Gas-Liquid Flow Through Horizontal Eccentric Annuli: CFD and Experiments Compared

Conference Paper · January 2011

DOI: 10.1115/AJK2011-24007

4 authors:

Mehmet Sorgun
Izmir Katip Celebi Universitesi

Reza Ettehadi Osgouei
Baker Hughes Incorporated

Evren M. Ozbayoglu
University of Tulsa

Murat Ozbayoglu
TOBB University of Economics and Technology

Proceedings of the ASME-JSME-KSME 2011 Joint Fluids Engineering Conference
AJK-Fluids2011
July 24-29, 2011, Hamamatsu, Shizuoka, JAPAN

AJK2011-24007

GAS-LIQUID FLOW THROUGH HORIZONTAL ECCENTRIC ANNULI: CFD AND EXPERIMENTS COMPARED

Mehmet SORGUN
Ataturk University,
Department of Civil Engineering, Erzurum,
Turkey

Reza E. OSGOUEI
Universiti Teknologi Petronas, Perak Darul
Ridzuan, Malaysia

M. Evren OZBAYOGLU
University of Tulsa,
McDougall School of Petroleum Engineering
Department, Tulsa-OK, USA

A. Murat OZBAYOGLU
TOBB Economy & Technology
University, Computer Engineering Department,
Ankara, Turkey

ABSTRACT

Although flow of two-phase fluids is studied in detailed for pipes, there exists a lack of information about aerated fluid flow behavior inside a wellbore. This study aims to simulate gas-liquid flow inside horizontal eccentric annulus using an Eulerian- Eulerian computational fluid dynamics (CFD) model for two-phase flow patterns i.e., dispersed bubble, dispersed annular, plug, slug, churn, wavy annular. To perform experiments using air-water mixtures for various in-situ air and water flow rates, a flow loop was constructed. A digital high speed camera is used for recording each test dynamically for identification of the liquid holdup and flow patterns. Results showed that CFD model predicts frictional pressure losses with an error less than 20% for all two-phase flow patterns when compared with experimental data.

INTRODUCTION

The aerated fluids have many significant benefits like minimize formation damage, increase rate of penetration, reduce drill pipe sticking. There have been numerous models developed for estimating the pressure gradients while liquid-gas mixtures are flowing in pipe, including pure mechanistic models, semi-empirical models, absolute empirical models, etc. However, both hydraulic behavior and mechanism of transport of drill cuttings by using the drilling fluids formed by gas-liquid mixture are not fully understood yet. It is, therefore, necessary to understand better the hydraulics of aerated mudflows in order to calculate accurately desired bottom hole pressure and optimal flow rates for drilling.

Numerous studies on two-phase flow in pipes and annulus are conducted over the years. Hasan and Kabir [1] proposed a model for predicting flow regime, void fraction, and pressure drop in deviated oil wells. Model compared with experimental data and Beggs and Brill [2] method. The agreement between the predictions and the data is quite good. Xiao et al.[3] introduced a mechanistic model predicting two-phase flow pattern, liquid holdup and pressure drop for gas-liquid flow in horizontal and near horizontal pipelines. Chokshi et al. [4] carried out an extensive experimental study and developed a mechanistic model including estimation of flow pattern and pressure drop for two phase flow through vertical tubing. Das et al. [5] proposed a mechanistic model based on the Das et al. [6] experiments for air-water up flow through concentric annuli in order to identify the regime boundaries as functions of the annulus dimensions and phase superficial velocities. Sunthankar et al. [7] developed flow pattern maps for horizontal air-water flow with drillpipe rotation (100 rpm). They showed that the pressure loss increased with drillpipe rotation for air-water flow mainly because of added turbulence to the system due to the rotating drillpipe. Zhou et al. [8]

performed extensive aerated mud experiments in a unique field-scale elevated pressure and elevated temperature flow loop. Results showed that the two-phase flow patterns can be affected by temperature. A decrease of pressure loss in the annulus is observed as the temperature increased. Zhang et al. [9] introduced a multiphase heat transfer model based on the energy balance equations and analyses of the temperature differences and variations in the liquid film, gas core and slug body for different flow patterns of gas-liquid pipe flow at all inclinations from -90 to +90 from horizontal. Julia et al. [10] studied the axial development of gas-liquid two-phase flow regimes in upward vertical flow in an annular channel and developed a flow regime map in a vertical annulus.

