \pi \cdot (r_o^2 - r_i^2)$  where  $r_o$  is the outer radius of and  $r_i$  is the inner radius. (ECLIPSE 300 thermal option only.)

UNITS: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (PVT-M)

DEFAULT: item 10 of Record 1

14. Volumetric heat capacity of the pipe wall (ECLIPSE 300 thermal option only).

UNITS: kJ/m<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/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 RPTSCHEM 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  /
/

```

## Example 2

ECLIPSE 100

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    /
/
```

## WELSOMIN **Minimum oil saturation for automatic opening**

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
WELSOMIN
0.2 /
```

## WELSPECL **General specification data for wells in local grids**

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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`)

## 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

*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 "[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.

*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.

---

**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.

ECLIPSE 300

**Note:** Remember to specify the well separator conditions with `WSEPCOND` when using `WELSPECL`.

ECLIPSE 100

**Note:** In a parallel run, if an LGR is switched off, a well cannot be defined within an LGR.

## Examples

### Example 1

ECLIPSE 100

```
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

ECLIPSE 300

```
WELSPECL
PROD1 G1 LGR1 6 8 6000 OIL 6* 1 /
/
```

## WELSPECS General specification data for wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 300

**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 /
/
```

## WELTARG Resets a well operating target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WCONPROD](#) and [WCONINJE](#).

DEFAULT: If the quantity in item 2 is VFP, LIFT or 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 [WTMULT](#).

## Example

```
WELTARG
PROD1  ORAT  1500 /
PROD2  BHP   3000 /
/
```

## WFOAM

## Sets foam concentration for injection wells

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 list 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<sup>3</sup> (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (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      /
/

```

## WFRICSEG **Converts friction well WFRICTN data into a multisegment well**

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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**

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 multibranch) 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'

```
WFRICTN
-- Name      Diam  Roughness  Scalefac
   HORIZ     0.25  1.0E-3    1.0 /
-- I J K  Tlen1  Tlen2   Dirn   RangeEnd  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 /
/
```

### 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

```
WFRICTN
-- Name      Diam  Roughness  Scalefac
   HORIZ     0.25  1.0E-3    1.0 /
-- I J K  Tlen1  Tlen2   Dirn   RangeEnd  Diam
   3 5 6    0.0    1*    I      8 /
/
```

### 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).

```
WFRICTN
-- Name      Diam  Roughness  Scalefac
   MULTI     0.25  1.0E-3    1.0 /
-- I J K  Tlen1  Tlen2   Dirn   RangeEnd  Diam
   3 5 6    0.0    1*    I      8      / main stem (3-8, 5, 6)
   5 5 6    2*          J      1      0.19 / side branch (5, 5-1, 6)
/
```

Or, equivalently,

```
WFRICTN
-- Name      Diam  Roughness  Scalefac
   MULTI     0.25  1.0E-3    1.0 /
-- I J K  Tlen1  Tlen2   Dirn   RangeEnd  Diameter
   3 5 6    0.0    1*    I      8      / main stem (3-8, 5, 6)
   5 5 6          / branch-point (5, 5, 6)
   5 4 6    2*          J      1      0.19 / side branch (5, 4-1, 6)
/
```

## WFRICTNL Designates a well in a local grid as a friction well

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 multibranch) 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 WFRICTNL 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)

DEFAULT: Zero, for the first of these records, or distance to the end of the previous connection or range, for subsequent records

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 [WELSPECL](#))

- 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`.

```

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.

```

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:  $\text{sm}^3/\text{day}$  ([METRIC](#)),  $\text{Mscf}/\text{day}$  ([FIELD](#)),  $\text{scc}/\text{hr}$  ([LAB](#)),  $\text{sm}^3/\text{day}$  ([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:

```
WCONPROD
'GASWEL*' SHUT GRAT 2* 1.0 2* 3000 /
/
```

Place the special gas producers in the drilling queue along with any other wells:

```
QDRILL
PROD4 PROD5 PROD6 GASWEL1 GASWEL2 /
```

Declare the wells as special gas producers and set their incremental rates and maximum number of increments:

```
WGASPROD
GASWEL1 3000 4 /
GASWEL2 4000 2 /
/
```

## WGMEM

## Write geomechanics mechanical earth model (MEM) output data

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

## WGORPEN Sets data for GOR penalty option

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 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 [DATES](#), [TIME](#) or [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 [WCONPROD](#) or [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 [WCONPROD](#). If the wells are placed under oil rate control in [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 [WCONPROD](#) or [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 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 *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. Base GOR

A zero or negative value turns the GOR Penalty option off for the specified well(s).

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC),  $\text{Mscf}/\text{stb}$  (FIELD),  $\text{scc}/\text{scc}$  (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

3. Maximum Rate Limit

UNITS:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

## 4. Initial average GOR

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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)

*ECLIPSE 300 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 /
/

```

## WH2NUM

## Two-phase WAG hysteresis saturation function region numbers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## WHISTCTL Control mode instructions for history matching wells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#))

*ECLIPSE 300 only*

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

## Specifies data for tubing head temperature (THT) calculations

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 WTHHT.

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 keyword [VFPPROD](#)).

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 blank record, containing just a slash (/).

### Example

```

WHTEMP
PA1  1* 40.0 /
PA2  1 /
PA3  2 1*/
/

```

## WI

## 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  
/
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

## WINJEDET

## Make-up of a lumped component in terms of its detailed components for an injection well

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 the 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 /
/

```

## WINJGAS

## Specify the nature of injection gas

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

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 [GCONTROL](#)).

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 /
/

```

## WINJMIX

## Specifies an injection mixture by fraction

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 WAVAAILIM 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 /
/

```

## WINJMULT

## Pressure-dependent injectivity multipliers

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$1.0 + a(P_w - P_f) \text{ when } P_w > P_f \text{ or} \quad \text{Eq. 3.232}$$

$$1.0 \text{ when } P_w < P_f \quad \text{Eq. 3.233}$$

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.

## ECLIPSE 300

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

**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.

**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 **all** connections in the well if the wellbore pressure subsequently declines.

**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 **irreversible** for **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 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 **not** set to 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.

## Example

Injection wells IW1 and IW2 use the default 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.

```

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 /
/

```

## WINJOIL

## Specify the nature of injection oil

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## WINJORD

## Specifies an injection mixture by order

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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).

DEFAULT: 0

## 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 `GCONTROL`).
- 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 /
/
```

## WINJTEMP The nature of injection fluid

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

*Thermal only*

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](#).

*Thermal only*

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), atma (LAB), atma (PVT-M)

*Thermal only*

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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
'I*' 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](#).

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## WLIFT

## Automatic retubing and THP or lift switching data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), stb/day for OIL or LIQUID, or Mscf/day for GAS (FIELD), scc/hr (LAB), sm<sup>3</sup>/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 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 `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 `PRORDER`) or if the well dies under THP control.

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), stb/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (PVT-M)

DEFAULT: 0.0

## 8. New THP limit

When THP switching is performed, this replaces the THP limit previously specified with keyword `WCONPROD` or `WELTARG`. The THP switching is performed in stages if an incremental value is set in item 11.

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (PVT-M)

DEFAULT: 0.0

No THP switching operation is performed if a default or zero value is entered.

---

**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 *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 `PRORDER`) or if the well dies under THP control.

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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 `VFPPROD`.

DEFAULT: 1.0E20 (infinity, that is 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.

