Oil-Water Two-Phase Flow Characteristics in Horizontal Pipeline – A Comprehensive CFD Study

Anand B. Desamala, Ashok Kumar Dasamahapatra, Tapas K. Mandal

Abstract—In the present work, detailed analysis on flow characteristics of a pair of immiscible liquids through horizontal pipeline is simulated by using ANSYS FLUENT 6.2. Moderately viscous oil and water (viscosity ratio = 107, density ratio = 0.89 and interfacial tension = 0.024 N/m) have been taken as system fluids for the study. Volume of Fluid (VOF) method has been employed by assuming unsteady flow, immiscible liquid pair, constant liquid properties, and co-axial flow. Meshing has been done using GAMBIT. Quadrilateral mesh type has been chosen to account for the surface tension effect more accurately. From the grid independent study, we have selected 47037 number of mesh elements for the entire geometry. Simulation successfully predicts slug, stratified wavy, stratified mixed and annular flow, except dispersion of oil in water, and dispersion of water in oil. Simulation results are validated with horizontal literature data and good conformity is observed. Subsequently, we have simulated the hydrodynamics (viz., velocity profile, area average pressure across a cross section and volume fraction profile along the radius) of stratified wavy and annular flow at different phase velocities. The simulation results show that in the annular flow, total pressure of the mixture decreases with increase in oil velocity due to the fact that pipe cross section is completely wetted with water. Simulated oil volume fraction shows maximum at the centre in core annular flow, whereas, in stratified flow, maximum value appears at upper side of the pipeline. These results are in accord with the actual flow configuration. Our findings could be useful in designing pipeline for transportation of crude oil.

Keywords—CFD, Horizontal pipeline, Oil-water flow, VOF technique.

I. INTRODUCTION

TWO-PHASE flow is an important phenomena appearing in many industrial applications such as chemical plants and nuclear power generation. However, to understand the phenomena is still difficult because of complex phenomena underlying its behavior. The design parameters such as pressure drop, holdup and flow regimes in a single phase flow in conduits can be modeled easily. However the existence of secondary phase such as water can lead to increase the complexity in hydrodynamics and creates different challenges in modeling the system. Knowledge of the hydrodynamics of such two-phase flow (liquid-liquid) is essential for the design of extractor, mixture-settlers, transportation pipeline, pipeline networks, downstream separators, emulsifier etc. Over last ten years, Computational Fluid Dynamics (CFD) has become industrial simulation tool for an engineering system

Anand B. Desamala, Ashok Kumar Dasamahapatra, Tapas K. Mandal are with the Department of Chemical Engineering, IIT Guwahati, Guwahati-781039, India (phone: +91 9678425465; e-mail: a.desamala@iitg.ernet.in).

investigation which includes fluid flow, design, performance determination and analysis. This improvement has been made due to easy accessibility and enormous increase in computer memory capacity and speed, resulting in a reduction in costs of simulation compared to experimental work.

Many authors have used CFD simulation tool to estimate the hydrodynamics of two phase flow. For example, Huang et al. [1] have used k-ε turbulence model to study the effect of eccentricity on friction factor and holdup for both laminar and turbulent cases. They have noticed that, the friction factor increases with eccentricity. Ko et al. [2] have found that, the model prediction of pressure distribution and wave length is better than the K-ω turbulence model. They have used shear stress transport model to solve the kinetic energy and dissipation equations of turbulent wavy core flow. Ghosh et al. [3] have simulated the core annular down flow using Eulerian–Eulerian based VOF technique in FLUENT. Al-yaari et al. [4] have simulated stratified flow in horizontal pipeline using RNG k-ε turbulence model. Recently, Kaushik et al. [5] have simulated the core annular flow through sudden contraction and expansion using VOF technique. They have validated their simulation results with the experimental results and satisfactory match is observed.

In the present study, computational fluid dynamics (CFD) simulation has been executed to investigate volume fraction profile, area average pressure across a cross section and velocity profile along the radius of different flow patterns of viscous oil-water two-phase flow through a horizontal pipeline. Volume of Fluid (VOF) approach including effect of surface tension has been tried to adopt for prediction of flow regimes.