In this study, a CFD model based on the commercial software package Ansys CFX 10.0 were used to simulate gas-liquid flow inside horizontal fully eccentric using an Eulerian- Eulerian computational fluid dynamics (CFD) model for two-phase flow patterns in an annulus, i.e., dispersed bubble, dispersed annular, plug, slug, transition zone, wavy annular. This study aims to evaluate the performance of the CFD model frictional pressure loss predictions for two-phase (gas-liquid) flows in horizontal annulus.

THEORY

Inhomogeneous (Interfluid Transfer) Model

In this study, digital image processing techniques was used in order to identify the types of flow patterns, to determine the boundaries of flow patterns and to calculate the liquid holdup in horizontal wells. There is not a standard method for classifying and defining flow patterns, i.e., the investigator's personal interpretation is the usual approach for the development of flow patterns map in annulus geometry. In the present study, flow patterns observed in the horizontal annulus test section are described and named using similar analogies with gas–liquid flow in pipes. Also classification of these flow patterns and determination of boundaries between them were carried out by using quadratic discriminant analysis originally developed in 1936 by Fisher [11]. In order to calculate liquid hold up, all recorded videos were converted to frames by using the Matlab video acquisition toolbox. The extracted frames were collected according to developed flow patterns. In order to label the area of each phase in frames, two different algorithms were developed by using the Image Processing Toolbox software. These algorithms are: 1) labeling by using boundary, 2) labeling by using Pixel intensity (see Osgouei [12] for more details).So before applying CFD cods, flow patterns and liquid hold up were identified based on experiments.

Inhomogeneous model are used to simulate gas-liquid flowing through annulus. In inhomogeneous model, each fluid has different flow field and fluids interact using interphase transfer terms.

Mixture Model

Non-dimensional interphase coefficients may be correlated in terms of the mixture Reynolds number and Prandtl number defined as follows:

$$\mathrm{Re}_{\alpha\beta} = \frac{\rho_{\alpha\beta}\left|U_\beta - U_\alpha\right| d_{\alpha\beta}}{\mu_{\alpha\beta}} \quad (1)$$

$$\mathrm{Pr}_{\alpha\beta} = \frac{\mu C_{P\alpha\beta}}{\lambda_{\alpha\beta}} \quad (2)$$

where $\rho_{\alpha\beta}$, $\mu_{\alpha\beta}$, $C_{P\alpha\beta}$ and $\lambda_{\alpha\beta}$ are density, viscosity, specific heat capacity and thermal conductivity of the mixture respectively, r is the volume fraction and $d_{\alpha\beta}$ is an interfacial length scale. $\rho_{\alpha\beta}$ and $\mu_{\alpha\beta}$ can be written as

$$\rho_{\alpha\beta} = r_\alpha \rho_\alpha + r_\beta \rho_\beta \quad (3)$$

$$\mu_{\alpha\beta} = r_\alpha \mu_\alpha + r_\beta \mu_\beta \quad (4)$$

Free Surface Model

The free surface model attempts to resolve the interphase between the fluids. If there are just two phases in the simulation, the following equation is used for interfacial area density:

$$A_{\alpha\beta} = \left|\nabla r_\alpha\right| \quad (5)$$

When more than two phases are present, this is generalized as follows:

$$A_{\alpha\beta} = \frac{2\left|\nabla r_\alpha\right|\left|\nabla r_\beta\right|}{\left|\nabla r_\alpha\right| + \left|\nabla r_\beta\right|} \quad (6)$$