```

WLIFT
--well  min  flow  new VFP  new  new      max  new  max  ALQ  THP
--name  flow  phase  tab no   ALQ  efffac  WCT  THP  GLR  incr  decr
  PROD1 1*   1*    0    5.0  0.8    0.5  /
  PROD2 500  LIQ    3    5.0  0.8    /
  PROD3 500  LIQ    4    1*   1*    1*  60  1*  1*  20/
/

```

## WLIFTOPT Gas lift optimization data for wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/day (METRIC), Mscf/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: The largest ALQ value in the well's VFP table (if item 2 is YES).  
Remain unchanged (if item 2 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  $\alpha_w \cdot \Delta Q_o / (\Delta Q_{gli} + \alpha_g \cdot \Delta Q_g)$  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  $\Delta Q_{gli}$  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:  $\text{sm}^3/\text{day}$  (METRIC), Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (PVT-M)

DEFAULT: 0.0

6. Incremental gas rate weighting factor  $\alpha_g$

The incremental gradient is calculated according to the formula  $\alpha_w \cdot \Delta Q_o / (\Delta Q_{gli} + \alpha_g \cdot \Delta Q_g)$  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  $\Delta Q_{gli}$  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

```
.....  
WLIFTOPT  
-- well      optimize    max lift    weighting  min lift  
-- name      gas lift?    gas rate   factor      gas rate  
--           SM3/DAY    SM3/DAY  
  'PA*'      YES  
  'PB*'      YES          1*          1.01       -1.0       / keep these flowing  
  PC1        NO          0.0  
  PC2        NO          5E4                          / no gas lift  
  /                          / fixed gas lift  
.....
```

## WLIMTOL

## Tolerance fraction for economic and other limits

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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).

ECLIPSE 300

---

**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.

---

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- 52 A list of up to 50 well names.

The wells must previously have been declared with the WELSPECS keyword.

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 /
/
.....

```

## WLISTARG Resets operating targets/limits for a list of wells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/

```

## WLISTDYN Sets up a dynamic lists of well names

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 **WELLDIMS** keyword. A well may belong to more than one list at a time, up to a maximum number as set in item 11 of **WELLDIMS**.

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 **WLIST** 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 **WLIST** 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 **ACTIONX** 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.

```

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%.

```

WLISTDYN
 '*HIGHWCT' 'GRP1' 'WWCT' 'HIGHEST' 3 /
/
WTMULT
 '*HIGHWCT' ORAT 0.95 /
/
TSTEP
100.0 /
WTMULT
 '*HIGHWCT' ORAT 0.95 /
/
TSTEP
100.0 /

```

## WLISTNAM Sets up lists of well names, for use in WLISTARG

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELLDIMS](#).

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 - A list of up to 50 well names.

51

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:

```
WLISTNAM
-- 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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

```
WMF
324*1.0
324*0.0
/
```

## WMFVD

## 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 /

```

## WNNAMES

**Water component names**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

```
WNNAMES
Fresh Brine/
```

## WNETCTRL Selects well THP or rate limits to be set by network

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```
WNETCTRL  
'PROD*' THP /  
'GINJ*' GRAT /  
/
```

## WNETDP

## Fixed pressure drops between network node and THP

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## WORKLIM

## Time taken for each automatic workover

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
```

## WORKTHP

## Workover procedure if well dies under THP control

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELSOMIN](#)), 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](#))

*ECLIPSE 100 only*

PLUG Plug back the well (see keyword [WPLUG](#))

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 /
/
```

## WPAVE

## Well block average pressure controls

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WELSPCECS](#)), 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  $\bar{P}_w$  for a given well is a weighted combination of the connection factor weighted average pressure  $\bar{P}_{w,cf}$  and the pore volume weighted average pressure  $\bar{P}_{w,pv}$

$$\bar{P}_w = F2\bar{P}_{w,cf} + (1-F2)\bar{P}_{w,pv} \quad \text{Eq. 3.234}$$

### Connection factor weighted average

This is the average over connections of the average block pressure  $\bar{P}_k$  at each (open or declared, depending on item 4) connection  $k$ , weighted according to the connection transmissibility factors  $T_k$

$$\bar{P}_{w,cf} = \frac{\sum_k T_k \bar{P}_k}{\sum_k T_k} \quad \text{Eq. 3.235}$$

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  $P_{o,k}$

$$\bar{P}_k = F1P_{i,k} + (1-F1)\frac{\sum_{o,k} P_{o,k}}{N_{o,k}} \quad \text{Eq. 3.236}$$

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  $V_{o,k}$

$$\bar{P}_k = \frac{V_{i,k}P_{i,k} + \sum_{o,k} V_{o,k}P_{o,k}}{V_{i,k} + \sum_{o,k} V_{o,k}} \quad \text{Eq. 3.237}$$

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  $V_j$

$$\bar{P}_{w, pv} = \frac{\sum_j V_j P_j}{\sum_j V_j} \quad \text{Eq. 3.238}$$

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).

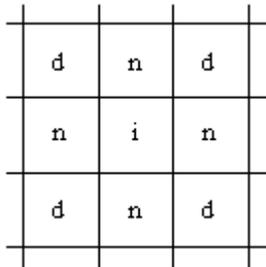


Figure 3.18. Grid block configuration in well block average pressure calculations

## WPAVEDEP Reference depth for well block average pressure calculation

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## WPIMULT

## Multiplies well connection factors by a given value

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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**

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

```

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')
/

```

## WPITAB

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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

```

WPITAB
  'P*' 2 /
  PROD2 1 /
  PROD3 0 /
/

```

## WPLUG

## Sets well plug-back lengths

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [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](#)"). 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 /  
/
```

## WPOLYMER Sets polymer/salt concentrations for injection wells

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword is used to specify the concentration of polymer and salt in the injection stream of each well. If the WPOLYMER 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 [WLIST](#).

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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

```

WPOLYMER
INJ1  0.3  1.0  /
INJ2  0.0  0.0  /
INJ3  0.3  1.0  /
/

```

## WPOLYRED Sets polymer/water mixture viscosity in wells

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 blank record, containing just a slash (/).

### Example

```

WPOLYRED
  INJ1  0.3  /
  INJ2  0.0  /
  INJ3  0.3  /
/

```

## WPOTCALC Control calculation of shut and stopped well potentials

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

ECLIPSE 100 only

---

**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

GRAT Gas rate

THP Tubing head pressure

' ' No regrouping for this well

DEFAULT: ' '

- 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

- 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

- 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

- 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 /
/

```

## WRFT

## Request output to RFT file

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

```
WRFT
/
```

## Example 2

Request current RFT data output for specified wells, plus output when any well is subsequently opened.