II. MODEL DEVELOPMENT

The flow domain was constructed and meshed in GAMBIT^{TM} and the solver chosen was ANSYS FLUENT^{TM}. The geometry and detailed dimensions of the concerned conduit used in the present work is shown in Fig. 1 (a). The geometry consists of a horizontal test section whose internal diameter is 0.025 m and material of construction used for the pipe was Perspex. Oil and water were introduced into the pipe through a T-junction at the entry section where water and oil enter into the pipe from the horizontal and vertical directions, respectively. The geometry of the system can be assumed to consist of four sections -water inlet, oil inlet, outlet of the pipe and the test section. An immiscible pair of liquids is chosen with coaxial flow assumed. An unsteady state solver has been employed for the computation purpose. Further, the

World Academy of Science, Engineering and Technology
International Journal of Chemical and Molecular Engineering
Vol:8, No:4, 2014

simulations were run in 2-Dimensional mode. Fig. 1 (b) represents the meshed geometry. From the mesh independent study, system with 47037 cells is selected as optimum mesh

elements and this number is used throughout the simulation. Quadrilateral mesh geometry is selected for accounting the surface tension effect more accurately.
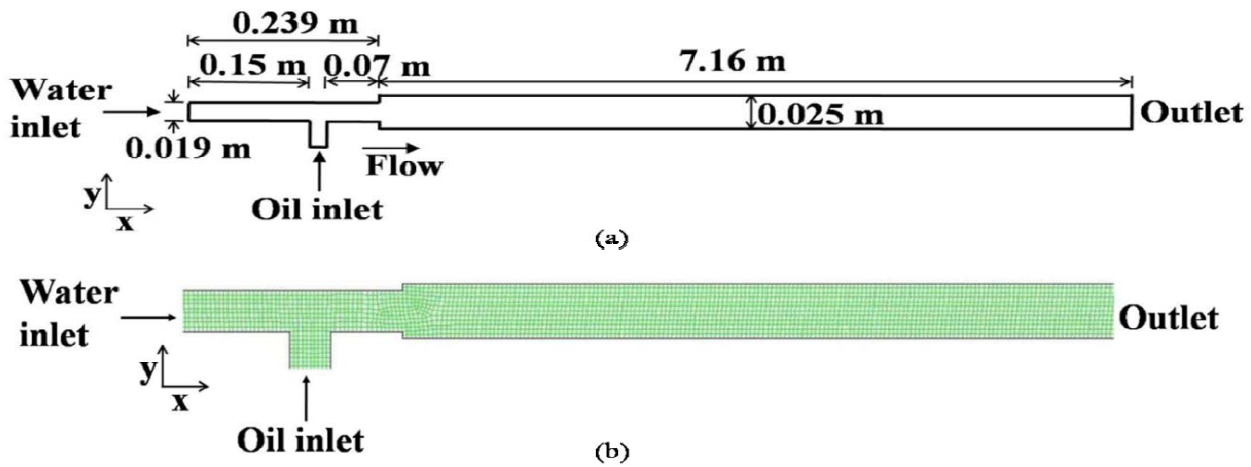
Fig. 1 Geometry of horizontal pipe (a) Detailed dimensions of pipe (b) Meshing of pipe

The computation has been performed by assuming unsteady flow, immiscible liquid pair, constant liquid properties, co-axial flow in the pipe and a Tee junction ('T') as the entry section. In the present model, the two fluids share a well defined interface. Volume of Fluid (VOF) approach for two phase modeling has been selected in Fluent. VOF solves a single set of momentum equations which is shared by both the fluids. We briefly discuss the governing equations and the treatment of the interface in the next section, details can be obtained from Fluent user's guide [6].

A. Governing Equations

In VOF approach, the continuity equation can, therefore be written as:

$$\frac{\partial(\rho)}{\partial t} + \nabla \cdot (\rho U) = \sum_q S_q \tag{1}$$

where, ρ, U, t, S are density, velocity, time and mass source respectively. In the present case 'S' is zero.