Volume conservation equation can be expressed as

$$\sum_{\alpha=1}^{N_p} r_\alpha = 1 \quad (7)$$

Governing Equations

Momentum equations can be written as

$$\frac{\partial}{\partial t}\left(r_\alpha \rho_\alpha U_\alpha\right) + \nabla\cdot\left(r_\alpha\left(\rho_\alpha U_\alpha \otimes U_\alpha\right)\right) = -r_\alpha \nabla p_\alpha + \nabla\cdot\left(r_\alpha \mu_\alpha\left(\nabla U_\alpha + \left(\nabla U_\alpha\right)^T\right)\right) + \sum_{\beta=1}^{N_p}\left(\Gamma_{\alpha\beta}^{+} U_\beta - \Gamma_{\beta\alpha}^{+} U_\alpha\right) + S_{M\alpha} + M_\alpha \quad (8)$$

where the phases are α, β and r_α is volume fraction of phase α, U_α is velocity, ρ_α is density, μ_α is viscosity and p_α is the static pressure for α phase. $S_{M\alpha}$ is the momentum source due to external body forces and M_α describes the interfacial forces acting on phase α due to the presence of other phase. $\left(\Gamma^{+}_{\alpha\beta}U_\beta - \Gamma^{+}_{\beta\alpha}U_\alpha\right)$ represents momentum transfer induced by interphase mass transfer.

Continuity equation can be written as

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \cdot (r_\alpha \rho_\alpha U_\alpha) = S_{MS\alpha} + \sum_{\beta=1}^{N_p} \Gamma_{\alpha\beta} \qquad (9)$$

where $S_{MS\alpha}$ is the user specified mass source and $\Gamma_{\alpha\beta}$ is the mass flow rate per unit volume from phase β and α [13].

Experimental Work

Two-phase (gas-liquid) flow experiments were carried out using Middle East Technical University flow loop.

The experimental study consists of two-phase air-water experiments in large-scale annuli. The flow loop consists of a transparent outer casing of 0.074 m ID. and inner pipe of 0.046 m OD. The length of flow loop is 6.4 m. Experiments were carried out without drill pipe rotation. The flow regime has been determined based on visual observations. A centrifugal pump is mounted with a flow capacity of 250 gpm, and the flow rate is measured and controlled using a magnetic flow meter and a pneumatic flow controller, respectively. Also, gas is introduced into the flow loop using a compressor of 120 scfm, and the rate is measured and controlled by a mass flow meter and a pneumatic flow controller, respectively. The experiments were performed in an eccentric annulus using water-air in horizontal position, without inner pipe rotation with the constant temperature 25° C (298.15 K, 77°F). The eccentricity ratio (e /(r_o - r_i) where e is eccentricity, ro and ri are outer casing radius and inner drillpipe radius) in horizontal section is 0.623. The pressure in the annular test section was varied in the range of 15.7-27.7 psi depending on water and air flow rate. During the flow tests, pressure drop is measured at a fully developed section on the test section using a digital pressure transducer. A high-speed digital camera is also mounted to the test section in order to record the experiments. The standard experimental procedure adapted was as follows: Using centrifugal pump, the liquid was pumped at a constant flow rate. Then, the air was introduced with the desired rate. Once both the air and liquid flow rates were stabilized, the data acquisition was activated in order to record flow rates, pressures at critical points, pressure drop inside the test section, etc. At the same time, high-speed camera was recording the flow in the test section for analysis of flow patterns and identification of gas and liquid volume fractions in dynamic conditions. These recordings were used later to confirm the visual observations of the flow patterns.

