CFD Prediction of Stratified Oil-Water Flow in a Horizontal Pipe

Mohammed A. Al-Yaari* and Basel F. Abu-Sharkh

Department of Chemical Engineering, King Fahd University of Petroleum & Minerals, Saudi Arabia

* Corresponding author; E-mail address: malyari@kfupm.edu.sa

***Abstract*—Oil–water two-phase flow in 0.0254 m horizontal pipe is numerically simulated using FLUENT 6.2. Oil-water stratified flow regime is simulated using Volume of Fluid (VOF) multiphase flow approach. RNG k–ε turbulence model is adopted. Mesh independent study has been achieved to decide on the mesh size. The phase separation is investigated for the tested stratified flow points. The stratified flow pattern results were compared with published experimental results. CFD Numerical simulation predicted the stratified flow pattern and smoothness and the type of the interface. On the other hand, while the oil layer was clearly predicated by the CFD model, water layer was not clearly predicted as a clear segregated layer.**

Index Terms— **Flow patterns, CFD simulation, Oil–water flow, Stratified flow**

I. Introduction

THE co-current flow of immiscible liquid-liquid is encountered in a variety of industrial processes such as petroleum transportation since water and oil are generally produced together. The presence of water, during the transportation of oil from the well to the processing facility, has a significant effect. Transportation is particularly important in offshore oil fields, where they can be exploited by using pipelines of up to 200 km long, to transfer well fluids to an onshore or to an existing platform facility. These pipelines lie on the seabed in a horizontal or near horizontal orientation. Changes in water fraction in the pipe can have a significant influence on power required to pump the fluid due to corresponding changes in pipeline pressure drop.

In general, the introduction of water into oil transportation pipelines can have several effects such as a complex interfacial structure between oil and water which complicates the hydrodynamic prediction of the fluid flow. Water-in-oil or oil–in-water dispersions influence the pressure gradient dramatically. However, increasing the water fraction toward the phase inversion, where the continuous phase becomes water, leads to a high pressure gradient and high power consumption. This can result in a reduction in production capacity. At high water fraction, as the water continuous zone is entered, the pressure gradient decreases again (Soleimani [16]).

Oil production in many parts of the world is characterized by high water cut. This may be attributed to the production practice or the nature of reservoir formation. Excessively high production rates often result in formation water intrusions that ultimately produced with the oil. Also, water produced in the process is frequently re-injected into reservoirs to maintain enough pressure for production enhancement. Thus, more water is produced and the water cut may reach high values ranging between 80-95%. This places an overwhelming economic burden on oil-water separation facilities to cope with the increased oil production and the associated large water production.

Oil-water flow patterns have been investigated by several researchers (Russell et al [14]; Charles et al [8]; Oglesby [12]; Brauner and Maron [5]; Arirachakaran et al [3]; Kurban et al [11]; Theron and Unwin [17]; Angeli [2]; Trallero et al [18]; Soleimani [16]; Shi et al [15]; Al-Yaari et al [1]).

Under certain flow and fluid conditions (low flow rates) stratified liquid-liquid flow in pipelines appears naturally in several engineering applications such as oil and process industries. In this connection several modeling studies on this topic have appeared in the open literature. For horizontal pipes notable works include those of Russell et al [14], Charles et al[8], Brauner and Maron [5], [6], Arirachakaran et al [3]. Two-fluid models employing full Navier-Stokes equations were developed by Kurban [11].

Theron and Unwin [17] developed stratified (liquid-liquid) flow model for interpreting production logging data for

Manuscript received October 6, 2011. This work was supported by the National Science, Technology and Innovation Plan (NSTIP) under project No. 09-OIL 788-04.

Mohammed A. Al-Yaari is with Chemical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia (corresponding author to provide phone: +966-3-860-7279; fax: +966-3-860-4234; e-mail: malyari@ kfupm.edu.sa).

Basel F. Abu-Sharkh is with Chemical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia (e-mail: Sharkh@ kfupm.edu.sa).

horizontal wells in order to assist in measuring the oil and water flow rates.