The equation of momentum conservation can be expressed as follows:

$$\frac{\partial(\rho U)}{\partial t} + \nabla \cdot (\rho U \cdot U) = -\nabla P + \nabla \cdot [\mu(\nabla U + \nabla U^T)] + (\rho g) + F \tag{2}$$

The left hand side corresponds to convection and the first term on the right hand side corresponds to pressure, while the other terms represent diffusion, gravitational body force and external body force (or source term as surface force) respectively.

Generally, the Volume of Fluid solves the problem of updating the phase volume fraction field, provided the fixed grid, the phase volume fraction, and the velocity field are available in a given time step. If the volume fraction of qth

fluid in a cell is denoted as 'α_q', the following three possibilities would arise:

- $\alpha_q = 0$: the cell does not contain fluid q.
- $\alpha_q = 1$: the cell is occupied solely by fluid q.
- $0 < \alpha_q < 1$: the cell contains the interface

Depending on the local value of α_q, the density and viscosity in each cell are given by:

$$\rho = \sum_1^p \rho_q \alpha_q \tag{3}$$

$$\mu = \sum_1^p \alpha_q \mu_q \tag{4}$$

A separate continuity equation for 'α_q' is considered as follows:

$$\frac{\partial \alpha_q}{\partial t} + (U_q \cdot \Delta) = S_{\alpha q} \tag{5}$$

For each of the cells the following relationship is also valid:

$$\sum_1^p \alpha_q = 1 \tag{6}$$

where "p" is the number of phases. For the present two phase flow, p = 2.

B. Initial and Boundary Condition

In all cases, the flow has been initialized by filling up the pipe with water from the water inlet with a specified inlet superficial velocity. Oil is then introduced in the pipe. The main steps followed during the simulation are:

- 2-D pressure based segregated solver with implicit formulation is selected as solver under unsteady state condition.

World Academy of Science, Engineering and Technology
International Journal of Chemical and Molecular Engineering
Vol:8, No:4, 2014

- Volume of Fluid (VOF) model is selected with number of phases q = 2. Explicit VOF scheme is selected so that the discretization scheme for VOF changes to Geo-Reconstruct (to get the surface tension effect).
- Test fluids are defined using material data base of Fluent and the properties are changed according to the present work.
- The operating pressure is set as atmospheric pressure (default setting), and gravity is considered in Y-direction as -9.81 m/s^2.
- The inlet velocities of both the fluids are assumed to be uniform and specified as follows:
  - At x = 0, y = 0; U_x= U_{water} and U_y = 0 (m/s)
  - At x = 0.15 m and y = -0.0595 m; U_y = U_{oil} and U_x = 0 (m/s)
- The wall is assumed to be stationary and no slip condition is imposed. A contact angle of 8.5^o [7] is taken to account for the wetting behavior of the wall with the fluids.
- Pressure outlet boundary is selected and the diffusion flux variables at the exit are taken as zero.
- Time step size used in all simulations is 0.001 s.

TABLE I
DIFFERENT VELOCITIES USED FOR SIMULATION

Sl.No.	Water velocity (U_{SW}), m/s	Oil velocity (U_{SO}), m/s	Simulation result
1	0.13	0.024	Slug flow (S)
2	0.23	0.1	Stratified wavy flow (SW)
3	0.23	0.24	Stratified mixed flow (SM)
4	0.4	0.4	Annular flow (A)