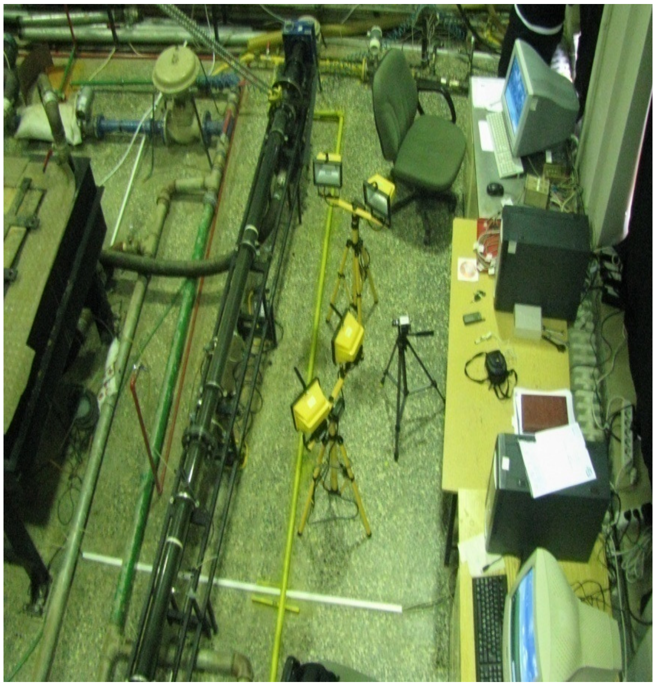

Figure 1. Middle East Technical University, Petroleum & Natural Gas Engineering Flow Loop (METU-PETE Flow Loop)

CFD Simulation of horizontal two-phase gas-liquid flow

First of all, horizontal eccentric annulus geometry was drawn using Ansys Workbench. Boundary conditions of the inlet of annulus was defined as inlet velocities and volume fractions of gas and water while the outlet was specified as static atmospheric pressure and volume fractions of gas and water. In CFD modeling of two phase flow, one of the most important considerations is mesh generation of annular flow area. Therefore, in this study, different mesh sizes were tried and this process went on until model results became independent of mesh size. As a result, the geometry was divided approximately 2.8 x 10^6 tetrahedral meshes. Finally, simulations were solved using finite element method for gas-liquid flow through fully eccentric annulus in order to estimate frictional pressure loss. The two-phase flow pattern classifications obtained from CFD simulations are presented in figure 2-5. In these figures, white color represents water and black color represents gas.

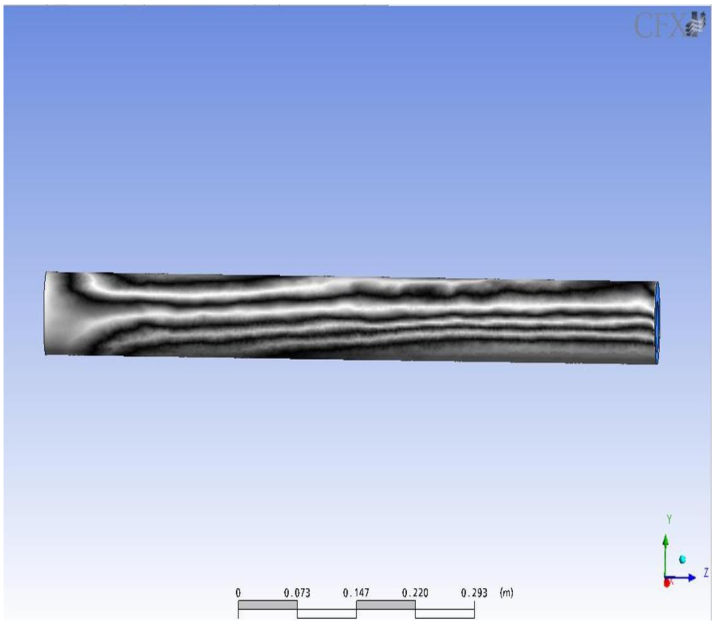

Figure 2. Dispersed bubble flow pattern for Horizontal (Air-Water) Flow (Vsl=2.74 m/s &Vsg=4.29 m/s)