Hui et al [10] simulated stratified oil–water two-phase turbulent flow in a horizontal tube numerically using a volume of fluid model. They applied RNG k–ε model combined with a near-wall low-Re turbulence model to each phase, and they adopt continuum surface force approximation for the calculation of surface tension. They performed a time-dependent simulation and they analyzed final solution which corresponds to steady-state flow. They reported that their results of pressure loss, slip ratio, local phase fraction profile and the axial velocity profile are verified by experimental data in literature.

Awal et al [4] achieved CFD simulation tool to investigate inline oil and water separation characteristics under downhole conditions. Specifically, they investigated the startling sensitivity of well inclination in the 80-100 degrees range. They chose the *Eulerian-Eulerian* model, which is computationally most comprehensive but more suitable for multiphase systems with the dispersed phase exceeding 10% v/v. The base case model for their CFD simulation study was a horizontal well. They run the problem in 3D using the standard k-ε turbulence model. They simulated oil (μ= 2.3 cp, ρ=850 kg/m^3) and water flowing in 6.3 in ID well.

Carlos F. [7] developed a 2D model for fully-developed, turbulent-turbulent oil-water stratified flow. The model is based on a numerical solution of the basic governing differential equations using a finite-volume method in a bipolar coordinate system, applying a simple mixing-length turbulence model. In addition, he presented a modified turbulent diffusion model. He tested and compared his multi-fluid model, incorporated in the commercial CFD code FLUENT, with oil-water dispersed flow data.

De-Sampaio et .al [9] reported numerical and experimental investigation of stratified gas–liquid two-phase flow in horizontal circular pipes. They simulated Reynolds averaged Navier–Stokes equations (RANS) with the k–ω turbulence model for a fully developed stratified gas–liquid twophase flow. They assumed smooth interface surface without considering the effects of the interfacial waves. Based on their comparison of the numerical results with experimental results, they reported that the k–ω model can be applied for the numerical simulation of stratified gas–liquid two-phase flow.

Rashmi et al [13] investigated numerically, using commercial CFD package FLUENT 6.2 in conjunction with multiphase model, the three-dimensional dispersed flow of Oil-water in a horizontal pipe (ID=0.0024 m). They used k-ε model to describe the turbulence in continuous phase. They reported numerical results in terms of the phase distribution profiles and average in-situ hold-up. They argued that their predicted results were seen to be in good agreement qualitatively as well as quantitatively with experimental results available in the literature.

The main focus of the present study is on the numerical determination of the oil-water stratified flow pattern using CFD modeling. The segregated flow is simulated using VOF multiphase with RNG-k-ε turbulent models and compared with those flow pattern monitored experimentally by Al-Yaari, et al [1].

II. Mathematical Model

The commercial FLUENT software package, FLUENT 6.2, was used for solving the set of governing equations. The numerical method employed is based on the finite volume approach (Fluent, 2001). Fluent provides flexibility in choosing discretization schemes for each governing equation. The discretized equations, along with the initial condition and boundary conditions, were solved using the segregated solution method to obtain a numerical solution.

The Volume of Fluid (VOF) model is employed to predict the oil–water stratified flow behavior in a 0.0254 m horizontal pipe. In the VOF method, two phases are modeled as two separated phases.

In the region near the wall, the gradient of quantities is considerably high and requires fine grids close to the wall to capture the change of quantities. This causes the calculation to become more expensive meaning time-consuming, requiring greater memory and faster processing on the computer, as well as expensive in terms of complexity of equations. A wall function, which is a collection of semi-empirical formulas and functions, provides a cheaper calculation by substituting the fine grids with a set of equations linking the solutions' variables at the near-wall cells and the corresponding quantities on the wall.

III. Numerical Solution

A. Geometry

Figure 1 & 2 show a photograph and schematic layout of the experimental setup, respectively. The flow patterns of two-phase flow in a straight-tube depend on different parameters such as diameter, pipe roughness, pipe material, velocity, physical properties, …etc (Al-Yaari et al [1]).