III. RESULTS AND DISCUSSION

Simulations have been performed at different phase velocities in a 2D-approach using VOF method. VOF method successfully predicts the slug (S), stratified wavy (SW), stratified mixed (SM) and annular flow (A). The velocity ranges and corresponding simulation results (flow pattern) are shown in Table I. These velocities (in Table I) are selected for respective flow regimes of the horizontal flow reported by Anjali et al. [8]. Fig. 2 (a) depicts that, at low phase velocities (U_{SW} = 0.13 m/s and U_{SO} = 0.02 m/s) oil slugs appear with a distinct liquid bridge between two consecutive slugs in continuous water phase. The size of the slugs is comparable with the internal diameter of the pipe. This is known as slug flow (S). In stratified wavy flow (SW), oil and water are separated with a wavy oil/water interface (See Fig. 2 (b)). By increasing oil velocity (U_{SO} = 0.24 m/s) keeping water velocity (U_{SW} = 0.23 m/s) constant, the wave ruptures to form droplets at the interface. This results in the formation of three layers or mixed flow pattern with oil at top, water at bottom and mixed layer comprising of oil and water droplets in the middle. The simulated result at this velocity is shown in Fig. 2 (c). At moderate velocities of oil (U_{SO} = 0.4 m/s) and water (U_{SW} = 0.4 m/s), oil flows at the center of the tube as a continuous core and the water which flows as an annulus is in contact with the pipe wall (Fig. 2 (d)). This flow pattern is known as annular flow (A). These simulation data are validated with the experimental flow pattern map of Anjali et al. [8]. For this the simulation data are superimposed on the experimental flow pattern map of (Fig. 3). The scattered points in Fig. 3 shows different flow patterns reported by Anjali et al. [8] and circled points shows the simulation data of present work. The validation shows good conformity of slug, stratified wavy, stratified mixed, and annular flow.
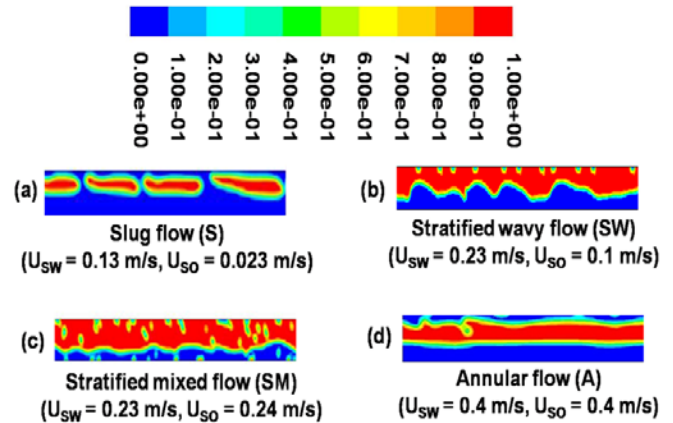

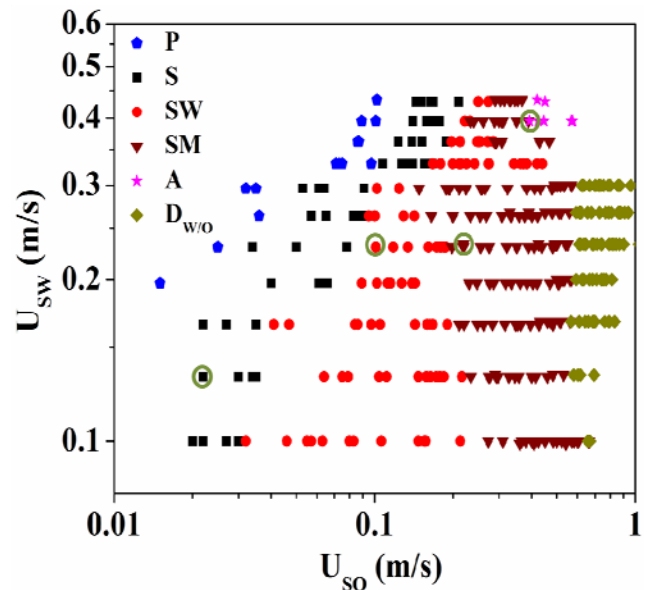Fig. 2 Contours of oil phase at different velocities

Fig. 3 Validation of present work with experimental map of Anjali et al. [8]

