HEFAT2008
6th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
30 June to 2 July 2008
Pretoria, South Africa
Paper number: KW1

COMPUTATIONAL STUDY OF STRATIFIED TWO PHASE OIL/WATER FLOW IN HORIZONTAL PIPES

W.A.S. Kumara, G. Elseth, B.M. Halvorsen and M.C. Melaaen*
*Author for correspondence
Telemark University College and Tel-Tek,
PO.Box 203, N-3901 Porsgrunn,
Norway,
E-mail: Morten.C.Melaaen@hit.no

ABSTRACT

Stratified oil/water two-phase flow in a horizontal tube is numerically simulated using commercial CFD package FLUENT 6.3. The simulations are based on Volume of Fluid (VOF) model. It solves a single momentum equation shared by the fluids, and the volume fraction of each of the fluid in each computational cell is tracked throughout the domain. The RNG k-ε model together with standard wall treatment as the near-wall modelling method is used for turbulence modelling. The effects of surface tension along the interface between two fluids are calculated using Continuum Surface Force (CSF) model. The simulation is performed in a time-dependent way so that the numerical stabilization could be achieved. The final solution which corresponds to steady-state flow is analyzed. Results of pressure drop, slip ratio, interface height and the axial velocity profiles are verified by experimental data. The predictions of pressure drop, slip ratio and interface height are observed to compare favourably with experimental measurements, and estimated flow quantities such as axial velocity profiles are also satisfactory.

INTRODUCTION

The simultaneous flow of oil and water in pipelines is a common occurrence in the petroleum industry. Increased offshore oil and gas exploration and production have resulted in transportation of well fluids in pipelines over relatively long distance. Often, the fluid delivered by the well contains water, which is already present within the stratum. Water fractions often increase during the producing life of a well. The well might be economical to operate even for water cuts as high as 90% [1]. The presence of water must be properly accounted in designing and predicting the flow behavior in both wells and pipelines. Numerous experimental studies have been published in recent years in oil/water flow through pipes [1-7]. But very few references are found in literature related to numerical studies of oil/water flow systems [3]. In the present paper, Computational Fluid Dynamics (CFD) is applied to predict flow behaviour of stratified oil/water flows in horizontal pipes.

NOMENCLATURE

A	[m^2]	Flow cross-sectional area
$C_{1\varepsilon}$,$C_{2\varepsilon}$,	[-]	Coefficient in equation (11)
C_μ	[-]	Coefficient in equation (12)
F	[N/m^3]	Surface tension term
g	[m/s^2]	Gravitational acceleration
G	[m/s^3]	Generation of turbulence kinetic energy
k	[m^2/s^2]	Turbulence kinetic energy
n	[-]	Unit normal vector to a surface
p	[N/m^2]	Pressure
s	[-]	Slip ratio
S	[kg/m^{3}s]	Source term in equation (1)
t	[s]	Time
v	[m/s]	Velocity vector
U	[m/s]	Superficial velocity
x,y,z	[m]	Cartesian axis directions
Special characters		
α	[-]	Volume fraction of phases
ε	[m^2/s^3]	Turbulence dissipation
μ	[kg/ms]	Dynamic viscosity
κ	[1/m]	Curvature
ρ	[kg/m^3]	Density
σ	[N/m]	Surface tension coefficient
σ_k, σ_ε	[-]	Turbulent Prandtl numbers for k and ε
Subscripts		
O		Oil phase
M		Mixture
q		Phase q
SO		Superficial for oil phase
SW		Superficial for water phase
t		Turbulence
W		Water phase

In early effort to understand and model oil/water flow and predict the influence on pressure gradient when water is introduced into an oil pipeline, Russel and Charles (1959) carried out an analysis for the case when both oil and water flow are laminar [7]. They developed a numerical procedure, in order to model the laminar stratified oil/water flow in a circular pipe. Extending this analysis, Charles and Lillelehet (1969) used the similarity method developed by Lockhart and Martinelli (1949) for gas/liquid flow, to present pressure gradient data in the stratified flow of two liquids when one was in laminar and the other in turbulent flow [7]. Arirachakaran et al. (1983) developed a model for stratified oil/water flow based on no-slip between phases [9-10]. This no-slip model is expected to give acceptable pressure drop predictions when close to no-slip conditions apply such as for low viscosity oils and intermediate water cut (0.30-0.70). For higher oil viscosities the slip between the oil and water is generally higher and the deviation increases as observed by Herm Stapelberg and Mewes [11].

