Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-31176

MODELING OF NEWTONIAN FLUIDS IN ANNULAR GEOMETRIES WITH INNER PIPE ROTATION

Mehmet SORGUN
Middle East Technical University
Department of Petroleum and Natural Gas Engineering, Ankara, Turkey

Jerome J. SCHUBERT
Texas A&M University, Petroleum Engineering Department, College Station, USA

Ismail AYDIN
Middle East Technical University
Department of Civil Engineering, Ankara, Turkey

M. Evren OZBAYOGLU
University of Tulsa, Petroleum Engineering Department, Tulsa, USA

ABSTRACT

Flow in annular geometries, i.e., flow through the gap between two cylindrical pipes, occurs in many different engineering professions, such as petroleum engineering, chemical engineering, mechanical engineering, food engineering, etc. Analysis of the flow characteristics through annular geometries is more challenging when compared with circular pipes, not only due to the uneven stress distribution on the walls but also due to secondary flows and tangential velocity components, especially when the inner pipe is rotated. In this paper, a mathematical model for predicting flow characteristics of Newtonian fluids in concentric horizontal annulus with drill pipe rotation is proposed. A numerical solution including pipe rotation is developed for calculating frictional pressure loss in concentric annuli for laminar and turbulent regimes. Navier-Stokes equations for turbulent conditions are numerically solved using the finite differences technique to obtain velocity profiles and frictional pressure losses. To verify the proposed model, estimated frictional pressure losses are compared with experimental data which were available in the literature and gathered at Middle East Technical University, Petroleum & Natural Gas Engineering Flow Loop (METU-PETE Flow Loop) as well as Computational Fluid Dynamics (CFD) software. The proposed model predicts frictional pressure losses with an error less than ± 10 % in most cases, more accurately than the CFD software models depending on the flow conditions. Also, pipe rotation effects on frictional pressure loss and tangential velocity is investigated using CFD simulations for concentric and fully eccentric annulus. It has been observed that pipe rotation has no noticeable effects on frictional pressure loss for concentric annuli, but it significantly increases frictional pressure losses in an eccentric annulus, especially at low flow rates. For concentric annulus, pipe rotation improves the tangential velocity component, which does not depend on axial velocity. It is also noticed that, as the pipe rotation and axial velocity are increased, tangential velocity drastically increases for an eccentric annulus. The proposed model and the critical analysis conducted on velocity components and stress distributions make it possible to understand the concept of hydro transport and hole cleaning in field applications.

INTRODUCTION

Accurate prediction of the distribution of frictional pressure losses within the wellbore, especially inside the annulus, is a major factor for proper determination of bottomhole pressure and minimum hydraulic requirements as well as proper determination of serious drilling problems such as loss of circulation, improper rig power selection, hole cleaning problems etc. In the literature, there are a number of studies on pipe rotation effects on frictional pressure loss for Newtonian and Non-Newtonian fluids in annuli[1-5]. The exact solution for incompressible flow in a concentric annulus with pipe rotation, which is also called helical flow, is proposed by Coleman and Noll[6]. Ahmed and Miska[7] adopted this solution for yield-power law fluid and results showed a good agreement with experimental data. Yamaguchi et al.[8] carried out experimental study on spherical coutte flow of a viscoelastic fluid. Riberio et al.[9] solved the governing equations for an incompressible, isothermal, laminar, viscous flow by using finite element approach. They compared 2-D and 3-D finite element modeling results with Mitsuishi and Aoyagis[10] experimental data and model predictions were close to observed results. Wei et al.[11] extended Lou and Peden's[12] study for laminar, helical flow of power law fluids in eccentric annulus. Wan et al.[13] and Escudier et al.[14] investigated numerically the effects of eccentricity and pipe rotation on frictional pressure loss for Newtonian and non-Newtonian fluids. Inner-cylinder rotation caused an increase in the magnitude of the axial pressure gradient during the flow of a Newtonian fluid in eccentric annuli since inertia effects increased due to pipe rotation. In a slightly eccentric annulus, shear-thinning effects could be more dominate and counteract inertial effects, whereas in a highly eccentric annulus, inertial effects predominated for non-Newtonian fluids. Mirzazadeh et al.[15] presented an analytical solution for steady-state, purely tangential flow of a viscoelastic fluid inside concentric annulus. Lugo and Blanco[16] analyzed the efficiency of pipe rotation and eccentricity for non-Newtonian Fluids by using Computational Fluid Dynamics. Results show that pipe rotation is effective when rotation speed is greater than 20 rpm for eccentricity greater than 40%. Woo et al.[17] conducted experimental study related to the effects of the rotational speeds, the flow rates on pressure losses and skin friction coefficients for laminar flows of Newtonian and non-Newtonian fluids inside concentric annulus. More recently, Ozbayoglu and Sorgun [18] developed empirical correlations for determining the frictional pressure loss in eccentric annulus for non-Newtonian fluids including the effect of pipe rotation. They pointed out that, if rotation effects are taken into consideration while there is a rotation, frictional pressure losses were underestimated when compared with the actual case.

