CFD Calculations of Cuttings Transport through Drilling Annuli at Various Angles

*Uduak Mme and Pål Skalle

Department of Petroleum Engineering and Applied Geophysics, NTNU, Trondheim, Norway
*Corresponding Author E-mail: uduak.mme@ntnu.no

Abstract

Efficient transportation of cuttings is a vital factor for a good drilling program. Cutting transport in annulus is a complex problem that is affected by numerous parameters. Predicting effective cuttings transport requires all of these parameters to be considered simultaneously.

To better understand the effects of some of the parameters affecting the cutting transportation, Reynolds-Averaged Navier-Stokes or k-epsilon turbulent model, the Navier-Stokes equations, the continuity equation and the power law of non-Newtonian viscosity model were adopted to establish the mathematical model of the cutting transport process in the annulus of the well. The constants of the power law model were evaluated experimentally for three different mud types. The CFD simulation to solve the governing equations was carried out by using ANSYS (FLUENT) commercial code. A combination of hole-drill pipe annulus section was simulated and a model called “Discrete phase model” was used to observe the transportation of cuttings in the annulus.

CFD computations were conducted to determine the effects of annular flow behavior, drilling fluid shear thinning behavior, cutting size and shape on cutting transportation in various orientation of a well bore. In directional drilling, where an eccentric annulus is often used, there is tendency for the cuttings to accumulate in the narrowest gap where the velocity is lowest. Since turbulence tends to suppress such accumulation, knowledge of the velocity profiles in the annulus are essential in the design and operation of these drills. This work evaluates the performance of the numerical method used, comparing the results obtained with those in other reported works, aiming to validate the simulation strategy by the interpolation routines as well as the couplings algorithms adopted.

Keywords: Drilling, Computational fluid dynamic (CFD), Cuttings, Annulus, discrete phase model
Introduction
The circulating drilling fluid rising from the bottom of the well bore carries the cuttings toward the surface. Under the influence of gravity the cuttings tend to sink through the ascending fluid, but by circulating a sufficient volume of mud fast enough to overcome this effect, the cuttings are brought to the surface. The effectiveness of mud in removing the cuttings from the hole depends on several factors such as fluid viscoelastic properties, annular velocity, angle of inclination, drilled cuttings size and their shape. Figure 1 demonstrates the process of hole cleaning in an oil well.

![Diagram of hole cleaning process](image)

**Figure 1:** The flow of drilling fluid and cuttings upward transportation

Various drilling fluids have been widely used in the oil and gas industry to improve lifting capacity of drilled cuttings. Specialized petroleum laboratory at China’s Shandong University has developed an advanced fluid mixed with nanosized particles and superfine powder that significantly improve drilling speed (Saeid et al., 2006). Recent experiments have demonstrated that, nanofluids have attractive properties for application where heat transfer, drag reduction, binding ability or sand consolidation, gel formation, wettability alteration, and corrosive control is of interest (Phuoc et al., 2007). Abouzar et al. (2008) shows that, carbon black nano particles in drilling mud produced a more continues and integrated mud cake, therefore less filtrate and mud cake thickness.

Experimentally, Syed and Jamal (1983) investigated the transport problem in vertical wellbores. Sifferman and becker (1992) conducted experiments in full scale inclined wellbore and Belavadi and Chukwu (1994) have carried out experimental work to measure the cuttings transport efficiency by analysis of the transport ratio,
dimensionless quantities and graphical correlations. Udo Zeidler (1970) studied experimentally the dynamics of the drilled particles to investigate the drilling mud carrying capacity in a vertical well bore.