The mathematical model developed by Taitel and Dukler [12] for gas/liquid flows was used to calculate the pressure drop of oil/water flows by Kurban et al. [13]. Oil and water were represented as two separate regions and empirical correlations were used for the wall and the interfacial shear stresses. The two fluid model developed for gas/liquid flow has been extended to predict oil/water flows [14]. The validity of the model and its practical significance for analyzing stratified flows were evaluated in view of experimental data of the in situ flow configuration and the associated pressure drop in an oil/water system reported by Valle and Kvandal [8]. Finally, the accuracy of the two-fluid model for oil/water flows was evaluated by comparing its predictions for laminar flows with the results of the exact solution of the Navier–Stokes equations for laminar-stratified flows with curved interfaces [14].

The existing mechanistic models for stratified oil/water flow have been developed based on the interpretation of the dominant physical mechanisms of the flow process. For the lack of knowledge about the distribution of wall and interface shear in stratified pipe flows, tuned or empirical or semi-empirical relationships are often used, with a resulting loss in computational accuracy. Therefore Computational Fluid Dynamics (CFD) techniques have been applied to model stratified oil/water flow. Rhyne proposed a mechanistic multi-layer model based commercial CFD code, CFX by AEA Technology, and data from oil/water flow project at the NSF I/UCRC, Corrosion in Multi-phase Systems Center [15]. However, the multi-layer model needs detailed experimental data for the implementation.

In the present paper, stratified oil/water two-phase flow in a horizontal pipe is numerically simulated using the Volume of Fluid (VOF) model. The RNG k-ε model together with standard wall treatment as the near-wall modelling method is used for turbulence modelling. The Continuum Surface Force (CSF) model proposed by Brackbill et al. [16] is used to include the effect of surface tension. The simulation is performed in a time-dependent way and the final solution which corresponds to steady-state flow is analyzed. The predictions of pressure drop, slip ratio, interface height and the axial velocity profiles are verified by experimental data.

MODELING METHOD

The commercial CFD software package, FLUENT 6.3, which is based on the finite volume approach [17], was used for solving the set of governing equations. The discretized equations, along with the initial and boundary conditions, were solved using the segregated solution method to obtain a numerical solution. Using the segregated solver, the conservation of mass and momentum were solved iteratively and a pressure-correction equation was used to ensure the conservation of momentum and conservation of mass. The RNG *k–ε* model together with standard wall treatment as the near-wall modelling method was used to treat turbulence phenomena in both phases.

A sketch of the geometry of the computational domain for oil/water stratified flow is given in Figure 1.

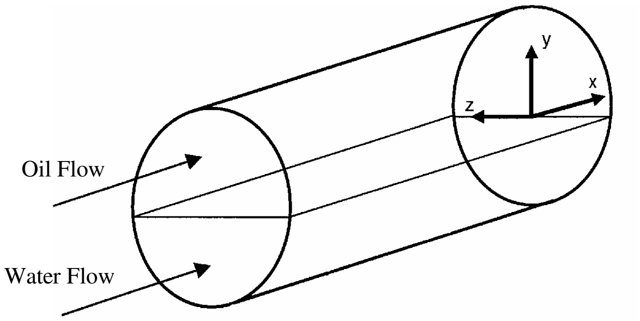


Figure 1 Schematic representation of stratified oil/water flow

The VOF model is a surface-tracking technique applied to a fixed Eulerian mesh. In this model, fields for all variables and properties are shared by the phases and represent volume average values. A single set of momentum equation is solved and the volume fraction of all the fluids in each computational cell is tracked throughout the domain [17]. This is accomplished by the solution of a continuity equation for the volume fraction of one of the phases. For the qth phase, the equation has the following form [17]:

$$\frac{\partial \alpha_q}{\partial t} + \vec{v} \cdot \nabla \alpha_q = \frac{S_{\alpha_q}}{\rho_q} \qquad (1)$$