In this study, an analytical model for calculating frictional pressure losses and velocity profiles of Newtonian fluids in concentric annuli with pipe rotation was proposed. The equation of motion for incompressible fluid and steady flow for laminar and turbulent regimes is solved using finite difference approximation. Also, a computer code was developed based on the proposed model for predicting frictional pressure loss and fluid velocity distribution inside the annulus including drillpipe rotation. The performance of the proposed model is analyzed by comparing the pressure loss estimations of the model with the experimental data available from the literature (McCann et al.[2]) and data obtained from METU-PETE Flow Loop as well as a Computational Fluid Dynamics (CFD) software program based on finite element model.

MATHEMATICAL MODEL

Concentric annular geometry is represented as a narrow slot in order to simplify and speed-up the calculation process. For conduit cross sections other than simple circular tubes, it is a common practice to use an effective diameter definition for representing annular geometries, termed as the hydraulic diameter, D_h, which is defined as

$$D_h = \frac{4x(cross-sectional)area}{wetted-perimeter} \tag{1}$$

The wetted perimeter is the perimeter in contact with the fluid (Munson et al., 2009). For concentric annulus

$$D_h = \frac{4\pi(r_o^2 - r_i^2)}{2\pi(r_o + r_i)} = 2(r_o - r_i) = D_o - D_i \tag{2}$$

and the hydraulic diameter of parallel plate is 2H

$$D_h = 2(2H) \tag{3}$$

Finally,

$$H = \frac{D_o - D_i}{4} \tag{4}$$

The equation of motion and the equation of continuity in the Cartesian coordinates for turbulent, fully developed flow conditions and incompressible fluid, as illustrated in Fig.1, take the form:

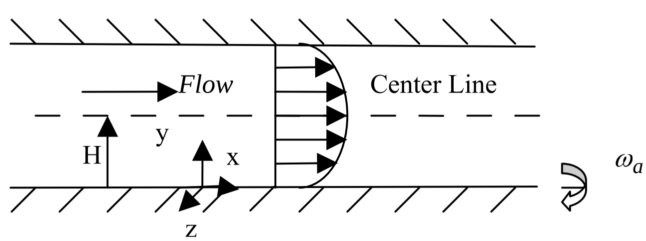

Figure 1: Coordinate System

In this proposed model, Ω is the rotation speed of inner pipe (rpm). Therefore, angular velocity (ω_a (rad/s)) is defined as:

$$\omega_a = \Omega\left(\frac{2\pi}{60}\right) \tag{5}$$

The assumptions used in the analysis were:

- Steady state flow
- Main flow is +x direction (u_x=u, u_y=v , u_z=w_z).
- Fluid is incompressible
- There is no variation of velocity in the axial direction)
- Isothermal system (physical properties are constant)

The equation of continuity is defined as

$$\frac{\partial \rho}{\partial t} + \nabla . \rho v = 0 \tag{6}$$

and for incompressible, ρ = constant, Eq.(6) reduces to

$$\nabla . v = 0 \tag{7}$$

and the equation of continuity may be obtained in rectangular coordinates

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w_z}{\partial z} = 0 \tag{8}$$

The equation of motion in terms of τ is expressed as

$$\rho \frac{Dv}{Dt} = -\nabla p - \nabla . \tau + \rho g \tag{9}$$

and for cartesian coordinates and turbulent flow

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w_z\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \left[\frac{\partial}{\partial x}\tau_{xx} + \frac{\partial}{\partial y}\tau_{yx} + \frac{\partial}{\partial z}\tau_{zx}\right] +$$

$$\rho g_x - \frac{\partial(\rho u' v')}{\partial y} - \frac{\partial(\rho u'^2)}{\partial x} - \frac{\partial(\rho u' w_z')}{\partial z} + \rho \omega_a^2 y \tag{10}$$

The constitutive equation of Newtonian fluids is expressed as

$$\tau_{xy} = \tau_{yx} = \mu\left[\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right] \tag{11}$$

The total shear stress (T_{yx}) can be written as

$$T_{yx} = \tau_{xy} - \rho u' v' \tag{12}$$

and

$$T_{yx} = (\mu + \mu_t)\frac{\partial u}{\partial y}$$

(13)

The effective viscosity can be expressed as

$$\mu_e = \mu + \mu_t \tag{14}$$

According to mixing length model, turbulent viscosity can be expressed as

$$\mu_t = l_m^2 \left|\frac{\partial u}{\partial y}\right| \tag{15}$$

The mixing length and damping function are presented as

$$l_m = H\left[0.14 - 0.08\left(1 - \frac{y}{H}\right)^2 - 0.06\left(1 - \frac{y}{H}\right)^4\right] f_\mu \tag{16}$$

$$f_\mu = 1 - \exp\left(-\frac{y^+}{26}\right) \tag{17}$$

where $y^+ = \frac{y u_*}{\mu}$, $u_* = \sqrt{\frac{\tau_w}{\rho}}$, $\tau_w = -\left(\frac{\partial p}{\partial x} - \rho \omega_a{}^2 H\right) * H$

This model can be extended for non-Newtonian fluids by using viscosity, μ in equation 18.

$$\mu = \frac{1}{4} K \left(\frac{D_o - D_i}{2u}\right)^{1-n} \left(\frac{3n+1}{n}\right)^n \tag{18}$$

where u is the axial fluid velocity, K is the consistency index (Pa s^n), and *n* is the Power-Law index (dimensionless).