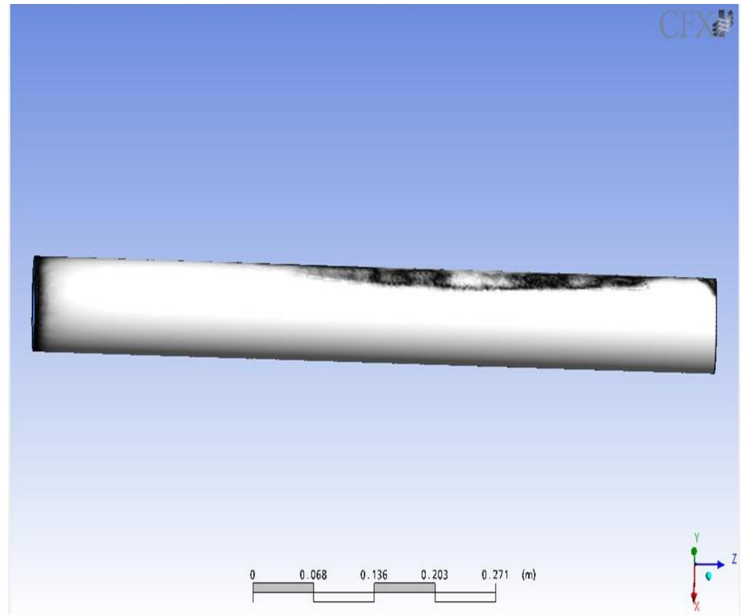

Figure 4. Plug flow pattern for Horizontal (Air-Water) Flow (Vsl=0.76 m/s &Vsg=0.09 m/s)

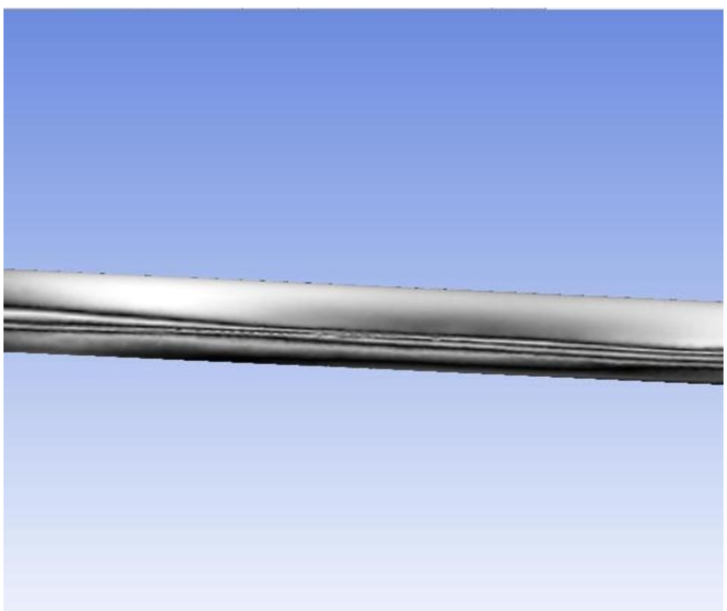

Figure 3. Dispersed annular flow pattern for Horizontal (Air-Water) Flow (Vsl=2.44 m/s &Vsg=14.94 m/s)