```
WRFT
PROD1 /
'A*' /
/
```

## WRFTPLT

## Requests output of well RFT, PLT and segment data to the RFT file

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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)

- 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 WRFPTPLT 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.

*ECLIPSE 100 only*

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.

```

WRFPTPLT
--Well      RFT      PLT      Segment
--Name      Data      Data      Data
  'A*'      FOPN      /
/

```

### Example 2

Request current PLT data output for specified wells, now and at subsequent report times.

```

WRFPTPLT
--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.

```

WRFPTPLT
--River      RFT      PLT      River
--Name      Data      Data      Data
  MYRIVER      NO      NO      REPT/
/

```

## WSALT

## Sets salt concentrations for injection wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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/sm<sup>3</sup> ([METRIC](#)), lb/stb ([FIELD](#)), gm/scc ([LAB](#)), kg/sm<sup>3</sup> ([PVT-M](#))

*ECLIPSE 100 only*

[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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

- 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](#).
- 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
- I - location of connecting grid block(s)
 

DEFAULT: Negative (allows any I-location)
- J - location of connecting grid block(s)
 

DEFAULT: Negative (allows any J-location)
- K - location of connecting grid block(s)
 

DEFAULT: Negative (allows any K-location)
- Number of first completion in range
 

DEFAULT: Negative; allows any completion number up to item 7 (if set)
- 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

```
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)
/
```

## WSCLENL Multiplies deposited well scale by a given value within local grids

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#).

WSCLENL 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 WSCLENL. This keyword should be used when the well is completed in a local grid.

The WSCLENL 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 WSCLENL 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

```

WSCLENL
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')
/

```

## WSCTAB

## Assigns scale deposition and damage tables to wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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](#), [SCDATAB](#) 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 [SCDPTRAC](#).

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 [SCDATAB](#).

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.

DEFAULT: 0

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$\delta P = \left( \frac{\rho_{mix}}{\rho_{cal}} \right) \cdot \left( \frac{\mu_{cal}}{\mu_{mix}} \right)^y \cdot \rho_{mix} \cdot K \cdot q^2 \quad \text{Eq. 3.239}$$

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:

$$K = a_{AICD} \cdot q^{x-2} \quad \text{Eq. 3.240}$$

where

$a_{AICD}$  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:

$$\mu_{mix} = \alpha_o \cdot \mu_o + \alpha_w \cdot \mu_w + \alpha_g \cdot \mu_g \quad \text{Eq. 3.241}$$

where

$\alpha_{o,w,g}$  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 [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) 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,  $a_{AICD}$

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<sup>3</sup>)(rm<sup>3</sup>/day)<sup>x</sup>) (METRIC), psi/((lb/ft<sup>3</sup>)(rft<sup>3</sup>/day)<sup>x</sup>) (FIELD), atm/((gm/cc)(rcc/hr)<sup>x</sup>) (LAB), atm/((kg/m<sup>3</sup>)(rm<sup>3</sup>/day)<sup>x</sup>) (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 the 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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

DEFAULT: 1000.25 kg/m<sup>3</sup> 62.416 lb/ft<sup>3</sup>, 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

ECLIPSE 100 only

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<sup>3</sup>/day (METRIC), ft<sup>3</sup>/day (FIELD), cc/hr (LAB), m<sup>3</sup>/day (PVT-M)

DEFAULT: No limit set

13. Volume flow rate exponent. This quantity should be greater than or equal to 0.0.
14. Viscosity function exponent.
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
-- exp exp
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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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
```

0	1.28	1.02
5	1.36	1.03
45	2.49	2.22
70	2.20	3.78
80	1.67	3.53
88	1.43	2.27
90	0.0	0.0 /

## WSEGDFMD Drift flux model definition for multisegment wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WSEGDFPA](#) keyword. Some of these parameters can also be set separately for individual segments with the keyword [WSEGFMOD](#).

If you wish to use WSEGDFMD, 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 [WSEGDFPA](#).

The WSEGDFMD 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](#)].

SHI-04 The optimized parameters determined by Shi *et al* (2004) [[Ref. 96](#)].

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 [WSEGDFIN](#)

If no table is entered with [WSEGDFIN](#), 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](#)]:

$$m(\theta) = 1.85(\cos \theta)^{0.21}(1 + \sin \theta)^{0.95} \text{ for gas-liquid slip, and}$$

$$m(\theta) = 1.07\cos \theta + 3.23 \sin 2\theta - 2.32\sin 3\theta; (\theta \leq 88^\circ) \text{ for oil-water slip}$$

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 (/).

Parameter	ORIGINAL	SHI-03	SHI-04
Profile coefficient A	1.2	1.2	1.0
Profile coefficient B	0.3	0.6	irrelevant
Profile multiplier $F_v$	1.0	1.0	irrelevant
Drift velocity ramp parameter $a_1$	0.2	0.05	0.06
Drift velocity ramp parameter $a_2$	0.4	0.13	0.21

Table 3.52: Parameters for gas-liquid slip

Parameter	ORIGINAL	SHI-03	SHI-04
Profile coefficient A	1.2	1.0	1.0
Profile coefficient $B_1$	0.4	irrelevant	irrelevant
Profile coefficient $B_2$	0.7	irrelevant	irrelevant
Drift velocity exponent $n'$	2.0	0.95	1.0

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 multisegment wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword may be used to alter the parameters that control the Drift Flux slip model in multisegment 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.

$$C_0 = \frac{A}{[1 + (A-1)\beta^{*2}]} \quad \text{Eq. 3.242}$$

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

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.

$$\beta^* = \frac{(\beta - B)}{(1 - B)} \quad \text{Eq. 3.243}$$

subject to  $0 \leq \beta^* \leq 1$

where

$$\beta = \max\left(\alpha_g, F_v \frac{\alpha_g |j|}{v_{sgf}}\right) \quad \text{Eq. 3.244}$$

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*.

$$\begin{aligned} C'_0 &= A \quad \text{when } (\alpha_{ol} \leq B1) \\ C'_0 &= 1 \quad \text{when } (\alpha_{ol} \geq B2) \\ C'_0 &= A - (A - 1) \left( \frac{\alpha_{ol} - B1}{B2 - B1} \right) \quad \text{when } (B1 < \alpha_{ol} < B2) \end{aligned} \quad \text{Eq. 3.245}$$

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  $v'_d$

$$v'_d = 1.53v'_c(1-\alpha_{ol})^{n'} \quad \text{Eq. 3.246}$$

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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 /
```

## WSEGXSS Segment external source/sink

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$Q = R(P_{ext} - P_{seg}) \quad \text{Eq. 3.247}$$

while  $P_{ext} > P_{seg}$ , where

$Q$  is the volumetric import rate measured at surface conditions

$R$  is a constant set in item 5

$P_{ext}$  is an external pressure set in item 6

$P_{seg}$  is the segment's pressure

The import rate is subject to an upper limit set in item 4.