Explicit Solution of Governing Equation

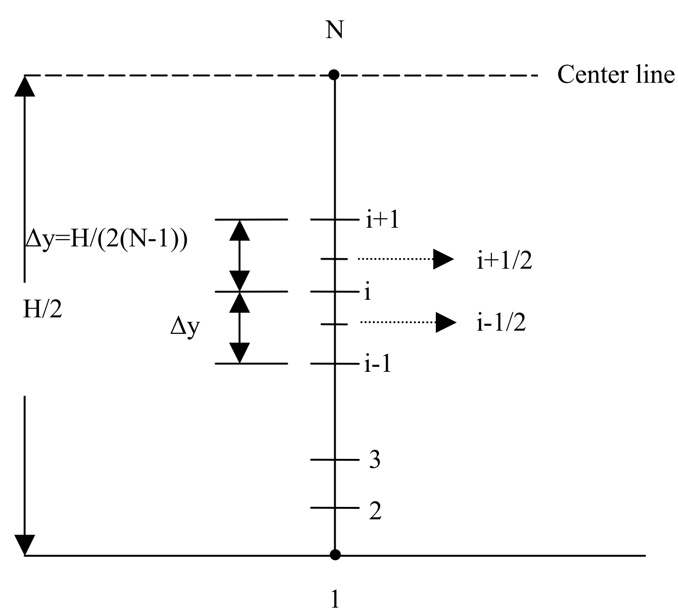

Figure 2: Discrete domain by using finite difference technique

BC1=0; Boundary condition at wall, u=0

BC2=0; Boundary condition at centerline, du/dy=0 ,

Using assumptions, Eq.(10) can be written as

$$\frac{\partial}{\partial y}T_{yx} = \frac{\partial p}{\partial x} - \rho\omega_a^2 y \quad (19)$$

$$M = \frac{\partial p}{\partial x} - \rho\omega_a^2 y \quad (20)$$

$$\frac{\partial}{\partial y}T_{yx} = M \quad (21)$$

By using finite difference approximation, Eq.(21) can be expressed as

$$\mu_{i+1/2}\frac{u_{i+1}-u_i}{\Delta y} - \mu_{i-1/2}\frac{u_i - u_{i-1}}{\Delta y} = (y_{i+1/2} - y_{i-1/2})M \quad (22)$$

$$-A_i u_{i+1} + B_i u_i - C_i u_{i-1} = D_i \quad (23)$$

where $A_i = \frac{\mu_{i+1/2}}{\Delta y}$, $B_i = A_i + C_i$, $C_i = \frac{\mu_{i-1/2}}{\Delta y}$ and

$$D_i = (y_{i+1/2} - y_{i-1/2})\frac{\partial p}{\partial x}$$

CFD SIMULATIONS

Simulations for calculating frictional pressure losses of Newtonian fluids with pipe rotation were set up using a commercial Computational Fluid Dynamics (CFD) package. Navier-Stokes fluid dynamics equation is numerically solved applying finite element method. Simulations were carried out for two horizontal wellbore sections (2.91-1.8 inches, 1.5-1.25 inches). One of most important part of CFD simulations is mesh generation in order to accurately simulate pipe annular flow. Thus, different mesh size was tried in this study and this process was went on until simulation results were no dependent on mesh size. For all of the cases, the geometry is divided approximately 2.3 x 10^6 tetrahedral mesh and the flow was assumed to be steady, incompressible, isothermal and k-ε model used for turbulent flow. After the meshed geometry is imported to CFX Pre, the boundary conditions and initial values have to be described. The inlet was defined as an inlet velocity which depends on the average velocity at the inlet. The inner drill pipe was described as a rotational wall depending on the pipe rotation speed. The outlet was specified as atmospheric pressure. Pressure and velocity profile within the annulus are obtained from the CFD simulations.

The pipe length, L is selected for simulation greater than the maximum entrance length, L_e, in order to eliminate the end effects and to obtain fully developed flow [20],

$$L_e = 0.06(D_o - D_i)N_{R_e} \quad \text{(Laminar flow)} \quad (24)$$

$$L_e = 4.4(D_o - D_i)(N_{R_e})^{\frac{1}{6}} \quad \text{(Turbulent flow)} \quad (25)$$

EXPERIMENTAL WORK

Flow Loop

The experimental testing has been performed using METU-PETE Flow Loop in order to verify the proposed model and to investigate pipe rotation effects on frictional pressure losses in eccentric annuli. The flow loop consists of 2.91 in – 1.8 in annular test section with a length of 12 ft annular section. A centrifugal pump is mounted with a flow capacity of 250 gpm, and the flow rate is controlled and measured using a magnetic flow meter and a pneumatic flow controller, respectively. Inner pipe can be rotated by a rotation system with a rotation speed range of 0-200 rpm. During the flow tests, pressure drop is also measured at a fully developed section on the test section using a digital pressure transducer. Ninety-nine tests have been conducted for a flow rate range of 0-150 gpm and a pipe rotation range of 0-120 rpm.