The source term on the right-hand side of the above equation is assumed to be negligible for the case of stratified oil/water flow without interfacial mass transfer. The volume fraction equation will not be solved for the primary phase and it is computed based on the following constraint:

$$\sum_{q=1}^{n} \alpha_q = 1 \qquad (2)$$

The single momentum equation shown below is solved throughout the domain. The momentum equation is dependent on the volume fractions of two phases through the properties ρ and μ as:

$$\frac{\partial}{\partial t}\left(\rho \vec{v}\right) + \nabla \cdot \left(\rho \vec{v}\vec{v}\right) = -\nabla p + \nabla \cdot \left[\mu\left(\nabla \vec{v} + \nabla \vec{v}^{T}\right)\right] + \rho \vec{g} + \vec{F} \qquad (3)$$

The volume fraction averaged density and viscosity take on the following form:

$$\rho = \sum \alpha_q \rho_q \qquad (4)$$

$$\mu=\sum\alpha_q\mu_q \tag{5}$$

The last term in equation (3), $\vec{F}$, is the external force per unit volume and can be modeled using the Continuum Surface Force (CSF) model developed by Brackbill et al. [16]. An interface is interpolated as a transient region with a finite thickness. Thus the surface tension localized in the region is converted into a volume force with the help of a Dirac delta function concentrated in the interface as:

$$\vec{F}=2\sigma\kappa\alpha_q\nabla\alpha_q \tag{6}$$

The curvature κ is computed from local gradients in the surface normal at the interface. Let n be the surface normal vector, defined as the gradient of α_q , the volume fraction of the qth phase:

$$n=\nabla\alpha_q \tag{7}$$

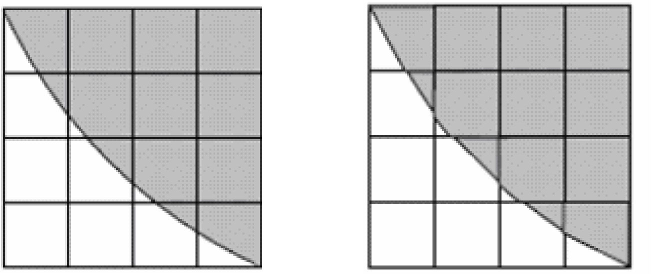
The curvature κ is defined in terms of the divergence of the unit normal, $\hat{n}$ [17]:

$$\kappa=\nabla\cdot\hat{n} \tag{8}$$

Where,

$$\hat{n}=\frac{n}{|n|} \tag{9}$$

In order to calculate convection and diffusion fluxes through the control volume faces, geometric reconstruction scheme is applied for the interface between fluids using a piecewise linear approach. It assumes that the interface between two fluids is a linear slope within each cell, for calculating the advection of fluid through the cell faces. Firstly, the position of the linear interface relative to the center of each partially filled cell is calculated, based on information about the volume fraction and its derivatives in the cell. Then, the advecting amount of fluid through each face is obtained using the computed linear interface representation and information about the normal and tangential velocity distribution on the face. Finally, the volume fraction in each cell is given using the balance of fluxes calculated during the previous step. As shown in Figure 2, the reconstruction of the interface is accomplished via the use of the geometric reconstruction scheme.