ECLIPSE 300

- Remove segment fluid at a rate given by

$$Q = R(P_{seg} - P_{ext}) \quad \text{Eq. 3.248}$$

while  $P_{seg} > P_{ext}$ , where

$Q$  is the volumetric rate of fluid removal

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 constant  $R$  and the external pressure  $P_{ext}$  are set in items 5 and 6.

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 [WCONINJE](#) or [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 SUMMARY section mnemonics  $S(W, G) IM(R, T)$  in ECLIPSE 100 and  $S(O, W, G) (IM, RM) (R, T)$  in ECLIPSE 300.

*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 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 *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 containing the external source/sink

An integer between 2 and the NSEGMX mnemonic in the [WSEGDIMS](#) keyword.

3. A character string specifying the source/sink type

The string must be one of the following:

WIMPR Water import at a specified rate

GIMPR Gas import at a specified rate

WIMPP Water import with a specified external pressure, equation [3.247](#)

GIMPP Gas import with a specified external pressure, equation [3.247](#)

*ECLIPSE 300 only*

FREMP Fluid removal with a specified external pressure, equation [3.247](#), with the removal rate expressed as an equivalent **wet gas** volume rate

*ECLIPSE 300 only*

WREMP Fluid removal with a specified external pressure, equation [3.247](#), with the removal rates expressed as an equivalent **water** volume rate

NONE This removes the external source/sink from the segment

DEFAULT: NONE

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:  $\text{sm}^3/\text{day}$  (METRIC), stb/day or Mscf/day (FIELD), scc/hr (LAB),  $\text{sm}^3/\text{day}$  (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:  $\text{sm}^3/\text{day}/\text{bars}$  (METRIC), stb/day/psi or Mscf/day/psi (FIELD), scc/hr/atm (LAB),  $\text{sm}^3/\text{day}/\text{atm}$  (PVT-M)

DEFAULT: 0.0

6.  $P_{ext}$ , the external pressure, for WIMPP, GIMPP, FREMP or WREMP

UNITS: barsa (METRIC), psia (FIELD), atma (LAB), atma (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:  $^{\circ}\text{C}$  (METRIC),  $^{\circ}\text{F}$  (FIELD),  $^{\circ}\text{C}$  (LAB),  $^{\circ}\text{C}$  (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:

```

WSEGXSS
-- 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$\delta P_f = \delta P_1 + \delta P_2 \quad \text{Eq. 3.249}$$

where

$$\begin{aligned} \delta P_1 &= A_1(Q_1 - \text{Lim}_1) \quad \text{when } (Q_1 > \text{Lim}_1) \\ \delta P_2 &= A_2(Q_2 - \text{Lim}_2) \quad \text{when } (Q_2 > \text{Lim}_2) \end{aligned} \quad \text{Eq. 3.250}$$

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  $\text{Lim}$  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  $\text{Lim}$  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

$$\delta P = A(Q - \text{Lim}) \quad \text{when } (Q < \text{Lim} < 0) \quad \text{Eq. 3.251}$$

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.

### ECLIPSE 100

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 first pressure drop term is zero

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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  or  $\text{Mscf}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`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:  $\text{sm}^3/\text{day}$  (`METRIC`),  $\text{stb}/\text{day}$  or  $\text{Mscf}/\text{day}$  (`FIELD`),  $\text{scc}/\text{hr}$  (`LAB`),  $\text{sm}^3/\text{day}$  (`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  /
/

```

## WSEGFMOD Specifies the multi-phase flow model

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `WSEGDFPA`.

DEFAULT: as set in item 1 of keyword `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 `WSEGDFPA`.

DEFAULT: as specified in keyword `WSEGDFMD` or item 4 of keyword `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

$$\beta = \max\left(\alpha_g, F_v \frac{\alpha_g |j|}{v_{sgf}}\right) \quad \text{Eq. 3.254}$$

A and B must obey the relation  $B < (2-A)/A$

DEFAULT: as specified in keyword `WSEGDFMD` or item 5 of keyword `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 `WSEGDFPA`

9. Coefficient B1 for the oil-water profile parameter  $C_0'$  (drift flux model only)

$$\begin{aligned}
 C_0' &= A && \text{when } (\alpha_{ol} \leq B1) \\
 C_0' &= 1 && \text{when } (\alpha_{ol} \geq B2) \\
 C_0' &= A - (A - 1) \left( \frac{\alpha_{ol} - B1}{B2 - B1} \right) && \text{when } (B1 < \alpha_{ol} < B2)
 \end{aligned}$$

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 (WSEGVVALV), a flow limiting valve (WSEGFLLIM), a labyrinth device (WSEGLLABY) 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

```

WSEGFMOD
-- Well  Segments  Flow  Local  Prof  Prof  Prof  Prof  Prof
-- Name  From  To    Model Slip  A    B    v mult  B1  B2
   PROD1  2    6    DF    1.1  1.3  0.3  1.0    0.5  0.9 /
   PROD1  7    12   DF    /
/

```

## WSEGHEAT Segment heat transfer coefficient

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$Q_{ht} = L \cdot (T_{seg} - T_{is,ic,ex}) / R_h$$

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:  $(\text{kJ/m/day/K})^{-1}$  (METRIC),  $(\text{Btu/ft/day/}^{\circ}\text{R})^{-1}$  (FIELD),  $(\text{J/cm/hr/K})^{-1}$  (LAB),  $(\text{kJ/m/day/K})^{-1}$  (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

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup>/sm<sup>3</sup> (METRIC), Mscf/stb (FIELD), scc/scc (LAB), sm<sup>3</sup>/sm<sup>3</sup> (PVT-M)