RESULTS AND DISCUSSION

The frictional pressure losses are obtained for numerous different fluid velocities obtained from the experimental data gathered at METU-PETE Flow Loop and McCann et al.[2] experimental results. The performances of the proposed model are also compared with the experimental data as well as a software based on finite element model (ANSYS).

Model Performance

Verification of this model are performed by comparing McCann et al.[2] experimental data. The pressure gradients are calculated for numerous flow rates at concentric annuli. As seen from Figure 3, this model predicts frictional pressure losses accurately when compared with McCann et al.[2] data.

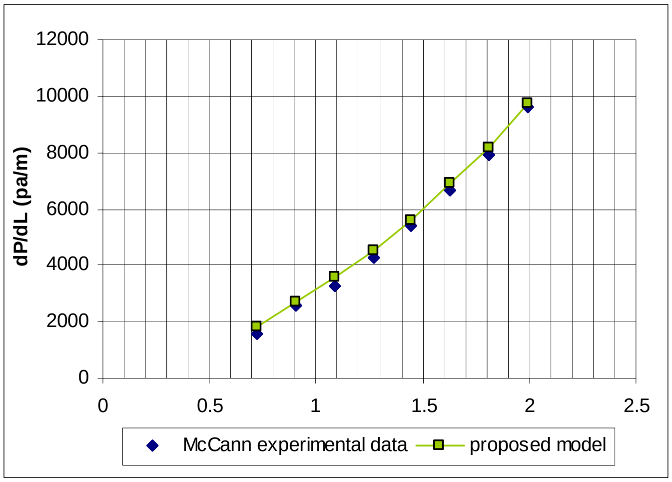

Figure-3 Comparison of the frictional pressure loss of proposed model and McCann experimental values, 1.5in. x 1.25 in concentric annuli

Then, experimental data obtained from METU-PETE Flow Loop was compared with both mathematical model and CFD simulations. A few results are presented in Figure 4-7. The annular frictional pressure losses are measured for concentric annuli with 0,40,60,80,100 and 120 rpm. As seen in these figures, the proposed model estimates the frictional pressure losses with high accuracy for most of the cases.

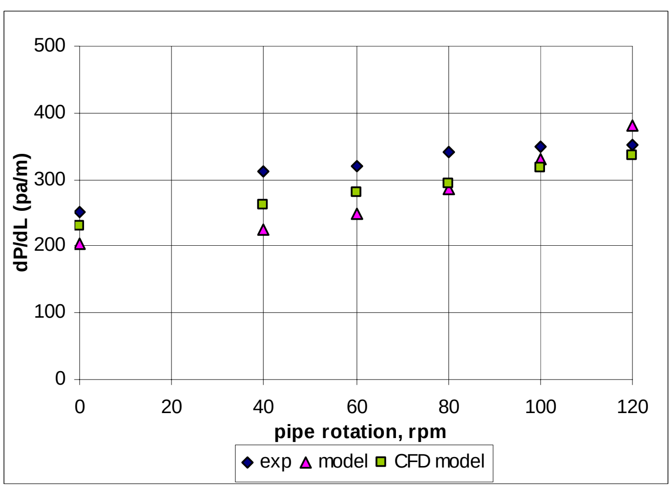

Figure 4- Comparison of proposed model and CFD simulation with experiment for axial fluid velocity= 0.64 m/s

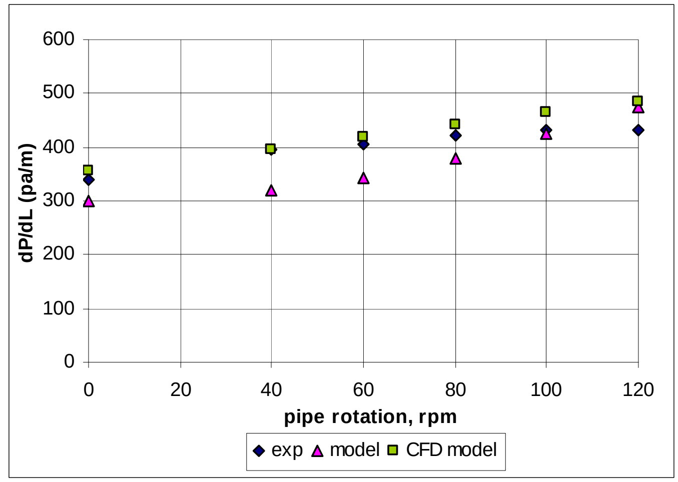

Figure 5- Comparison of proposed model and CFD simulation with experiment for axial fluid velocity= 0.78 m/s