A sketch of the geometry of the calculation domain is shown in Figure 3. The geometry consists of three parts, i.e. the oil inlet pipe, the water inlet pipe and the acrylic test section. The length of the oil and water inlet pipes is 30 Cm (11.8 times the diameter of the pipe). The test section (acrylic pipe) is 2.2 m. The diameter of the pipe for the present work is 0.0254 m.

The computational grid of 63,493, 82,933, 104,533 and 147,733 cells were generated and used for the mesh independent study to find out the optimum size of the mesh. The grid was generated using Gambit 2.2, which is compatible with Fluent 6.2.

A boundary layer, which contains four cells with a distance of the cell adjacent to the wall at 1 mm, and the growth factor of 1.2, is employed on the wall to improve the performance of the wall function and to fulfill the requirement of y^+, the dimensionless wall distance, for the cell adjacent to the wall which is in the range 50–500 (Fluent, 2001).. The dimensionless distance y^+ is defined by:

$$y^+ = \frac{\rho u_\tau y_p}{\mu}$$

To obtain better convergence and accuracy for a long pipe, the hexagonal shape and Cooper-type elements have been employed. The Cooper-type element is a volume meshing type in Gambit, which uses an algorithm to sweep the mesh node patterns of specified 'source' faces through the volume.

The space domain for the CFD analysis refers to 0.0254 m ID pipe. A three-dimensional mesh has been set up, by adding further volumes test section.

Figure 1 Photograph of the flow loop

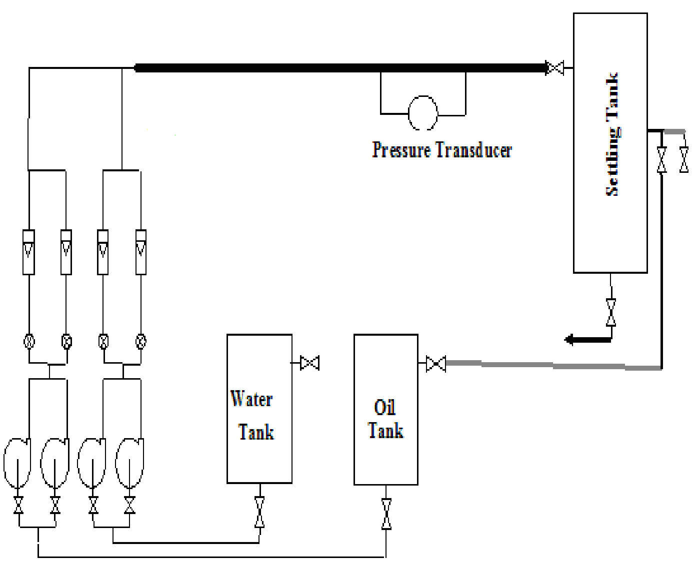

Figure 2 Schematic layout of the flow loop

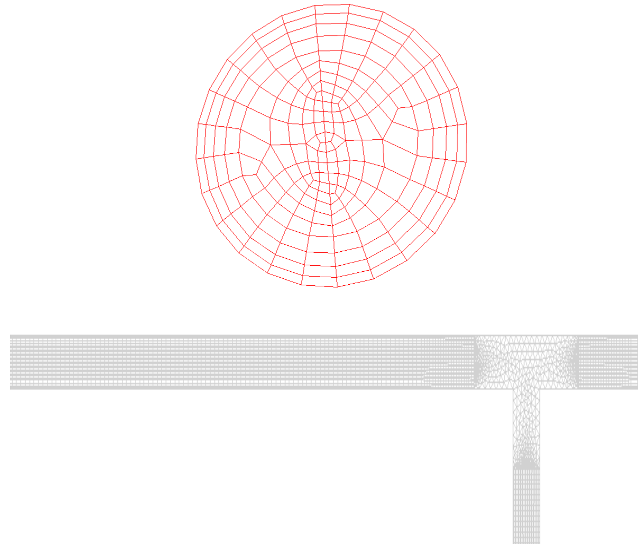

Figure 3 Mesh systems for the circular cross-sectional pipe test section

B. Boundary Conditions

There are three faces bounding the calculation domain: the inlet boundary, the wall boundary and the outlet boundary. Flat velocity profile for oil and water were introduced at the inlet of their sections. The outlet boundary condition of the latter was set up as a pressure outlet boundary. No slip was used to model liquid velocity at the wall.