(a) (b)

Figure 2 Sketch of the interface calculation: (a) Actual interface shape and (b) Interface shape represented by VOF the geometric reconstruction (piecewise-linear) scheme

The RNG k-ε model together with standard wall treatment as the near-wall modelling method is used for turbulence modelling. The RNG-based $k-\varepsilon$ turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called "renormalization group" (RNG) methods. It is similar in form to the standard $k-\varepsilon$ model, but additional terms and functions are included in the transport equations for k and ε . A more comprehensive description of RNG theory and its application to turbulence can be found in [17]. For the present system, the governing equations are:

Turbulent kinetic energy:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon \tag{10}$$

Dissipation rate of turbulent kinetic energy:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}G_k-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{11}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. The quantities σ_k and σ_ε are the turbulent Prandtl numbers for k and ε respectively. The turbulent kinetic energy and its dissipation rate are coupled to the governing equations via the relation:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{12}$$

The empirical constants for the turbulence model are assigned the following:

C_μ =0.0845, $C_{1\varepsilon}$ = 1.42, $C_{2\varepsilon}$ = 1.68, σ_k =1.0, σ_ε = 1.3

The standard wall function proposed by Launder and Spalding is used for modelling near wall flow in both phases [17].

At the inlet, uniform profiles for all the dependent variables are employed. The specified inlet conditions are assumed constant over the cross-sectional area. The axial velocity is calculated using specified mixture velocity and water cut. The velocities in the other coordinate directions are assumed to be zero. The turbulent kinetic energy and its dissipation rate are calculated from a turbulence intensity of 5 % and corresponding hydraulic diameter of each phase. The non-slip boundary condition is imposed on the wall of the pipe. The outlet boundary condition was set up as a pressure outlet with a constant outlet pressure. The gradients for the entire variables in exit direction were set to be zero.

NUMERICAL METHOD

The governing equations were solved with the finite volume method and the commercial CFD code FLUENT 6.3 is used as the numerical solver. The PISO algorithm is used to resolve the coupling between velocity and pressure. The more accurate second-order upstream advection scheme is applied to momentum, turbulent kinetic energy and turbulent dissipation rate equations. The PRESTO scheme is used for pressure discretization. These schemes improve the accuracy and the convergence of the solution. The convergence criterion is based on the residual value of the calculated variables, i.e., mass, velocity components, turbulent quantities. In the present calculations, the threshold residual value for all the variables were set to 10^{-3}.

The stratified oil/water two-phase turbulent flow in a 55.75 mm diameter, 5 m long horizontal tube is numerically simulated. An unstructured non-uniform grid system was used to discretize the governing equations. Figure 3 illustrates the grid topology used on one cross-section. A boundary layer mesh is used close to the wall. The three dimensional mesh, used for the computation consists of 214,956 control volumes. A fine grid is used close to the inlet of the pipe, where the flow field is developing. The length of the control volumes in axial direction is gradually increased towards the end of the pipe, where the flow field is fully developed giving negligible changes of flow properties along the pipe.

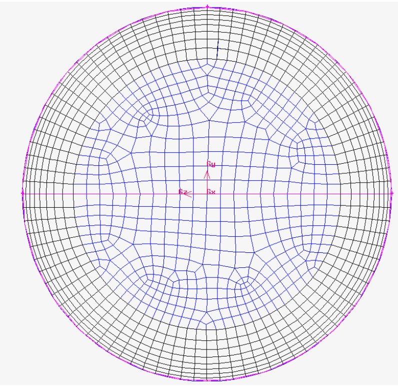


Figure 3 Grid topology on pipe cross-section

EXPERIMENTAL DETAILS

The experimental activities have been performed in the multiphase flow loop at Telemark University College, Porsgrunn, Norway [1]. The multiphase flow loop consists of a 14m long pipe with an inner diameter of 55.75 mm. Water (density 998 kg/m^3, viscosity 1 mPa s) and Exxsol D60 oil (density 790 kg/m^3, viscosity 1.6 mPa s) were used as test fluids. The experiments have been performed at different mixture velocities and water cuts. The instantaneous local velocities were measured using Laser Doppler Anemometry (LDA), and time average cross-sectional distributions of oil and water were measured with a traversable gamma densitometer. The pressure drop along the test section of the pipe was also measured. Some of the experimental flow cases are simulated as listed in Table 1.