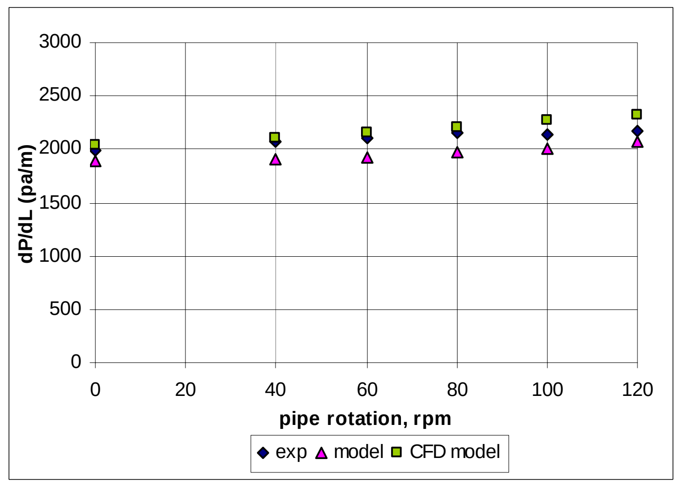

Figure 6- Comparison of proposed model and CFD simulation with experiment for axial fluid velocity= 2.02 m/s

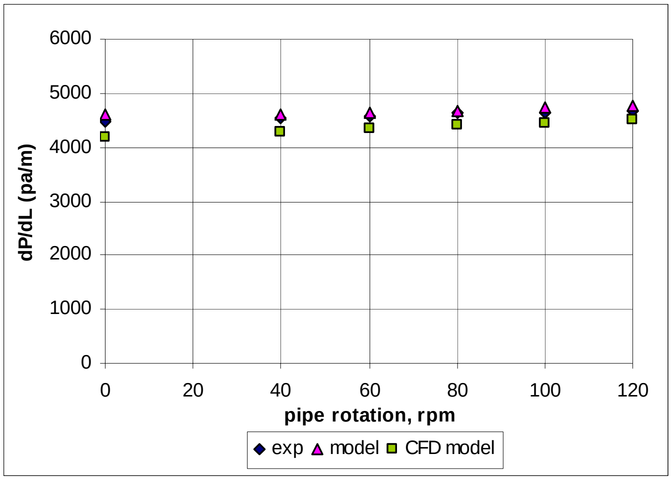

Figure 7- Comparison of proposed model and CFD simulation with experiment for axial fluid velocity=3.08 m/s

Effects of Pipe Rotation on Frictional Pressure Loss and Tangential Velocity in Concentric and Fully Eccentric Annuli

The effects of pipe rotation on tangential velocity for both concentric and fully eccentric annuli is also analyzed using CFD software. Pipe rotation effects on annular frictional pressure for concentric and fully eccentric annuli are shown in Figure 8 and 9. As seen from figure 8, pipe rotation does not have noticeable effects on frictional pressure loss of Newtonian fluid inside concentric annuli.

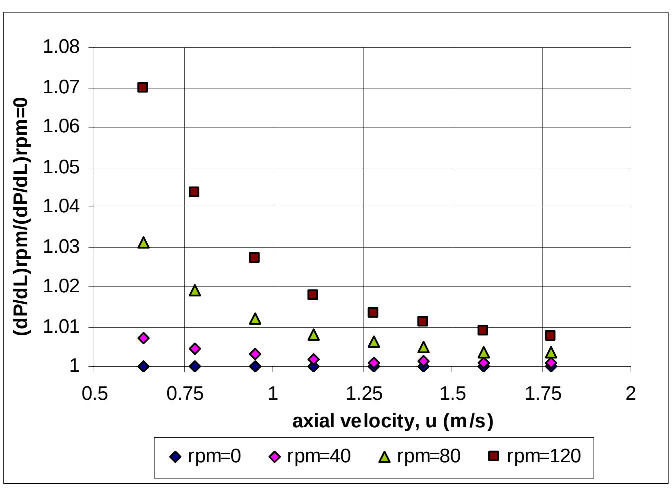

Figure 8- Pipe rotation effects on the frictional pressure loss of water through concentric annuli

However, for a fully eccentric annulus, as the pipe rotation is increased, a significant increase in frictional pressure loss is observed in Figure 9. Additionally, it can be seen that as the flow rates are increased, the effect of pipe rotation on frictional pressure losses diminishes.