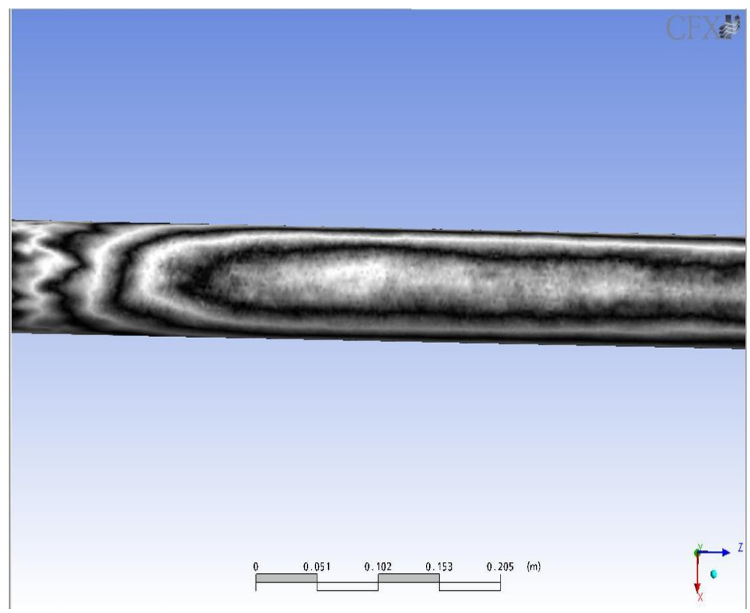

Figure 5. Wavy Annular Flow Pattern for Horizontal (Air-Water) Flow (Vsl=1.98 m/s &Vsg=7.64 m/s)

Results and Discussion

CFD simulations are tested using experimental data obtained from Middle East Technical University Flow Loop to verify the accuracy of model predictions. Comparison of CFD predictions and experimental results for dispersed bubble, dispersed annular, plug, slug, transition zone and wavy annular flow patterns are shown in Figure 6-11. The annular frictional pressure losses are measured for eccentric annuli with fluid super velocity (Vsl) 0.3-3 m/s and gas super velocity (Vsg) 0.3-24 m/s. As seen from these figures, the proposed model estimates the axial frictional pressure gradients with high accuracy for all flow patterns. Moreover, an error analysis for total 130 data points is carried out. CFD model could predict the pressure gradient with an error of less than 20% for 91 data points and 39 points that fall into an error range of between 20% and 30% and maximum deviation of 31.1 %.

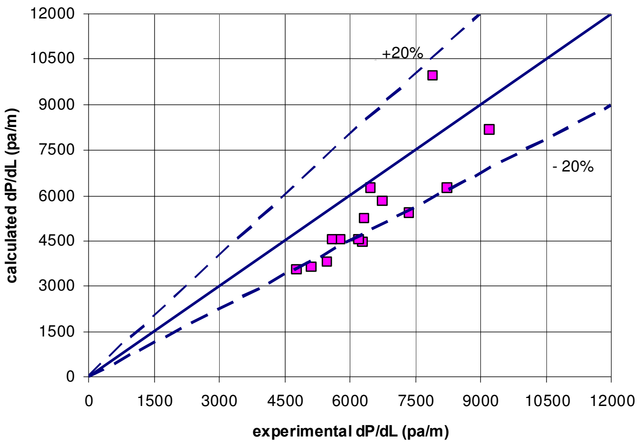

Figure 6. Comparison of CFD predictions and experimental data for dispersed bubble flow pattern

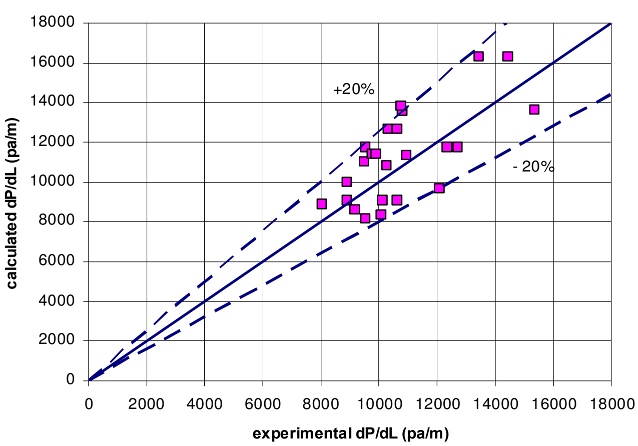

Figure 7. Comparison of CFD predictions and experimental data for dispersed annular flow pattern