RESULTS AND ANALYSIS

Predictions of axial pressure gradient from the present model are compared with the experimental data in Figure 4. In general, the agreement is acceptable, given the uncertainties in the experimental measurements and simulations. The predicted results agree well with the experimental data when the water cut is in the range 0.15-0.75. The experimental result shows a maximum pressure drop at water cut 0.85. This is probably due to the dispersion effect in the oil phase [2]. But the model gives a significant under-prediction of 18.4 % when water cut is 0.85. It may be due to inherent limitation of the present model in predicting dispersed flow conditions. The VOF model that solves a single momentum equation gives better results when the flow is stratified. When the water cut is 0.85, a significant difference between the predicted and experimental axial velocity profiles was observed and it can be the reason for the deviation of the pressure drop prediction.

Water Cut [-]	U_M	U_{SO}	U_{SW}
0.15	1.07	0.91	0.16
0.25	1.07	0.81	0.26
0.40	1.06	0.64	0.42
0.50	1.06	0.53	0.53
0.60	1.06	0.42	0.64
0.75	1.07	0.26	0.81
0.85	1.05	0.13	0.92

Table1 Experimental flow cases simulated

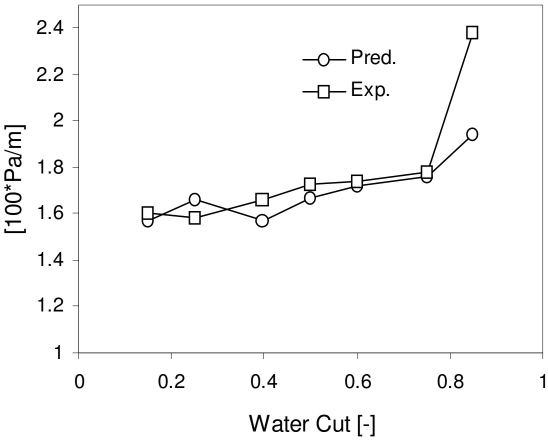


Figure 4 Comparison between predicted and experimental results of axial pressure gradient

The predicted interface height is compared with the experimental data presented by Elseth et al. [2] in Figure 5. The vertical distance from the bottom of the pipe to the point where the water cut is 0.5 is considered as the interface height. The agreement is quite favourable. However, the model under-predicts the experimental data with an absolute average error of 3.3 %. The deviation is higher at lower and higher water cuts. The error is 8.0 % and 3.4 % when the water cut is 0.15 and 0.85, respectively. This can be attributed to the inherent limitation of the present model in predicting dispersed flow. At low water cuts a significant part of the water layer is dispersed in oil layer and the flow regime can be categorized as oil continuous dispersed flow [1]. This flow situation can not be handled accurately with the present model based on VOF approach, which is designed for stratified flow. Therefore a deviation of the interface height predictions can be observed at low water cuts. At intermediate water cuts, the flow was stratified with weak mixing at the interface. Few droplets were observed close to the interface [1]. In this case the flow regime can be considered as stratified flow. Therefore the model gives better predictions at intermediate water cuts.