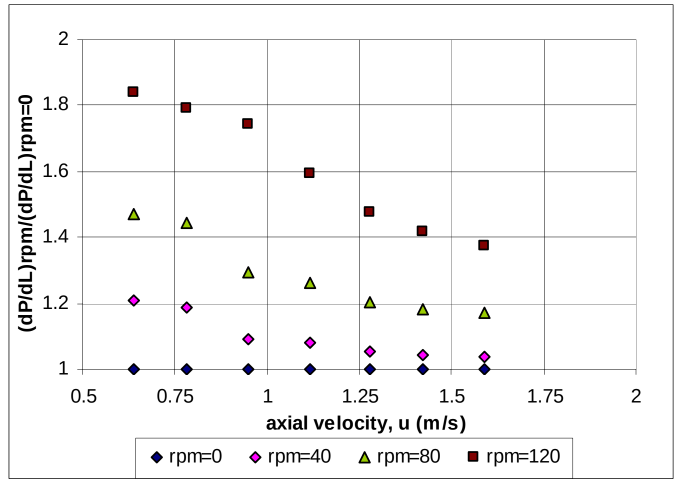

Figure 9- Pipe rotation effects on the frictional pressure loss of water in fully eccentric annuli

If Figure 10 is analyzed, it can be seen that rotating the pipe increases tangential velocity in concentric annulus. However, this increase only directly depends on the pipe rotation speed and the tangential velocity keeps constant for all axial velocity.

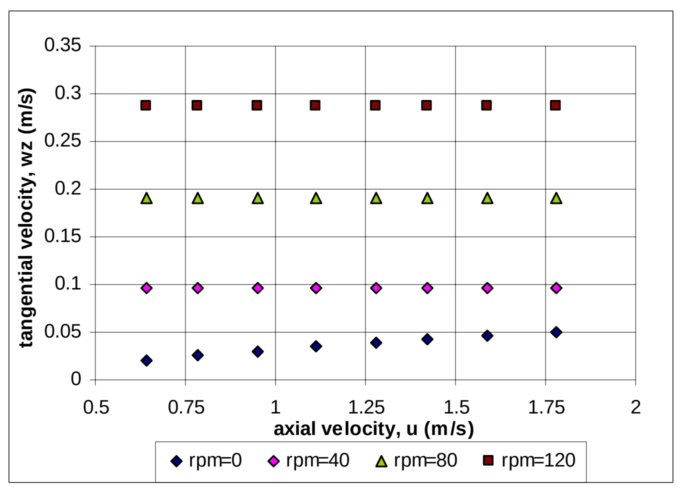

Figure 10- Pipe rotation effects on the tangential velocity in concentric annuli

Figure 11 demonstrates pipe rotation drastically increase tangential velocity of Newtonian fluid in fully eccentric annuli. Moreover, the tangential velocity are increasing as the axial velocity is increased. From the Figure 10 and Figure 11, it can be observed that the influence of pipe rotation on tangential velocity becomes more severe for fully eccentric annulus, when compared with concentric annuli.

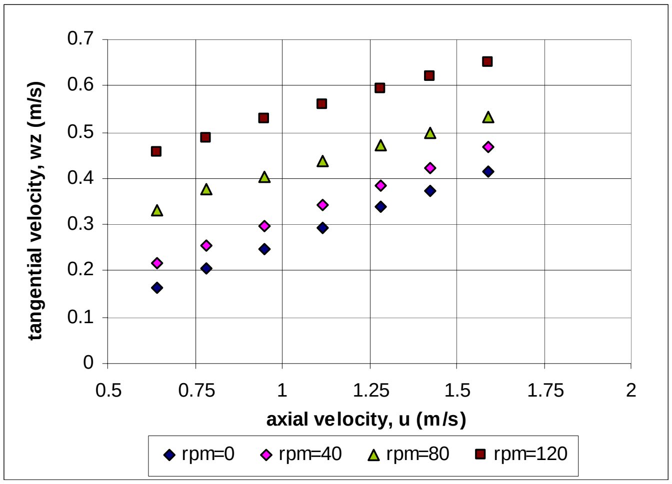

Figure 11- Pipe rotation effects on the tangential velocity in fully eccentric annuli

Error Analysis of Proposed Model

Experimental results and model predictions for annular frictional pressure losses are presented in Figure 12-14 in order to demonstrate the accuracy of the models. Figure 12 shows the lower values of pressure losses and higher values of pressure losses are presented in Figure 13. Solid lines in Figure 12 and 13 represent the perfect match between the experimental and calculated values, and the dashed lines present ± 15% error margin. As seen from these figures, most of the predicted values by using proposed model fall into ± 15% error margin. Furthermore, an error analysis is performed for the proposed model.

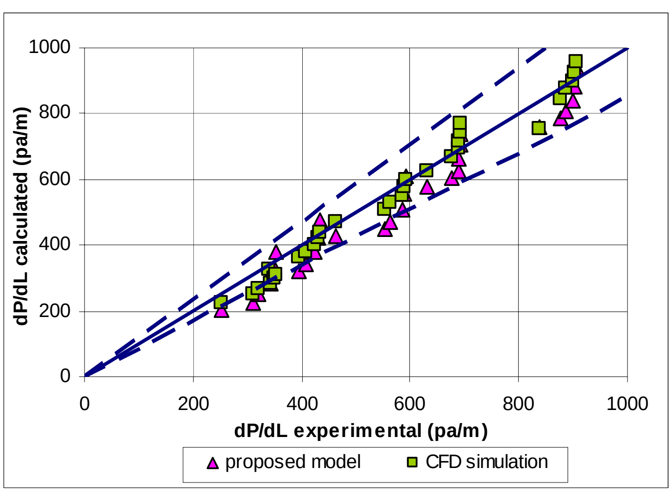

Figure 12- Comparison of the model and CFD estimates with experimental values of frictional pressure losses of water through concentric annulus for low pressure drop