Cho et al. (2002) studied the forces acting on cuttings layers based on continuity and Navier Stokes equations. They have analyzed the parameters of annular velocity, pressure gradient and fluid rheology. They have introduced the concept of three segments on layers approach. Another mechanistic model was built by Ramadan et al. (2002). They have studied the forces involved on spherical bed particles in inclined channel. Then, by forces equilibrium they determined the critical flow rate. Ramadan et al. (2005) have applied the three layers model on the inclined channel, set certain hypotheses and used the pseudohydrostatic pressure concept in wide range of their analysis. Mingqin et al. (2007) established a mechanistic model to predict the Critical Re-suspension Velocity CRV and the Critical Deposition Velocity CDV building on forces that act on the particles. They predicted that water as drilling fluid is effective in particles bed erosion, but the polymer solution is more helpful than water to prevent bed formation.

The objective of the recent study is to formulate mathematical base mode of turbulent non-Newtonian fluid in annular flow and to use the computational simulation to solve the equation numerically. In directional drilling, where an eccentric annulus is often used, there is tendency for the cuttings to accumulate in the narrowest gap where the velocity is lowest. Since turbulence tends to suppress such accumulation, knowledge of the velocity profiles in the annulus are essential in the design and operation of these drills. Besides, this study covers determination of the effect of the cuttings size, the cuttings shape and the inclination angle. The CFD analysis was carried out at different mud flow rate and drilling fluid properties were measured experimentally and used in the analysis.

**Experimental Work**

Three types of drilling mud: water-based Carboxymethyl cellulose (high viscosity CMC) and Xanthum gum biopolymer (XC) Power –law-type fluids were prepared and tested. Drilling mud varies as different percentages of additives are added. Additives, as its function, change the properties of the drilling fluid. Fluid 1 was obtained by adding 5grams of Carboxyl methyl cellulose (CMC) and 1gram of Xanthum gum biopolymer (XC) to a liter of distilled water. Fluid 2 was obtained by adding 7gram of CMC and 1gram of XC to a liter of distilled water and Fluid 3 was obtained by adding 9gram of CMC and 1gram of XC to a liter of distilled water. High speed mixer is used to mix the fluid and the additives together so that the mixture becomes homogeneous. The density of each mud type was measured by using Fann Scale Mud balance equipment, which is calibrated with fresh water (density 8.33 lb/gallon) and known to be accurate measuring device. VISCOMETER MODEL 35A 6-SPEED 115 V 60, is used in this experiment for measuring the rheological characteristics of the visco-elastic fluids. This viscometer combines accuracy with simplicity of design, and are recommended for evaluating materials that are Bingham plastics and Power law fluids. It gives more meaningful measurement than the marsh
Funnel equipment. Primarily, the experimental measurements were carried out to study the behaviour of the water-based Carboxymethyl cellulose (high viscosity CMC) and Xanthum gum biopolymer mud mixture and to use the experimental results in the CFD calculations.

The shear stress-shear rate graphs of the testing fluids have been approximated to the power low model of non-Newtonian fluid by curve fitting technique. The experimental results fit the power line trend with reasonable deviation. The Mean Square Error (MSE) is used as an indication to the deviation in the form:

$$\varepsilon^2 = \frac{\sum (\text{Experiment} - \text{Trendline})^2}{S} \times 100$$

where, $S$ is the number of the samples. The results of the MSE are 7.5% for Fluid1, 4.1% for the Fluid2 and 2.9% for the Fluid3. The properties of the fluids are shown in Table 1.

Table 1: Fluids physical properties and the power law constants values obtained from the least square fitting with the experimental data

<table>
<thead>
<tr>
<th>Fluid model</th>
<th>Concentration g/L</th>
<th>Density g/cm$^3$</th>
<th>n</th>
<th>$K$ Pa.s$^0$</th>
<th>$\gamma^*$ Sec$^{-1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid1</td>
<td>5 g CMC + 1 g XC</td>
<td>1.003</td>
<td>0.631</td>
<td>0.174</td>
<td>5-1022</td>
</tr>
<tr>
<td>Fluid2</td>
<td>7 g CMC + 1 g XC</td>
<td>1.005</td>
<td>0.591</td>
<td>0.411</td>
<td>5-1022</td>
</tr>
<tr>
<td>Fluid3</td>
<td>9 g CMC + 1 g XC</td>
<td>1.005</td>
<td>0.498</td>
<td>0.638</td>
<td>5-1022</td>
</tr>
</tbody>
</table>

* These are shear rate ranges within which n and K were determined.