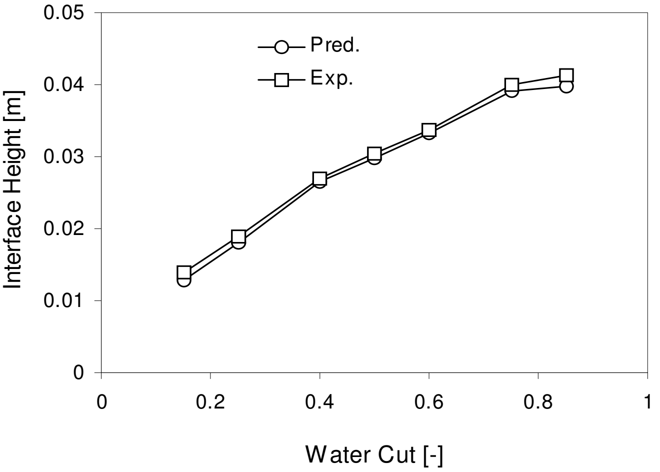


Figure 5 Comparison between predicted and experimental results of interface height

Finally at higher water cuts, most of oil is dispersed in water layer and water continuous dispersed flow is created. A large number of oil droplets are present at the interface and a thick interface region has been observed experimentally [1]. Again the present model fails to give better predictions for both interface height and pressure drop as shown in Figure 5 and Figure 4, respectively. The model gives more accurate results for both pressure drop and interface height at intermediate water cuts 0.25-0.75.

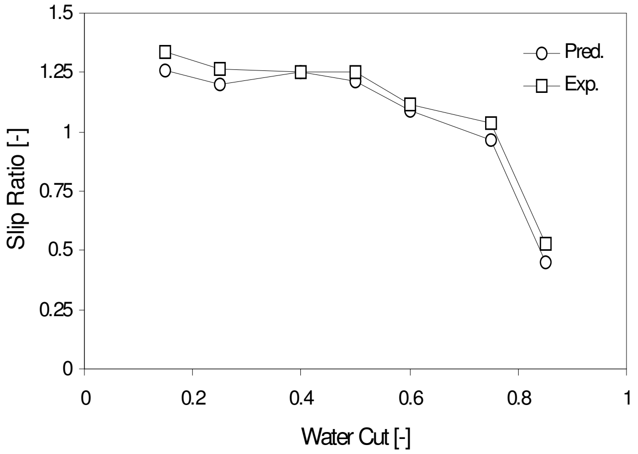


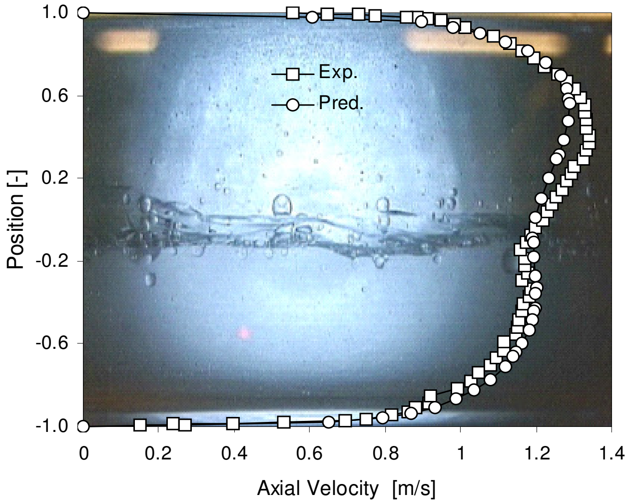
Figure 6 Comparison between predicted and experimental results of slip ratio

Figure 6 shows a comparison between the predicted slip ratio and experimental data of Elseth et al. [2]. The slip ratio s, is calculated by:

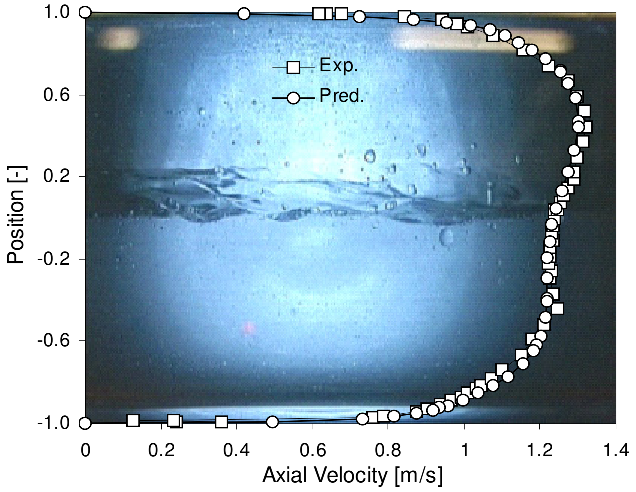
$$s = \frac{A_W}{A_O}\frac{U_{SO}}{U_{SW}} \tag{13}$$