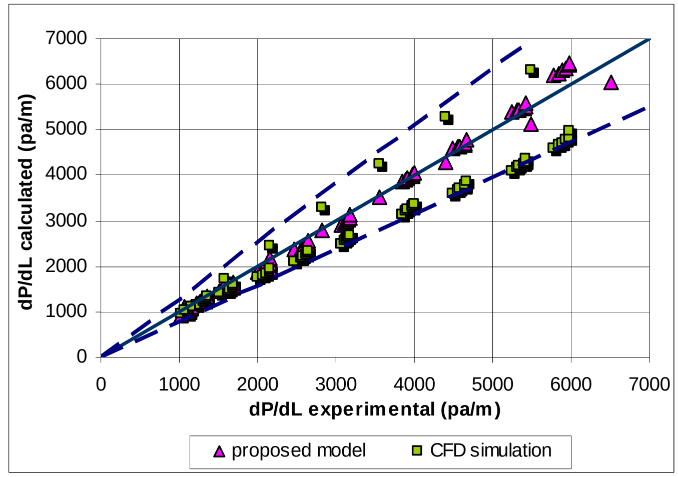

Figure 13- Comparison of the model and CFD estimates with experimental values of frictional pressure losses of water through concentric annulus for high pressure drop

There are 99 data points for different flow rates and pipe rotations, and the error distribution is presented in Figure 14. As seen from Figure 14, the model can estimate the frictional pressure loss with an error of less than 10% for 88 data points and 9 points that fall into an error range of 20% and only 2 data points showed a deviation in excess of 20% and maximum deviation of 23.6 %.

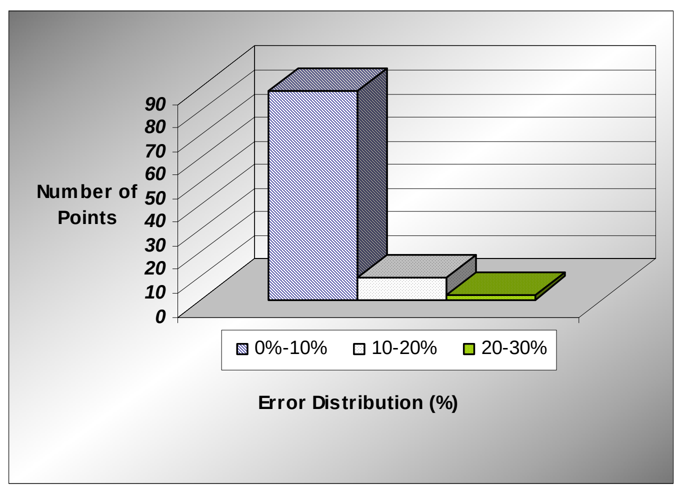

Figure 14- Comparison of the proposed model performance as a function of error distribution

CONCLUSION

A mechanistic model has been developed in order to estimate the annular frictional pressure losses of Newtonian fluids in concentric annuli with pipe rotation. The frictional pressure losses calculated using the proposed model are mostly within a

± 10% error interval in most cases when compared with experimental results. Additionally, the effect of the pipe rotation on frictional pressure loss and tangential velocity is examined using the proposed mechanistic model and CFD simulations. It has been observed that pipe rotation has no noticeable effect on frictional pressure loss of Newtonian fluid in concentric annuli. However, for fully eccentric annulus, pipe rotation drastically increases frictional pressure loss, particularly at lower flow rates. Also, pipe rotation significantly increase tangential velocity, especially if the pipe is in an eccentric position.

Nomenclature

v, u, w_z	Velocity [L/t]
f_f	Friction factor
P	Pressure [m(Lt^2)]
D	Diameter [L]
μ	Viscosity [m/(Lt)]
μ_t	Turbulent viscosity [m/(Lt)]
g	Gravitational Constant [L/t^2]
τ	Shear stress [m/Lt^2]
ρ	Density [m/L^3]
w_a	Angular speed [L/T]