Since all the values of n produced by the three fluid models are less than 1, then it is a shear thinning fluid or so-called, pseudo plastic fluid. Similar behaviour is reported by Kim and Yoo (2006) which resulted from their experimental measurement and they concluded that the shear-thinning character of Xanthan gum is more pronounced than those of other polysaccharide gums. The values of consistency index, K are increasing while the values of flow behaviour index, n are decreasing as the concentration of the fluid additives increase, as can be noticed in Table 1.

Modeling and CFD Analysis

Flow physics

The physical aspects of any fluid flow are ruled by three main principles: the conservation of mass, the second law of Newton and the conservation of energy. These basic principles may be expressed in terms of mathematical equations, which are mostly partial differential equations. The computational fluid dynamics technique tries to solve the equations that rule the fluids flow numerically, while the mathematical solution advances in space and time to obtain the complete numerical flow field description.
The comparison of the results of numerical fluid dynamics with experimental data has taken on an important role in validating and establishing limits of many approaches for the ruling equations. Traditionally, it has been shown to be an effective, low cost alternative for total scale measures. This situation has led to an increase in the development of numerical simulation routines and commercial codes.

The flow in the well is simulated as annulus flow. The inner surface of the annulus is the drilling pipe wall and the outer surface is the well wall. The equations governing the flow are derived from Navier-Stokes and continuity equations. The flow in the annulus is considered 2-D flow, laminar or turbulent depending on the flow velocity, but the fluid is non-Newtonian. The fluid moves in z-direction and there is no radial velocity component, i.e.,

\[ U_r = 0. \]

The derived incompressible Navier-Stokes (N-S) equations are as follows:

**Conservation of Mass (Continuity):**

\[ \frac{\partial U_i}{\partial z_i} = 0 \quad \text{(1)} \]

**Newton’s Second Law (Momentum):**

\[ \frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial z_j} = - \frac{\partial P}{\partial z_i} + \frac{\partial}{\partial z_j} \left( \mu \left( \frac{\partial U_i}{\partial z_j} + \frac{\partial U_j}{\partial z_i} \right) \right) \quad \text{(2)} \]

Where \( \mu \) is the physical property of the fluid called molecular viscosity. The molecular viscosity term in Equation 2 must be replaced by any model which describes the visco-elastic behaviour of the non-Newtonian fluids. The Power law model is chosen for this study. Power law model describes the relationship of shear stress and shear rate as:

\[ \tau = K \gamma^n \quad \text{or} \quad \mu = \frac{K \gamma^n}{\gamma} \quad \text{(3)} \]

Where

\( \tau = \text{Shear stress} \)

\( \mu = \text{Fluid viscosity} \)

\( K = \text{Consistency index} \)

\( \gamma = \text{Shear rate} \)

\( n = \text{Flow behaviour index} \)

Value of \( n \) indicates the type of the drilling mud. If \( n \) is lower than 1, then it will be shear thinning fluid. If \( n \) is 1, then the fluid is Newtonian fluid. The value more than 1 will correspond when the fluid is shear thickening fluid.
In Reynolds-Averaged Navier-Stokes (RANS) approach, Reynolds decomposition is applied to the Navier-Stokes equations, which decomposes the turbulent variables into instantaneous (fluctuating) and mean (time averaged) components, and flow equations are averaged over a time scale that is long compared to that of the turbulent motion. The main idea of Reynolds averaging is to decompose the flow to averaged and fluctuating component:

\[ U_i = \overline{U}_i + u_i, P = \overline{P} + p \]  

This process is called Reynolds decomposition. The upper case letters with overbar represent the mean values, the lower case letters represent the fluctuating values on the right hand sides in expressions 4. Since \( \overline{u}_i = 0 \) and \( \overline{u}_j = 0 \), inserting relations 4 into Equations 1 and 2 and time averaging, one obtains relations 6 and 7. By time averaging the velocity component results in expression 5.