DEFAULT: 0.0

8. Vaporized oil concentration in the gas,  $R_v$ , 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:  $\text{kg}/\text{sm}^3$  (METRIC),  $\text{lb}/\text{stb}$  (FIELD),  $\text{gm}/\text{scc}$  (LAB),  $\text{kg}/\text{sm}^3$  (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:  $\text{kg}/\text{sm}^3$  (METRIC),  $\text{lb}/\text{stb}$  (FIELD),  $\text{gm}/\text{scc}$  (LAB),  $\text{kg}/\text{sm}^3$  (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
/

```

## WSEGITER Iteration parameters for multisegment wells

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 3<sup>rd</sup> 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  $W_p$

$$P_{seg} = P_{out} + W_p \Delta P + (1 - W_p) \Delta P_{prev} \quad \text{Eq. 3.256}$$

$P_{seg}$  is the pressure in a segment

$P_{out}$  is the pressure in its outlet segment (towards the wellhead)

$\Delta P$  is the calculated pressure drop across the segment

$\Delta P_{prev}$  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

$$N_C = N_R + 1 + N_R \frac{\log(1/F_R)}{\log(F_I)} \quad \text{Eq. 3.257}$$

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 3<sup>rd</sup> record of keyword TUNING (or TUNINGL or TUNINGS).

ECLIPSE 100

DEFAULT: 20

ECLIPSE 300

DEFAULT: 40

- 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

- 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

- 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

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$\delta P = \delta P_{\text{form}} + \delta P_{\text{fric}} \quad \text{Eq. 3.258}$$

$\delta P_{\text{form}}$  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

$$\delta P_{\text{form}} = C_u (\gamma_{\text{inlet}} + (N_c - 1)\gamma_{\text{labyr}} + \gamma_{\text{outlet}}) \rho v^2 \quad \text{Eq. 3.259}$$

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,

$$\delta P_{\text{fric}} = 2C_u f \frac{L}{D} \rho v^2 \quad \text{Eq. 3.260}$$

$f$  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

$N_c$  of the device,

$$L = L_1 + (N_c - 1)L_2 \quad \text{Eq. 3.261}$$

where  $L_1$  and  $L_2$  are constants for the device.

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 3.19. 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 `WELSEGS`, then defined as labyrinth segments with keyword `WSEGLABY`. Labyrinth 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 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  $\delta P = \delta P_{\text{fric}} + \delta P_{\text{form}}$  across a labyrinth segment is reported as the friction pressure loss; the acceleration pressure loss is set to zero.

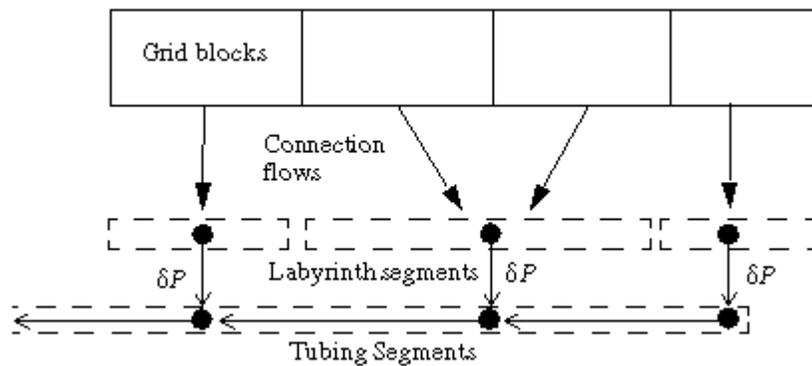


Figure 3.19. Labyrinth segments branching from the tubing

The `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 *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`.

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) 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<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (PVT-M)

DEFAULT: 6.0E-5 m<sup>2</sup>, 6.46E-4 ft<sup>2</sup> 0.6 cm<sup>2</sup>

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,  $\gamma_{inlet}$

DEFAULT: 0.35

12. The dimensionless form loss coefficient for the outlet,  $\gamma_{outlet}$

DEFAULT: 0.5

13. Incremental form loss coefficient for each configuration number,  $\gamma_{labyr}$

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 [WSEGLIM](#), [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   Chnls   Area   L1   L2   Diam   Rough   Cin   Cou   Claby
-- Name      From   To
  PROD1      23    23    8     /
  PROD1      24    24    6     /
  PROD1      25    25    4     4     /
/
    
```

## WSEGLINK Defines the chord segment links for specifying looped flow paths

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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  /

```

PROD 22 31 /  
/

## WSEGMULT Multipliers for segment frictional pressure drops

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

$$\max \left[ A + B(WOR)^C + D \left( \frac{GOR}{GOR_{min}} \right)^E, 1.0 \right] \quad \text{Eq. 3.262}$$

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 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 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

DEFAULT: 0.0

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  $GOR_{\min}$  in the multiplying function

$GOR_{\min}$  must be set  $> 0.0$  if the coefficient  $D$  is non-zero.

UNITS:  $\text{sm}^3/\text{sm}^3$  (METRIC), Mscf/stb (FIELD), scc/scc (LAB),  $\text{sm}^3/\text{sm}^3$  (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

```

WSEGMULT
-- Name      Segments      A      B      C      D      E      GORmin
  PROD1      1      6      2.0 /
  PROD1      7      7     -9.0 10.0 2.0 /
/

```

## WSEGPROP Modifies properties of well segments

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (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: m<sup>3</sup> (METRIC), ft<sup>3</sup> (FIELD), cc (LAB), m<sup>3</sup> (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: m<sup>2</sup> (METRIC), ft<sup>2</sup> (FIELD), cm<sup>2</sup> (LAB), m<sup>2</sup> (PVT-M)

DEFAULT: Unchanged from previous value.

9. **Thermal Option only:** Volumetric heat capacity of the pipe wall.

UNITS: kJ/m<sup>3</sup>/K (METRIC), Btu/ft<sup>3</sup>/°R (FIELD), J/cc/K (LAB), kJ/m<sup>3</sup>/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

```
WSEGPROP
-- Name      Segments  Diam  Rough  Area  Vol
  PROD1      15 17      0.25 /
  PROD2      12 12      0.2  /
/
```

## WSEGPULL Defines a segment to represent a pullthrough pump for a down-hole separator

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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:

$$\delta P = \delta P_{\text{pump}} - \delta P_{\text{reduc}} \quad \text{Eq. 3.263}$$

$\delta P_{\text{pump}}$  represents the pressure boost from a pump working at a specific net power,

$$\delta P_{\text{pump}} = \min\left(C_u \frac{\text{Pow}_{\text{max}}}{V}, \delta P_{\text{max}}\right) \quad \text{Eq. 3.264}$$

$C_u$  is a units conversion constant, given in [table 3.54](#)

$\text{Pow}_{\text{max}}$  is the maximum net power of the pump (set in item 3)

$V$  is the local volumetric flow rate of fluid through the pump

$\delta P_{\text{max}}$  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. In this case the excess water remains in the production flow stream that goes to the surface. 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:

$$\delta P_{\text{reduc}} = F(Q_o - Q_{o \text{ lim}}), \quad \text{when } (Q_o > Q_{o \text{ lim}}) \quad \text{Eq. 3.265}$$

$F$  is a gradient factor (set in item 6)

$Q_o$  is the flow rate of oil through the device, from the water outlet of the separator

$Q_{o \text{ lim}}$  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  $\delta P_{\text{reduc}}$  to cancel out  $\delta P_{\text{pump}}$  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,  $\delta P \ V / C_u$ , can be written to the Summary file with the SUMMARY section keyword SPPOW.

Unit system	$C_u$	power	flow rate	pressure
METRIC	864.0	kW	m <sup>3</sup> /day	bar
FIELD	3.301E5	hp	ft <sup>3</sup> /day	psi
LAB	3.553E4	W	cc/hr	atm
PVT-M	852.7	kW	m <sup>3</sup> /day	atm

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,  $\delta P_{\max}$

UNITS: bars (METRIC), psi (FIELD), atm (LAB), atm (PVT-M)

5. Limiting rate of oil carryover from the water offtake through the pump,  $Q_{o\lim}$

UNITS: sm<sup>3</sup>/day (METRIC), stb/day (FIELD), scc/hr (LAB), sm<sup>3</sup>/day (PVT-M)

DEFAULT: 0.0

6. Gradient factor,  $F$ , for the pressure reduction term

UNITS: bars/(sm<sup>3</sup>/day) (METRIC), psi/(stb/day) (FIELD), atm/(scc/hr) (LAB), atm/(sm<sup>3</sup>/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 `WSEGLABY`, `WSEGFLIM`, `WSEGVALV`, `WSEGSICD` and `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 /
/
```

## WSEGSEP Defines a segment to represent a down-hole separator

x	ECLIPSE 100
	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

NONE 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,  $E_{max}$

A number between 0.0 and 1.0

The holdup fraction,  $\alpha^{of}$ , 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,  $\alpha_{lim}$ , below which the separation efficiency begins to decrease

When the phase's holdup fraction remaining in the separator segment,  $\alpha^{sep}$ , lies below the value  $\alpha_{lim}$ , the separation efficiency decreases proportionally towards zero,  $E = E_{max}(\alpha^{sep} / \alpha_{lim})$ .