IV. FLOW CHARACTERISTICS

We have simulated the hydrodynamics (viz., volume fraction profile, area average pressure across a cross section and velocity profile along the radius) of stratified wavy and annular flow at different phase velocities. The schematic representation of radial distribution of flow is shown in Fig. 4.
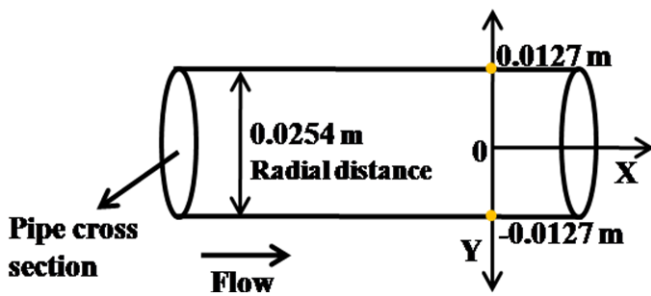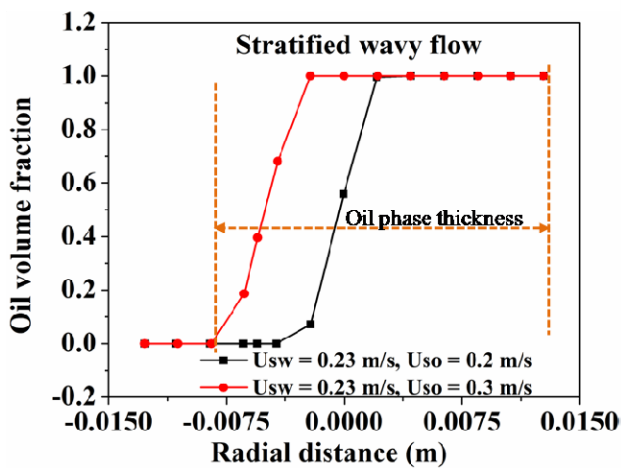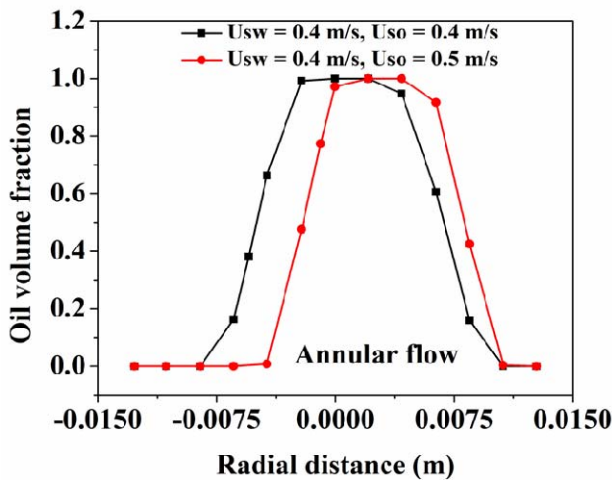
World Academy of Science, Engineering and Technology
International Journal of Chemical and Molecular Engineering
Vol:8, No:4, 2014

Fig. 4 Schematic representation of radial distribution of flow

volume fraction for stratified wavy and annular flow is shown in Figs. 5 (a) and (b) respectively. In Fig. 5 (a), variation of oil fraction by increasing oil velocity (U_{SO} = 0.2 to 0.3 m/s) by keeping water velocity (U_{SW} = 0.23 m/s) constant is shown. During stratified wavy flow, oil phase thickness increases as oil velocity increases. It is clearly depicted in Fig. 5 (a). During the annular flow, the oil phase flows in the center as core and water phase is in complete contact with the pipe wall (Fig. 2 (d)). This is confirmed from the radial distribution of oil volume fraction which is shown in Fig. 5 (b).
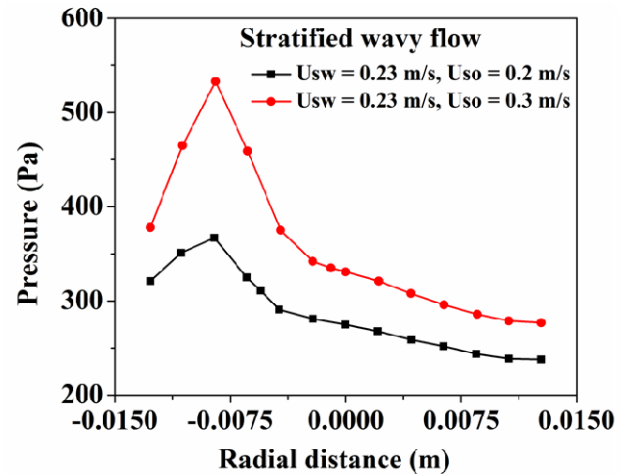