Where A_W and A_O is the flow area of water and oil, respectively. U_{SO} is the superficial velocity of oil and U_{SW} is the superficial velocity of water. The predicted flow area was calculated assuming a flat interface. As shown in Figure 6 the predicted slip ratio is observed to compare favourably with experimental measurements. However the model slightly under-predicted the slip ratio with an average absolute error of 5.5 %. The deviation is higher at lower and higher water cuts compared to intermediate water cuts. The slip ratio is mainly influenced by flow area occupied by oil and water when the same superficial velocities are used for both experiments and simulations. Therefore slight deviation in interface prediction, gives an error in predicted slip ratio due to inaccuracy of calculating the flow area occupied by oil and water. This can be the reason for the under-predicted slip ratio at lower and higher water cuts.

The predicted axial velocity profiles are compared with Laser Doppler Anemometry (LDA) measurements presented by Elseth [1] in Figure 7, where agreement is seen to be quite reasonable. In Figure 7(a) and (b) still pictures of the flow are used as the background in order to visualise the flow. Figure 7(a) shows the axial velocity profiles when the mixture velocity and water cut are 1.06 and 0.40, respectively. As shown in the image of Figure 7(a) the flow is stratified with very weak mixing at the interface. Few droplets were formed by break-up of the interfacial waves [1]. Therefore the present model based on VOF approach is used to predict the flow phenomena. Both experimental and the simulated results shows the maximum velocity in oil phase as expected. However the maximum velocity in oil phase is slightly under-predicted and also the position of the peak velocity is located slightly above compared to the experimental data. The velocity in water phase is slightly over-predicted but closely follows the experimental results. The complex effects of interfacial waves and droplets on interfacial friction and thereby pressure and velocity fields can not be predicted accurately with the present model. This can be the reason for the deviations in velocity profile. At water cut 0.50 the oil/water flow is stratified with only small waves at the interface. Mixing at the interface is very weak and few droplets were observed close to the interface [1]. Therefore the predicted axial velocity profile agrees very well with the experimental data as shown in Figure 7(b). It is notable that the model can predict the position and the magnitude of the peak velocities in both phases. However the expectations based upon classic laminar flow theory suggest higher velocity in the less viscous phase at equal volumetric flows as in this case with water cut 0.50. But both experimental and predicted results show an opposite effect giving maximum velocity in oil phase. The reason for this effect is not obvious. At low flow velocities (laminar flow) the interfacial friction is governed by the viscosities of different phases and therefore a higher mean velocity can be expected in less viscous fluid in order to maintain the continuity of the interfacial shear. However, for turbulent flows both viscosity and density influence the interfacial friction and therefore the position of the maximum velocity can not be predicted using classic laminar flow theory. At water cut 0.50 the Reynolds numbers for oil and water phases are 17,893 and 35,234, respectively. Therefore the flow is fully turbulent and the maximum velocity is located in oil phase.



(a) Mixture velocity 1.06 m/s and water cut 0.40



(b) Mixture velocity 1.06 m/s and water cut 0.50

Figure 7 Comparison of predicted and experimental axial velocity

CONCLUSION

Stratified oil/water two-phase flow in a horizontal tube is numerically simulated using commercial CFD package FLUENT 6.3. The simulations are based on Volume of Fluid (VOF) model. The RNG k-ε model together with standard wall treatment as the near-wall modelling method is used for turbulence modelling. The predictions of pressure drop, slip ratio and interface height are observed to compare favourably with experimental measurements. The predictions of the velocity profiles are quite satisfactory.