The value of  $\alpha_{lim}$  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 /
/

```

## WSEGSICD Defines segments to represent a spiral inflow control device

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

This keyword designates specific well segments to represent a spiral 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 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:

$$\delta P = \left( \frac{\rho_{cal}}{\rho_{mix}} \cdot \frac{\mu_{mix}}{\mu_{cal}} \right)^{1/4} \cdot \frac{\rho_{mix}}{\rho_{cal}} \cdot K \cdot q^2 \quad \text{Eq. 3.266}$$

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:

$$K = \frac{a_{SICD}}{\rho_{cal}} \quad \text{Eq. 3.267}$$

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:

$$\rho_{mix} = \alpha_o \cdot \rho_o + \alpha_w \cdot \rho_w + \alpha_g \cdot \rho_g \quad \text{Eq. 3.268}$$

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:

$$\mu_{mix} = (\alpha_o + \alpha_w) \cdot \mu_{emul} + \alpha_g \cdot \mu_g \quad \text{Eq. 3.269}$$

where

$\mu_{emul}$  is the viscosity of the oil-water emulsion at local conditions

$\mu_g$  is the gas viscosity at local conditions.

The calculation of  $\mu_{emul}$  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.

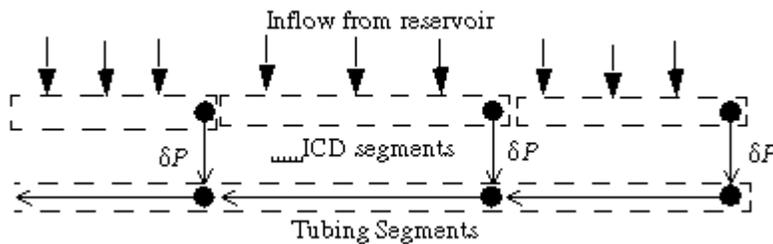


Figure 3.20. ICD segments branching from the tubing

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 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 SICD,  $a_{SICD}$

This is an empirical constant based on measurements of the calibrating fluid flow through a particular type of ICD.

UNITS: bars/(rm<sup>3</sup>/day)<sup>2</sup> (METRIC), psi/(rft<sup>3</sup>/day)<sup>2</sup> (FIELD), atm/(rcc/hr)<sup>2</sup> (LAB), atm/(rm<sup>3</sup>/day)<sup>2</sup> (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:

- 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)
- The scale factor is equal to the absolute value of  $l_{ICD}$ . (item 11 = 1)
- 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<sup>3</sup> (METRIC), lb/ft<sup>3</sup> (FIELD), gm/cc (LAB), kg/m<sup>3</sup> (PVT-M)

DEFAULT: 1000.25 kg/m<sup>3</sup>, 62.416 lb/ft<sup>3</sup>, 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<sup>3</sup>/day (METRIC), ft<sup>3</sup>/day (FIELD), cc/hr (LAB), m<sup>3</sup>/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*):

$$\mu_{wio} = \mu_o \cdot \left( \frac{1}{1 - \left( \frac{0.8415}{0.7480} \cdot \alpha_{wl} \right)} \right)^{2.5} \quad \text{Eq. 3.270}$$

$$\mu_{oiw} = \mu_w \cdot \left( \frac{1}{1 - \left( \frac{0.6019}{0.6410} \cdot \alpha_{ol} \right)} \right)^{2.5} \quad \text{Eq. 3.271}$$

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,  $= \alpha_w / (\alpha_w + \alpha_o)$

$\alpha_{ol}$  is the local oil in liquid fraction,  $= \alpha_o / (\alpha_w + \alpha_o)$

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.

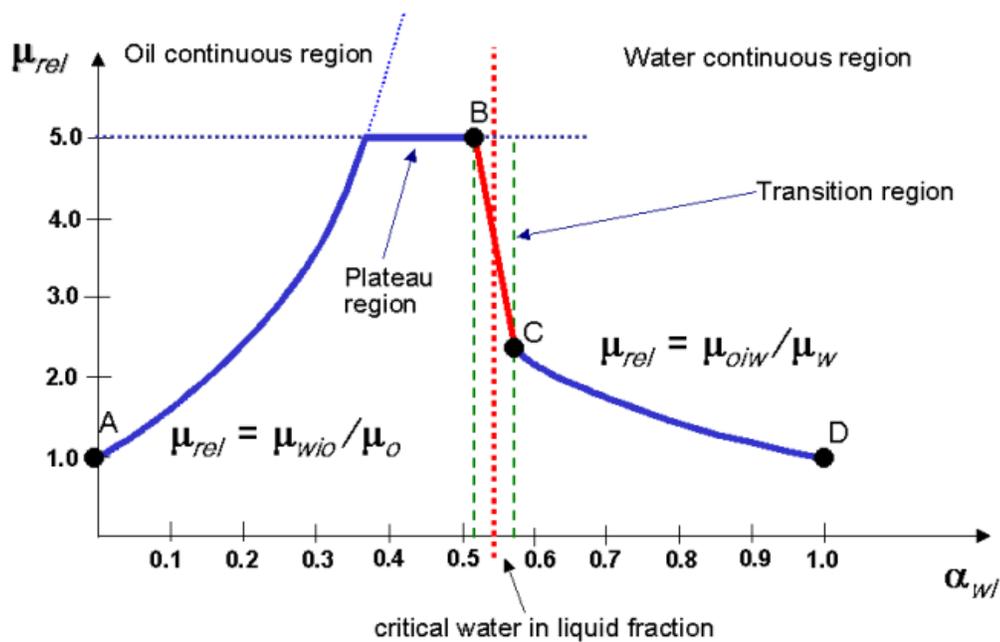


Figure 3.21. Relative emulsion viscosity versus water in liquid fraction

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:

$$\mu_{mix} = \alpha_o \cdot \mu_o + \alpha_w \cdot \mu_w + \alpha_g \cdot \mu_g \quad \text{Eq. 3.272}$$

## 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:

$$q_{ICD} = q_{seg} \times \frac{l_{ICD}}{l_{tubing}} \quad \text{Eq. 3.273}$$

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:

$$q_{ICD} = q_{seg} \times \frac{l_{ICD}}{l_{comp}} \quad \text{Eq. 3.274}$$

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