\[ \overline{U}_i \overline{U}_j = (\overline{U}_i + u_i)(\overline{U}_j + u_j) = \overline{U}_i \overline{U}_j + \overline{U}_j u_i + \overline{U}_i u_j + u_i u_j, \]

\[ \text{Note: } \overline{U}_i u_j = \overline{U}_i \int_0^T u_j dt = 0 \]

\[ \frac{\partial \overline{U}_i}{\partial z_j} = 0 \]

\[ \frac{\partial \rho \overline{U}_i}{\partial t} + \frac{\partial \rho \overline{U}_i \overline{U}_j}{\partial z_j} = - \frac{\partial \overline{P}}{\partial z_j} + \frac{\partial}{\partial z_j} \left[ \mu \left( \frac{\partial \overline{U}_i}{\partial z_j} + \frac{\partial \overline{U}_j}{\partial z_i} - \overline{\rho u_i u_j} \right) \right] \]

Equations 6 and 7 are called Reynolds-averaged Navier-Stokes equations. They have the same general form as the instantaneous Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values. Additional terms now appear that represent the effects of turbulence. These Reynolds stresses, \( -\rho u_i u_j \), must be modelled in order to close Equation 7. A common method employs the Boussinesq hypothesis to relate the Reynolds stresses to the mean velocity gradients:

\[ -\rho u_i u_j = \mu_i \left( \frac{\partial u_i}{\partial z_j} + \frac{\partial u_j}{\partial z_i} \right) - \frac{2}{3} (\rho \mu + \mu_i \frac{\partial u_i}{\partial z_i}) \delta_{ij} \]

The Boussinesq hypothesis is used in the \( k - \varepsilon \) model as well as other models. There have been numerous numerical methods proposed for the computer simulation of turbulent flows by solving the Reynolds equations. Among them, is the Standard \( k - \varepsilon \) model popularly used in industrial flow simulations. It is a two-equation eddy-viscosity model with the following specification:

\[ \mu_i = \rho \nu_i \]
Where

\[ \nu_t = C_\mu \frac{k^2}{\varepsilon} \]

\( C_\mu \) is a constant, \( k \) is the turbulent kinetic energy and \( \varepsilon \) is the rate of dissipation of turbulent kinetic energy.

\( k \) and \( \varepsilon \) are calculated from their transport equations:

\[ \frac{\partial \rho k}{\partial t} + \frac{\partial \rho \overline{U}_j k}{\partial z_j} = \frac{\partial}{\partial z_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial z_j} \right] + P_k + G_k - \rho \varepsilon \]

\[ \frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \overline{U}_j \varepsilon}{\partial z_j} = \frac{\partial}{\partial z_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial z_j} \right] + \frac{\varepsilon}{k} \left( c_{1\varepsilon} P_k + c_{3\varepsilon} G_k \right) - c_{2\varepsilon} \rho \varepsilon \]

The two terms \( P_k \) and \( G_k \) appearing above correspond to the production of turbulent kinetic energy and the generation of turbulence due to buoyancy respectively. The values for all constants appearing in the transport equations for \( k \) and \( \varepsilon \) are summarized in Table 2.

Table 2: Constants used in \( k - \varepsilon \) turbulence model

<table>
<thead>
<tr>
<th>( C_\mu )</th>
<th>( \text{Pr}_t )</th>
<th>( C_{1\varepsilon} )</th>
<th>( \sigma_k )</th>
<th>( C_{2\varepsilon} )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.00</td>
<td>1.44</td>
<td>1.00</td>
<td>1.92</td>
<td>1.30</td>
</tr>
</tbody>
</table>