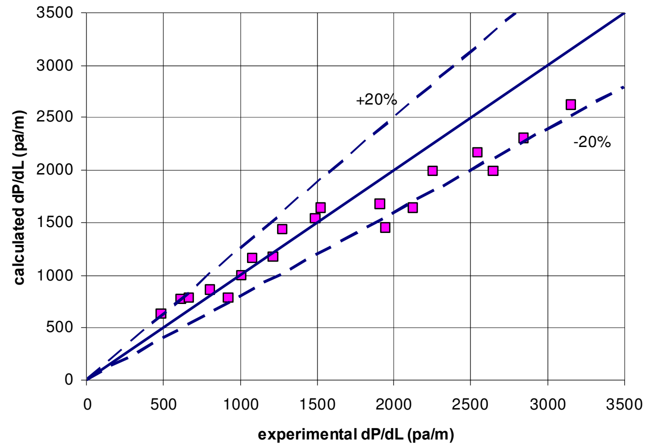

Figure 8. Comparison of CFD predictions and experimental data for plug flow pattern

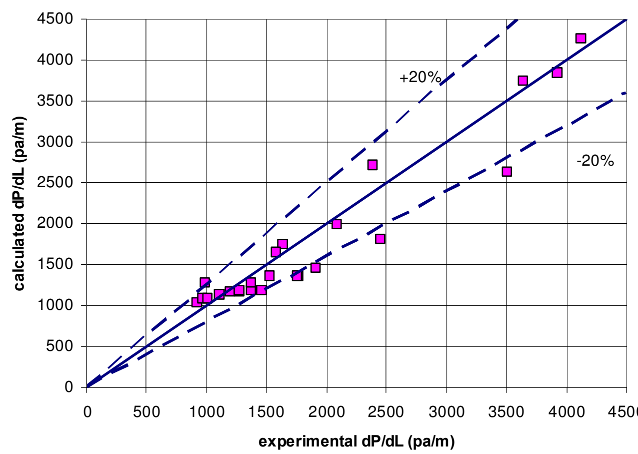

Figure 9. Comparison of CFD predictions and experimental data for slug flow pattern

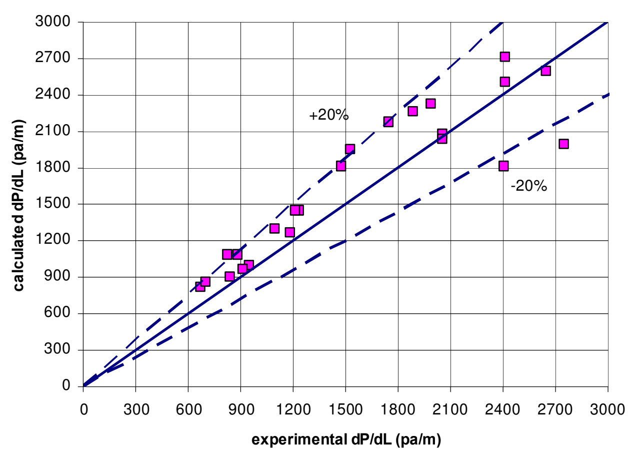

Figure 10. Comparison of CFD predictions and experimental data for transition zone flow pattern

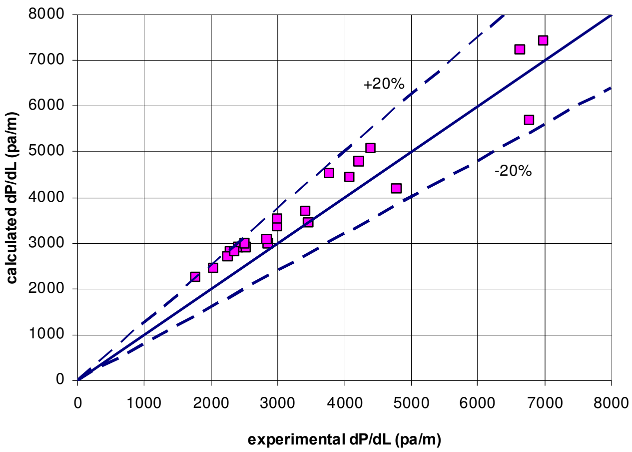

Figure 11. Comparison of CFD predictions and experimental data for wavy annular flow pattern