```

WSEGSICD
--Well  Segments ICD      ICD   dens  visc  crit  width  max  scal
--Name  From  To  stren  length cal  cal  local  trans  visc  fact
--
PROD1   23   23  0.00021  1*   1.0   0.45  0.60   1*   1*  1*/
PROD1   24   24  0.00021  1*   1.0   0.50  0.65   0.02  1*  1*/
PROD1   25   25  0.00042  15   1.0   0.45  0.50   1*    6  1*/
/

```

## WSEGSOLV Sets parameters affecting the multisegment well iterative linear solver

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
x	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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:  $\text{sm}^3/\text{day}$  (METRIC),  $\text{stb}/\text{day}$  (FIELD),  $\text{scc}/\text{hr}$  (LAB),  $\text{sm}^3/\text{day}$  (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 /
```

## WSEGTABL Calculate segment pressure drops from VFP tables

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 `VFP`. 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`), 1/64<sup>th</sup> 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

```

WSEGTABL
-- Well      Segments      VFP      P Drop      -ve      Scale      ALQ
-- Name     From To      Table     Compts     Flow      Pdrop      Value
  PROD1     5   12      4         FH         FIX       LEN /
  PROD1     4   4       5         F-         REV       NO  /
/
    
```

## WSEGVAlV Defines a segment to represent a sub-critical valve

x	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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,  $C_v$
4. Cross-section area for flow in the constriction,  $A_c$

This quantity is used to convert the volumetric flow rate of fluid through the segment into the flow velocity at the constriction,  $v_c$ .

UNITS:  $m^2$  (METRIC),  $ft^2$  (FIELD),  $cm^2$  (LAB),  $m^2$  (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 ( $4 \times \text{area} / \text{perimeter}$ ).

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)

DEFAULT: no change; the initial value is taken from item 8 of keyword WELSEGS unless a previous instance of WSEGVALV defined a value.

8. 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:

$$\delta P = \delta P_{cons} + \delta P_{fric} \quad \text{Eq. 3.275}$$

$\delta P_{cons}$  accounts for the effects of the constriction, and is calculated as:

$$\delta P_{cons} = C_u \frac{\rho v_c^2}{2C_v^2} \quad \text{Eq. 3.276}$$

$C_u$  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.

Unit system	$C_u$	Density	Velocity	Pressure
METRIC	1.0E-5	kg/m <sup>3</sup>	m/s	bars
FIELD	2.159E-4	lb/ft <sup>3</sup>	ft/s	psi
LAB	9.869E-7	gm/cc	cm/s	atm
PVT-M	9.869E-6	kg/m <sup>3</sup>	m/s	atm

Table 3.55: Conversion constant in each unit system for the pressure drop across a device

$\delta P_{\text{fric}}$  accounts for any additional friction pressure loss in the segment containing the valve. It is the standard expression for the homogeneous flow frictional pressure loss through a pipe:

$$\delta P_{\text{fric}} = 2C_u f \frac{L}{D} \rho v_p^2 \tag{Eq. 3.277}$$

$f$  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,

$$q_m = v_c A_c = v_p A_p \tag{Eq. 3.278}$$

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:

$$\delta P_{\text{cons}} = C'_u \frac{\rho q_m |q_m|}{2C_v^2 A_c^2} \tag{Eq. 3.279}$$

$A_c$  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.

Unit system	$C'_u$	Density	Volume flow	Pressure
METRIC	1.340E-15	kg/m <sup>3</sup>	m <sup>3</sup> /day	bars
FIELD	2.892E-14	lb/ft <sup>3</sup>	ft <sup>3</sup> /day	psi
LAB	7.615E-14	gm/cc	cc/hour	atm
PVT-M	1.322E-15	kg/m <sup>3</sup>	m <sup>3</sup> /day	atm

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:

$$K = \frac{C_u'}{(2C_v^2 A_c^2)} \quad \text{Eq. 3.280}$$

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:

$$\frac{A_c}{A_{max}} \quad \text{Eq. 3.281}$$

$A_{max}$  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,  $A_{max}$ . 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

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WSEGXSS](#) 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  /
/

```

## WSEPCOND Specify the separator conditions for wells

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 /
/
```

## WSF

## Water saturation functions (GASWAT/CO2STORE cases)

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

WSF
 0      0
.2      0
.4      .004
.5      .036
.6      .146
.7      .42
.78     1.00 /

```

## WSOLVENT Sets solvent fraction for gas injection wells

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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<sup>3</sup> (METRIC), lb/stb (FIELD), gm/scc (LAB), kg/sm<sup>3</sup> (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 /
/

```

## WTADD

## Adds to a well operating target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

WRAT [ 2 ] Water 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* CVAL [ 11 ] Calorific rate.

*ECLIPSE 300 only* NGL [ 12 ] NGL rate.

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

```
WTADD
  PROD1  ORAT  -200 /
  PROD2  RESV  1000 /
/
```

## WTAKEGAS Order in which subtractions are taken from gas production

	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WINJGAS](#) 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

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

UNITS: °C (METRIC), °F (FIELD), °C (LAB), °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 /
/
```

## WTEMPQ

## Query well name and well list templates

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [WLIST](#).

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.

For LGR amalgamations, any LGR within the amalgamation may be specified

DEFAULT: Apply query to all wells in the field

ECLIPSE 100

### Example

```

WTEMPQ
  \ *PROD* ' /
  ' INJ*' ' LGR-1' /
  '*WLIST?' /
/

```

## WTEST

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## 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](#).

ECLIPSE 100 only

If the string contains a C: Connections closed in automatic workovers are tested individually for reopening **while the well is still open**. A connection is reopened if it does not violate any of the limits set in [WECON](#) or [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.

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 [WTEST](#) keyword.

A value of 0 allows the well to be tested an unlimited number of times.

DEFAULT: 0

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

$$(T - T_o) / (\text{start-up time})$$

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 [WCYCLE](#)).

UNITS: day ([METRIC](#)), day ([FIELD](#)), hr ([LAB](#)), day ([PVT-M](#))

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 /  
/
```

## WTHPMAX Maximum THP design limit data

x	ECLIPSE 100
	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 (/).

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. 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

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

```

WTHPMAX
-- well   max THP   testing   testing   max THP
-- name   limit     limit     limit value to reopen
  PROD1   1000      ORAT      500        800 /
  PROD2   1000      RESV      1*         800 /
/
    
```

## WTMULT

## Multiplies a well operating target or limit

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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

WRAT [ 2 ] Water 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* CVAL [ 11 ] Calorific rate.

*ECLIPSE 300 only* NGL [ 12 ] NGL rate.

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.

See also keyword [WELTARG](#).

## Example

```
WTMULT
PROD1  ORAT  1.5 /
PROD2  RESV  0.8 /
```

## WTRACER Sets tracer concentrations for injection wells

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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,  $T_{\text{conc}}$ . 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  $T_{\text{conc}}$  is taken as the maximum tracer concentration.

*ECLIPSE 100 only*

4. The value of the cumulative tracer factor,  $T_{\text{cum}}$ . 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 H<sub>2</sub>S generating processes.