Fig. 5 Radial distribution of oil volume fraction (a) Stratified wavy flow (b) Annular flow

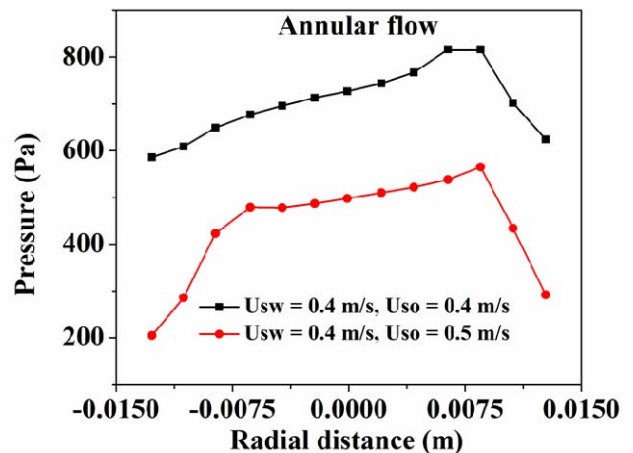*A. Volume Fraction*

It is one of the most important parameters used to characterize two-phase flows. It is the key physical value for determining numerous other important parameters, such as the two-phase density and the two-phase viscosity, for obtaining the relative average velocity of the two phases, and is of fundamental importance in models for predicting flow pattern transitions and pressure drop. The radial behavior of oil

Fig. 6 Radial distribution of pressure (a) Stratified wavy flow (b) Annular flow

B. Pressure

Pressure drop in two-phase flow is a major design parameter, governing the pumping power required to transport two-phase fluids. Variation of pressure (mixture) along the radial direction for stratified wavy and annular flow is shown in Fig. 6. The pressure will be maximum at the oil-water interface due to slip between the phases in stratified flow. Fig. 6 (a) depicts the same showing maximum pressure at interface. Where as in annular flow, there are two oil-water

World Academy of Science, Engineering and Technology
International Journal of Chemical and Molecular Engineering
Vol:8, No:4, 2014

interfaces because oil flows as core in center surrounded by water phase. At these two interfaces pressure will be high compared to the oil phase in center. However, maximum pressure is noticed at upper portion inside the pipe wall. This prediction is clearly shown in Fig. 6 (b).
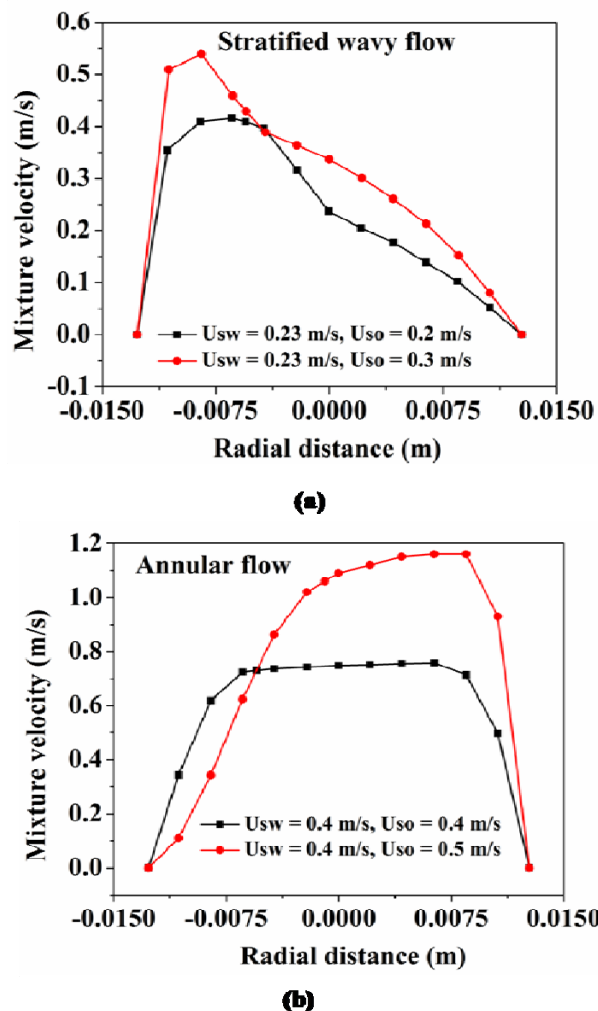