**CFD Analysis**

The CFD technique emerged as a result of the current increase in computer processing speed and available memory. Currently, this branch of fluid dynamics complements experimental and theoretical work, providing economically interesting alternatives through the simulation of real flows and allowing an alternative form for theoretical advances under conditions unavailable experimentally.
In this study, the computational mesh represents a virtual flow system, formed by the configurations of the well wall inner diameter of 380 mm and the drill pipe with a diameter of 200 mm. Apart from the simulation carried out to observe the effect of shear thinning behavior of the fluid in cuttings transportation, the drilling pipe was not rotating and is positioned at the center of the well bore. The total depth of the well is 2000 m. GAMBIT 2.4.6 software was used for the annulus modeling and discretization as shown in Figure 2. A one meter section of the geometry has a total mesh layout of 76,457 hexahedric cells that provides the system with a structured mesh.

In the simulation, Fluid model 1 (Fluid1) is used which has density of 1003 kg m\(^{-3}\), consistency index, \(K = 0.174\) and the flow behaviour index , \(n = 0.631\) as obtained from the laboratory tests. The mud feeding rate is varied within a range of 100 ft min\(^{-1}\) to 350 ft min\(^{-1}\) (0.508 m/s – 1.778 m/s ). The cuttings are assumed as Sandstone rock source which has density of 2300 kg m\(^{-3}\) and penetration rate of 0.125 kg sec\(^{-1}\).

In order to calculate the velocity components, the ruling equations were integrated in each computational cell throughout the domain, being discretized following the finite volumes approach. Then they were linearized and solved numerically. The calculations were done using the pressure discretization scheme following the PRESTO routine. For the pressure-velocity coupling the SIMPLEC algorithm was applied. Since the drilling fluid charging rate is varied to involve the laminar and the turbulent regions, the \(k - \varepsilon\) model of Lander and Spalding (1974) was selected for the turbulence simulation. The equations of mass, momentum, turbulent kinetic energy and turbulent dissipation rate were discretized using the QUICK routine scheme due to its better adaptation to hexahedric meshes. The commercial code used to conduct the simulation was ANSYS (FLUENT).
Results and Discussions
The simulation results of the wellbore cleaning performance are presented and discussed at various operational conditions.

Annular velocity profile

![Sample of the velocity profile in the drilling well annulus](image)

**Figure 3:** Sample of the velocity profile in the drilling well annulus

The simulation results of the velocity profile shown in Figure 3 are not consistent with the common velocity profiles of the laminar flow. The velocity profile in the annular cross-sectional area is flattening around the center and the velocity gradient near the wall is high compared with the Newtonian fluid flow profile. Same predicted results are reported by Sunil et al. (2002) for the case of pseudo plastic fluid in their mathematical modeling of the ceramic tape-casting process. Such characteristic is necessary in the cutting transport process, where the drag force is distributed almost evenly on the upstream side of the cutting particles. This will reduce the rotation of the particles during its motion within the mud towards the surface and reduce its trend to rotate and move towards the surfaces of the annulus. If the velocity profile is parabolic, as commonly experienced in the laminar flow, the drag forces magnitude will be higher in the annulus center side than that in the annulus wall sides.

Shear thinning behaviour
The qualitative effect of the drill pipe rotation is shown in Figure 4, in terms of the distribution of the molecular viscosity contours for the drill pipe angular velocity of 300 rpm. It is observed that the viscosity is less near walls (high shear regions) and it is high in the central part (low shear regions). The decrease in viscosity with increase in shear rate is known as shear thinning, and normally is a desirable property, because the viscosity will be relatively low at high shear rates prevailing in the drill pipe, thereby reducing pumping pressures, and relatively high at the low shear rates prevailing in the annulus, thereby increasing cutting carrying capacity. This result
reflect flow situations similar to information on the effect of shaft rotation found in the literature. In the eccentric geometry with a narrow annular space, for example, the same decrease in viscosity with increase in shear rate of a non-Newtonian fluid was observed and reported by Martins et al. (1999). These shear thinning effects of the drilling fluid are relevant in cleaning operations and cuttings transportation in horizontal drilling systems.