Conclusions

Numerical studies are conducted to estimate frictional pressure loss and to determine flow patterns of two phase liquid- gas flow through horizontal eccentric annuli by using Eulerian- Eulerian computational fluid dynamics (CFD) model. Experimental data were used in order to test the CFD model. Results show that CFD model is capable of predicting pressure loss of six flow pattern including dispersed bubble, dispersed annular, plug, slug, transition zone and wavy annular. It can be observed that CFD predictions of gas-liquid pressure loss showed reasonable agreement with the experimental results. Moreover, CFD model can predict flow patterns correctly when compared with experimental data.

REFERENCES

[1] Hasan A.R, Kabir C.S.,1988.'' Predicting Multiphase Flow Behavior in a Deviated Well'', SPE Production Engineering, (November 1988), pp. 474-482.

[2] Beggs H.D., Brill J.P., 1973. '' A Study of Two-Phase Flow in Inclined Pipes'', JCT, (May 1973), pp. 607-617.

[3] Xiao, J.J., Shoham, O., Brill, J.P., 1990."A Comprehensive Mechanistic Model for Two-Phase Flow in pipelines" SPE paper 20631. 1990 SPE Annual Technical Conference and Exhibition, New Orleans, 23-26 September.

[4]Chokshi, R.J., Schmidt Z., and Doty D.R.,1996.'' Experimental Study and the Development of a Mechanistic Model for Two-Phase Flow Through Vertical Tubing'', SPE paper 35676. 1996 Western Regional Meeting, Alaska, 22-24 May.

[5]Das G., Das P.K., Purohit N.K., and Mitra A.K.,1999.'' Flow Pattern Transition During Gas Liquid Upflow Through Vertical Concentric Annuli- Part I: Experimental Investigations'', Journal of Fluids Engineering, 121: 895-901.

[6]Das G., Das P.K., Purohit N.K., and Mitra A.K.,1999.'' Flow Pattern Transition During Gas Liquid Upflow Through Vertical Concentric Annuli- Part II: Mechanistic Models'', Journal of Fluids Engineering, 121: 902-907.

[7]Sunthankar A.A., Miska S., Kuru E., and Kamp A., 2000.''New Developments in Aerated Mud Hydraulics for Horizontal Well Drilling'', SPE paper 62897. 2000 SPE Annular Technical Conference and Exhibition, Dallas, Texas, 1-4 October 2000.

[8]Zhou L., Ahmed R.M., Takach N.E., Yu M., and Pickell M.B., 2004.'' Experimental Study of Aerated Mud Flows under Horizontal Borehole Conditions'', SPE paper 89531. SPE/ICoTA Coiled Tubing Conference and Exhibition, Houston, Texas, USA., 23-24 March.

[9]Zhang H.Q., Wang Q., Sarica C., and Brill J.P., 2004.''Unified Model of Heat Transfer in Gas-Liquid Pipe Flow'', SPE paper 90459. SPE Annular Technical Conference and Exhibition, Houston, Texas, USA., 26-29 September.

[10]Julia J.E., Ozar B., Dixit A., Jeong J.J., Hibiki T., and Ishii M., 2009.''Axial Development of Flow Regime in Adiabatic Upward Two-Phase Flow in a Vertical Annulus'', Journal of Fluids Engineering, 131: 021302 (1-11).

[11]Fisher, R.A., 1936.'' The Use of Multiple Measurements in Taxonomix Problems'', Annals of Eugenics, 7: 179-188.

[12]Osgouei R.E., 2010." Determination of Cutting Transport Properties of Gasified Drilling Fluids'', PhD dissertation, METU, Ankara, Turkey.

[13] ANSYS CFX-Solver Theory Guide (2009).