If a cumulative tracer factor is specified, then the tracer concentration in the injection stream is given by:

$$T_c = \text{MIN}(CI \cdot T_{\text{cum}}, T_{\text{conc}}) \quad \text{Eq. 3.282}$$

where

CI Total cumulative injection at the previous timestep

$T_{\text{cum}}$  Cumulative tracer factor

$T_{\text{conc}}$  Tracer concentration entered in item 3.

UNITS: 1/sm<sup>3</sup> (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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.  $\Delta P$  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

$$\text{BHP}^* = \text{THP} + F_p \cdot (\text{BHP}_{\text{tab}} - \text{THP}) \quad \text{Eq. 3.283}$$

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 /
/

```

## WVFPEXP

## Options to overcome certain problems with THP controlled wells

x	ECLIPSE 100
x	ECLIPSE 300
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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)

---

**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.

---

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.

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

- ECLIPSE 300 only* 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 `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.

**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.

ECLIPSE 100

Item 2 essentially replaces `OPTIONS` keyword [item 9](#). However, for back-compatibility, if `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 `WVFPEXP`.

### Stabilized VFP curves

Another use of the `WVFPEXP` keyword is to activate a check for wells operating on the flat part of a stabilized VFP curve. The VFP *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 `WTEST` keyword if they would be operating on the flat stabilized part of the VFP curve.

### 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.

ECLIPSE 100

By setting item 4 to `YES1` or `YES2` the well is prevented from changing to THP control as long as it would produce at a **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 [item 73](#) in `OPTIONS` keyword, which was formerly used to prevent control mode switching from happening in these circumstances for all wells.

ECLIPSE 300

The well will **always** be prevented from changing to THP control if it would produce at a **higher** rate under THP control. item 4 only controls the reporting of this action.

### Example

```
WVFPEXP
'PROD*' EXP YES YES1 /
/
```

## WWPAVE

## Individual well block average pressure controls

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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 [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 [WWPAVE](#) keyword overwrites any current controls specified by the [WPAVE](#) keyword and in turn can be overwritten by any subsequent use of the [WPAVE](#) 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 can be written to the Summary file using the SUMMARY section keywords [WPI1](#), [WPI4](#), [WPI5](#) and [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 *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. 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.

---

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 [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 /  
/
```

## X

This section contains an alphabetic listing of keywords starting with the letter X.

## XMF

# Specifies cell initial oil composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [EQUIL](#), the usual 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 [XMFVP](#).

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 XMF should be used with caution.

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 *ECLIPSE Technical Description* for further information.

## Example

```
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 /
```

## 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

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

## YLDNUM

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
x	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

**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 [GEOYLDF](#), [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 /
```

## YMF

## Specifies cell initial gas composition

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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
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 NCOMPS 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
/

```

## YOUNGMOD Young's modulus for rock stress balance

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```

-- Field units (psi)
YOUNGMOD
81*1.45E6 81*1.E4 81*1.E5 81*1.8E5 /

```

## YSTC

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
x	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 \* NDX \* 2 \* NDY \* 2 \* 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 NDX cells, input the near top values  $z_{1,1}$ ,  $z_{1,2}$ ,  $z_{2,1}$ ,  $z_{2,2}, \dots$ ,  $z_{i,1}$ ,  $z_{i,2}, \dots$ ,  $z_{NDX,1}$ ,  $z_{NDX,2}$ , followed by the far top values  $z_{1,3}$ ,  $z_{1,4}$ ,  $z_{2,3}$ ,  $z_{2,4}, \dots$ ,  $z_{i,3}$ ,  $z_{i,4}, \dots$ ,  $z_{NDX,3}$ ,  $z_{NDX,4}$ .
- Repeat for each subsequent row of 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 **REFINE** 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.

ECLIPSE 300

If the average value of  $DZ \leq 10^{-6}$  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.

```
ZCORN
24*1000
24*1020
24*1020
24*1040 /
```

## ZCRIT

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```

## ZCRITDET

**Critical Z-factors for detailed components**

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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 [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 Z-factor of each detailed component, in the order in which the detailed components are defined in the [LUMPING](#) keyword.

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 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*.

**Example**

```
ZCRITDET
1.0 0.96 0.994 0.991 /
```

## ZCRITS

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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  $V_c = Z_c RT_c / P_c$ .

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 /
```

## ZFACT1

## Z-factor first coefficient

	ECLIPSE 100
x	ECLIPSE 300
x	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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<sup>3</sup>/kg-M (METRIC), ft<sup>3</sup>/lb-M (FIELD), cc/gm-M (LAB), m<sup>3</sup>/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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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.

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 /
```

## ZFACTORS **Z-factor zeroth coefficient for surface conditions**

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
x	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

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

```
ZFACTORS
1.0 0.96 0.994 0.991 /

ZFACT1S
0.0 0.0 0.0 0.0 /
```

## ZI

## 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  
/
```

	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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.

x	<b>ECLIPSE 100</b>
x	<b>ECLIPSE 300</b>
	<b>SPECIAL</b>
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

## ZIPPY2

## Turns on automatic timestep selection control

x	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
	SOLUTION
	SUMMARY
x	SCHEDULE

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.

‘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.

<i>ECLIPSE 100 only</i>	‘ MAXSTEP = <i>real</i> ’	<p>Sets the maximum timestep.</p> <p>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.</p>
<i>ECLIPSE 100 only</i>	‘ NEXTSTEP = <i>real</i> ’	<p>Sets the next timestep.</p> <p>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 <a href="#">NEXTSTEP</a> keyword.</p>
<i>ECLIPSE 100 only</i>	‘ TIGHTEN = <i>real</i> ’	<p>Sets the amount to tighten convergence tolerances by.</p> <p>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.</p>
	‘ SIM = <i>real</i> ’	<p>Simple timestep selection algorithm</p> <p>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.</p>
<i>ECLIPSE 100 only</i>	‘ AGGRESSIVE=YES ’	Can only be used with the SIM option above
	‘ AGGRESSIVE=NO ’	<p>Attempts to increase timesteps faster when it feels safe to do so.</p> <p>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.</p>

## 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.

<i>ECLIPSE 100 only</i>	Use of ‘ NPTS ’ is compulsory
<i>ECLIPSE 100 only</i>	Use of ‘ PARTITION ’ is optional.
<i>ECLIPSE 100 only</i>	‘ NPTS = <i>integer</i> ’      Number of different timesteps for which data is stored.
<i>ECLIPSE 100 only</i>	‘ PARTITION = <i>integer</i> ’      Number of data partitions.
	<p>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.</p> <p>DEFAULT: 1 Definition of the cost</p>

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\*number of newtons + integer2\*number of  
integer1,  
integer2 ' linear.

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 timestepping 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 *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

```
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.

*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.

### 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.

**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 timestepping 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

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
	PROPS
	REGIONS
x	SOLUTION
	SUMMARY
	SCHEDULE

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 [ZMFVD](#)) 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

```
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 /
```

## ZMFVD

	ECLIPSE 100
x	ECLIPSE 300
	SPECIAL
	RUNSPEC
	GRID
	EDIT
x	PROPS
	REGIONS
	SOLUTION
	SUMMARY
	SCHEDULE

## 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 /
```