**Figure 4:** Distribution of molecular viscosity contours at different sections in the drilling well annulus. Viscosity range = 0.00718 – 0.0647 kg/m-s. Highest value is red and lowest is blue.

**Hole cleaning performance at various well inclinations**
The extent to which the wellbore is cleansed is understood by the fraction of cuttings concentration at the surface of the well at different operational flow velocity. This concentration is produced from the particles transported successfully from the well bottom to the surface. We assumed that there is no energy or mass transfer between particles during the upward motion in the annulus.

**Figure 5:** Effect of well orientations on cuttings transport
Three different orientations have been simulated with 3 mm particles diameter and 0.9 sphereicity. Figure 5 is showing the results at 5, 10 and 15° diverted well orientations. As the drilling direction tends to approach verticality, better cleaning performance can be achieved. At 100 ft/min, around 50% is transported when the well oriented by 5° diverging, while only 20% is transported when the diversion increased to 15°.

Effect of cuttings size
It was decided that three cutting sizes of equivalent diameters of 3, 5 and 7 mm for small, medium and large respectively be selected to study their contribution on the wellbore cleaning performance. The discrete phase model simulation solver in FLUENT which allows the user to inject drilled cuttings with various sizes and shapes into the fluid flow was chosen for this analysis. The simulation results of the concentration at the annulus outlet is shown in Figure 6. It could be concluded that the particles with diameter of 3 mm have better cleaning performance compared with the other larger sizes. This implies that fine particles are the easiest one to clean out.

![Figure 6: Effect of cuttings size on cuttings transport](image)

Effect of cuttings shape
In this analysis, drilled cutting shapes were standardized and compared to the shape of the sphere. This concept is widely known as the shape factor. The shape factor is simply a measure of the sphereicity of a certain cutting particles. A cubic shape is known to have a sphereicity of 85% of that of the sphere, while the sphere has a shape factor of 1. Three cutting shape factors of 1, 0.9 and 0.85 were selected to investigate its effect on the cleaning performance. Cuttings are produced in different sizes and shapes while drilling depending on the rate of penetration, type of formation and drilling fluid properties. The simulation results at various shape factors are shown in Figure 7.
It is clear that transportation of the cuttings is effective at high flow velocities for all types of cuttings shapes. An improvement in the cleaning performance is noticed as the shape factor increases.

**Conclusion**

Fluid flow in annular spaces has received a lot of attention from oil industries, both in drilling operations and in petroleum artificial rising. This work integrates theoretical formulation, experimental measurements and CFD simulation to study the performance of the oil well cleaning performance. In the laboratory experimental measurements, the density and non-Newtonian viscosity behavior of drilling mud are evaluated. Reynolds–Averaged Navier-Stokes and continuity equations combined with power low as definer of the viscosity term were identified to derive the theoretical modelling. GAMBIT software was used for the modeling of the annulus and FLUENT software was used for the simulation. The analyses were conducted for various mud feeding velocities ranging from 100-350 ft/min (0.508 m/s – 1.778 m/s), in 3 diverted orientations.

**Simulation of the mud flow in the annulus had shown that**

1. In the turbulent flow, the velocity profile was flattening over wide area of the annulus. This is necessary to produce uniform drag distribution to lift the cuttings during the transportation process.
2. For 15° diverted orientation, the effective cleaning performance is achieved when the drilling mud charging velocity is high for all types of investigated cuttings.
3. Investigation of cutting size were conducted for 3 mm, 5mm and 7mm and the results suggest that fine particles are the easiest one to clean out.
4. Cuttings shape with 1.0, 0.9 and 0.85 was investigated and it was found that higher sphericity have better cleaning efficiency.
5. Mud rheology plays essential role in cuttings transport. The shear thinning effect of our simulated fluid is relevant in cuttings transportation in horizontal drilling systems. To achieve optimum results for hole cleaning, the best way is with a low viscosity mud in turbulent flow.

References