The interfacial friction and the position of the interface have found to have a profound effect on flow predictions. The present model based on VOF approach has inherent deficiencies in predicting dispersed flow conditions at low and high water cuts. The model gives better predictions at intermediate water cuts 0.25-0.75. Although the present formulation is rather complex and demands much computation time, due to the nature of the multiphase turbulent flow and the fine grid required for its implementation, it does appear to demonstrate that the CFD technique can be successfully applied to predict oil/water stratified flow in pipes.

REFERENCES

[1] Elseth G., An Experimental Study of Oil/Water Flow in Horizontal Pipes, *PhD Thesis, The Norwegian University of Science and Technology (NTNU),* 2002.

[2] Elseth G., Kvandal H.K., and Melaaen M.C., Measurement of velocity and phase fraction in stratified oil/water flow, *International Symposium on Multiphase Flow and Transport Phenomena, Antalya, Turkey,* 2000, pp. 206–210.

[3] Lovick J. and Angeli P., Experimental studies on the dual continuous flow pattern in oil–water flows, *International Journal of Multiphase Flow,* 30, 2004, pp. 139–157

[4] Lovick J. and Angeli P., Droplet size and velocity profiles in liquid–liquid horizontal flows, *Chemical Engineering Science,* 59, 2004, pp. 3105 – 3115.

[5] Angeli P. and Hewitt G.F., Pressure gradient in horizontal liquid-liquid flows, *International Journal of Multiphase Flow,* 24, 1998, pp. 1183 – 1203

[6] Rodriguez O.M.H. and Oliemans R.V.A, Experimental study on oil–water flow in horizontal and slightly inclined pipes, *International Journal of Multiphase Flow*, 32, 2006, pp. 323–343.

[7] Valle A., Multiphase pipeline flows in hydrocarbon recovery, *Multiphase Science and Technology,* vol. 10 No.1, 1998, pp. 1–139.

[8] Valle A. and Kvandal H., Pressure drop and dispersions characteristics of separated oil/water flow, *Proceedings of the 1st Int. Symp. On Two-Phase Flow Modeling and Experimentation, Rome, Italy,* 1995.

[9] Arirachakaran, S., An experimental study of two-phase oil–water flow in horizontal pipes, *MS Thesis, U. of Tulsa*, 1983.

[10] Arirachakarn, S., Oglesby, K.D. and Brill, J.P., An Analysis of Oil/ Water Flow Phenomena in Horizontal Pipes, *SPE*, 1989. pp. 18836.

[11] Herm Stapelberg H. and Mewes D., Three phase flow (oil, water and gas) in horizontal tubes, *Proceeding of The International Conference on Multiphase Flows, 91-Tsukuba,* 1994, pp. 349-352.

[12] Taitel, Y. and Dukler, A.E., A model for predicting flow regime transitions in horizontal and near horizontal gas liquid flow. *AIChE* J. 22, 1976, pp. 47–55.

[13] Kurban, A.P.A. and Angeli, P., Stratified and dispersed oil–water flows in horizontal pipes. *BHRA95*, 1995, pp. 277–291.

[14] Brauner N., Moalem Maron D. and Rovinsky J., A two-fluid model for stratified flows with curved interfaces. *Int. J. Multiph. Flow 24 (6),* 1998, pp. 975–1004.

[15] Rhyne L., Oil–water mechanistic model. Advisory Board Meeting, *NSF I/UCRC Corrosion in Multiphase Systems Center,* April, 2000.

[16] Brackbill J.U., Kothe D.B. and Zemach C., A continuum method for modelling surface tension, *J. Comp. Phys*., 100, 1992, pp. 335–354.

[17] Fluent, Fluent 6.2 Users Guide, *Fluent Inc., Lebanon, NH, USA,* 2005.