Subscripts

i	Inner
o	Outer

References

1. Delwiche, R.A., Stratabit D.B., Lejeune, M.W.D., Mawet, P.F.B.N., Vighetto, R., “Slimhole Drilling Hydraulics”, paper SPE 24598, presented at the SPE Annual Technical Conference and Exhibition, Washington, 4-7 October, 1992.
2. McCann, R.C., Quigley, M.S., Zamora M., and Slater K.S.: “Effects of High-Speed Pipe Rotation on Pressures in Narrow Annuli”, paper SPE 26343 presented at the 1993 SPE/IADC Drilling Conference, Houston, 3-6 October.
3. Ooms G., Kampman-Reinhartz B.E.,’’ Influence of Drill Pipe Rotation and Eccentricity on Pressure Drop over Borehole During Drilling’’, paper SPE 56638, presented at the 74th Annual Technical Conference and Exhibition, Houston, USA, 3-6 October 1999.
4. Ozbayoglu M.E., Saasen A., Sorgun M., Svanes K., “Effect of Pipe Rotation on Hole Cleaning for Water-Based Drilling Fluids in Horizontal and Deviated Wells”, paper SPE 114965 presented at the 2008 IADC/SPE Asia Pacific Drilling Technology Conference & Exhibition in Jakarta, Indonesia , 25–27 August.
5. Duan M., Miska S., Yu M., Takach N., Ahmed R., Hallman J.,” The Effect of Drillpipe Rotation on Pressure Losses and Fluid Velocity Profile in Foam Drilling”, paper 114185, presented at SPE Western Regional and Pasific Section, California, U.S.A., 31 March-2 April 2008.
6. Coleman, B.D. and Noll, W.’’Helical Flow of General Fluids,’’ J. of App. Phy., 1959, Vol. 30, Issue 10, 1508-1512.
7. Ahmed, R., Miska S., ’’Experimental Study and Modeling of Yield Power-Law Fluid Flow in Annuli with Drillpipe Rotation’’ paper SPE 112604 presented at the 2008 IADC/SPE Drilling Conference in Orlando, March 4-6.
8. Yamaguchi H., Fujiyoshi J., and Matsui H.,’’ Spherical Couette flow of a viscoelastic fluid Part I: Experimental study of the inner sphere rotation’’ J. Non-Newtonian Fluid Mech., 69 (1997) 29-46.
9. Ribeiro P.R. and Podio A.L., ’’The Effect of Rotational Speed and Eccentricity on Annular Flows with Application to Slim Hole Drilling Hydraulics’’ paper SPE 26958 presented at the 1994 Latin American Petroleum Conference, Argentina, 27–29 April.
10. Mitsuishi, N., and Aoyagi, Y., 1973. ”Non-Newtonian Fluid Flow in an Eccentric Annuli,” Journal of Chemical Engineering of Japan, 6: 402-408.
11. Wei, X., Miska S.Z., Takash N.E., Bern P., Kenny P., ’’The Effect of Drillpipe Rotation on Annular Frictional Pressure Loss’’, Journal of Energy Resources Technology, 120:61-66, 1998.
12. Luo, Y., Peden, J.M., 1987. ’’Flow of drilling fluids through eccentric annuli’’ SPE 16692.
13. Wan, S., Morrison, D., Bryden, I.G., ’’The Flow of Newtonian and Inelastic Non-Newtonian Fluids in Eccentric Annuli with Inner-Cylinder Rotation’’, Theoretical Computational Fluid Dynamics, (2000) 13: 349-359.
14. Escudier, M.P., Oliveira, P.J. Pinho, F.T. ‘’Fully Developed Laminar Flow of Purely Viscous Non-Newtonian Liquids Through Annuli, Including The Effects of Eccentricity and Inner-Cylinder Rotation’’, Int. J. Heat and Fluid Flow, (2002) 23, 52-73.
15. Mirzazadeh M., Escudier M.P., Rashidi F., Hashemabadi S.H.,’’ Purely tangential flow of a PTT-viscolestic fluid within a concentric annulus’’, J.Non-Newtonian Fluid Mech. , (2005) 129, 88-97.
16. Lugo D.J, Blanco A.J.,’’ Influence of Pipe Rotation and Reciprocation in Flow of Non-Newtonian Fluids in Eccentric Annular Space’’, FEDSM2006-98278 presented at ASME 2006 2nd Joint U.S.-European Fluids Engineering Summer Meeting, 17-20 July.
17. Woo N., Kim Y., Hwang Y.,’’ Experimental Study on the Helical Flow in a Concentric Annulus With Rotating Inner Cylinder’’, Journal of Fluids Engineering., (2006) 128, 113-117.

18. Ozbayoglu M.E., Sorgun M., “Frictional Pressure Loss Estimation of Non-Newtonian Fluids in Realistic Annulus with Pipe Rotation’’ paper 2009 Canadian International Petroleum Conference (CIPC) in Canada, Calgary, Alberta, 16–18 June.
19. Harley C., Faghri A.,’’ Two-Dimensional Rotating Heat Pipe Analysis’’, Journal of Heat Transfer, Vol. 117,1995.
20. Bird R.B., Stewart W.E., Lightfoot E.N.: Transport Phenomena, John Wiley & Sons, Inc., NJ, 2002.
21. Munson B.R., Young F.D., Okiishi T.D., Huesbsch W.W,‘’ Fundamentals of Fluid Mechanics’’, John Wiley& Sons, Inc., NJ. (2009).
22. Muralidhar K., and Biswas G.,’’Advanced Engineering Fluid Mechanics’’, Alpha Science.(2005).
23. ANSYS Workbench 10.0 (2005).
24. ANSYS Fluids Analysis Guide, November 2004.