Fig. 7 Radial distribution of velocity (a) Stratified wavy flow (b) Annular flow

C. Velocity Profile

Radial distribution of mixture velocity of stratified wavy and annular flow is shown in Fig. 7. In stratified wavy flow, oil phase is in laminar and water phase is in turbulent flow. Hence the mixture velocity is high for water phase and low for oil phase (see Fig. 7 (a)). In annular flow, the velocity profile looks like a parabola showing maximum velocity at the centre (Which is occupied by oil phase). This is shown in Fig. 7 (b).

V. CONCLUSION

The attempts have been made to show the flow characteristics of oil-water flow in horizontal pipeline. CFD simulation has been done with the help of ANSYS FLUENT^{TM} software. Slug, stratified wavy, stratified mixed

and annular flow has been predicted well using VOF technique. The simulated result is also validated with the experimental results. A good agreement has been observed for these flow patterns. Other flow patterns like dispersion, either oil in water and water in oil have not been predicted yet. The flow characteristics (viz., volume fraction, pressure and velocity profile) of stratified wavy and annular flow are discussed. The simulation results show that in the annular flow, total pressure of the mixture decreases with increase in oil velocity due to the fact that pipe cross section is completely wetted with water. Simulated oil volume fraction shows maximum at the centre in core annular flow, whereas, in stratified flow, maximum value appears at upper side of the pipeline. These results are in accord with the actual flow configuration. Our findings could be useful in designing pipeline for transportation of crude oil.

REFERENCES

[1] A. Huang, C. Christodoulou, and D. D. Joseph, "Friction factor and hold up studies for lubricated pipelining part -2: Laminar and k-ε models of eccentric core flow," Int. J. Multiphase flow. vol. 20, pp. 481–491, 1994.
[2] T. Ko, H. G. Choi, R. Bai, and D. D. Joseph, "Finite element method simulation of turbulent wavy core-annular flows using a k-Є turbulence model method," Int. J. Multiphase Flow., vol. 28, pp. 1205–1222, 2002.
[3] S. Ghosh, G. Das, and P. K. Das, "Simulation of core annular downflow through CFD–A comprehensive study," Chem. Eng. Process., vol. 49, pp. 1222–1228, 2010.
[4] M. A. Al-Yaari, and B. F. Abu-Sharkh, "CFD prediction of stratified oil-water flow in a horizontal pipe," Asian Trans. Eng., vol. 01, pp. 68–75, 2011.
[5] V. V. R. Kaushik, S. Ghosh, G. Das, and P. K. Das, "CFD simulation of core annular flow through sudden contraction and expansion," J. Pet. Sci. Eng., vol. 86-87, pp. 153–164, 2012.
[6] Fluent 6.3 User's Guide, Fluent Inc., Lebanon, USA, 2006.
[7] A. B. Desamala, D. Anjali, V. Vinayak, G. B. Kumar, A. K. Dasmahapatra, and T. K. Mandal, "CFD simulation and validation of flow pattern transition boundaries during moderately viscous oil-water two-phase flow through horizontal pipeline," WASET J., vol. 73, pp. 1150–1155, Jan. 2013.
[8] D. Anjali, A. B. Desamala, A. K. Dasmahapatra, and T. K. Mandal, "Flow pattern of viscous oil-water flow: Prediction by probabilistic neural network (PNN) and validation with experimental data," Published in the proceedings of 64th annual session of the IIChe, 2011, December 26–29, Bangalore, India.