The main fluid phases' physical properties are reported in Table 1.

Table 1 Main physical properties for the fluid phases

Property	Water Phase	Oil Phase
Density (ρ), kg/m^3	998.2	780
Dynamic Viscosity (μ), Pa.s	0.001003	0.00157
Interfacial Tension, N/m	0.017 N/m @ 20°C	

C. Solution Strategy & Convergence

While Presto discretization scheme was used for pressure, first order upwind discretization scheme was used for the momentum equation volume fraction, turbulent, kinetic and turbulent dissipation energy. These schemes ensured, in general, satisfactory accuracy, stability and convergence. In addition, the steady-state solution strategy was employed.

The convergence criterion is based on the residual value of the calculated variables, i.e. mass, velocity components, turbulent kinetic energies, turbulent dissipation energies and volume fraction. In the present calculations, the threshold values were set to a hundred thousandth for continuity and a thousandth for the remaining equations. These values are considered small enough to produce accurate results.

Other solution strategies are: the reduction of under-relaxation factors of momentum, volume fraction, turbulence kinetic energy and turbulence dissipation energy to bring the non-linear equation close to the linear equation, subsequently using a better initial guess based on a simpler problem.

IV. Experimental & Simulation Results

In this section one presents, compares and discusses the experimental and numerical data.

A. Experimental Oil-Water Flow Patterns

The flow pattern was observed for a wide range of mixture velocities (0.5 m/s to 3.5 m/s) and input water volume fraction range of (0.1 to 0.9). This observation was made at 1.1 m from the inlet of the test section. A variety of flow patterns were formed in co-current flow of oil and water in the acrylic test section. In general, the flow pattern map depends on the geometry, liquid physical properties and wetting properties of the wall surface. The flow pattern classification was based on visual observation and does not necessarily represent unique hydrodynamic characteristics. Figure 4 shows the flow patterns observed which are defined as follows:

I. **Stratified wavy flow (SW)**. The phases are completely segregated with the interface between them showing a characteristic wavy nature.

II. **Stratified wavy / drops (SWD).** The entrainment of one or both phases as drops in the other has begun, the droplets being concentrated near the interface zone (stratified wavy with mixing interface).

III. **Stratified mixed / water layer (SMW)**. There are two layers in the flow: a lower clear water layer and an upper layer which can be oil continuous containing a dispersion of water droplets or water continuous containing a dispersion of oil droplets or a combination of the two.

IV. **Stratified mixed / oil layer (SMO)**. There are two layers in the flow: an upper clear oil layer and a lower layer which can be oil continuous containing a dispersion of water droplets or water continuous containing a dispersion of oil droplets or a combination of the two.

V. **Three layers flow**. There are clear oil and water layers at the top and bottom of the pipe respectively with a dispersed layer between them.

VI. **Dispersed flow**. One phase is completely dispersed as droplets in the other. The continuous phase changes from one fluid to the other at the phase inversion point.

The resultant flow pattern data for oil-water are plotted in Figure 5 in terms of input water volume fraction against superficial mixture velocity. As illustrated in this figure, the stratified flow pattern was observed for the whole examined range of the input water volume fraction at a very low superficial mixture velocity (0.5 m/s). As the mixture velocity increased to 1 m/s, the stratified flow pattern changed to stratified wavy with drops, three layers and stratified mixed/water layer flow patterns successively with increasing water fraction at input water volume fraction of 0.35, 0.55 and 0.75 respectively. Numerical study will focus on such type of flow pattern; i.e., stratified flow pattern.

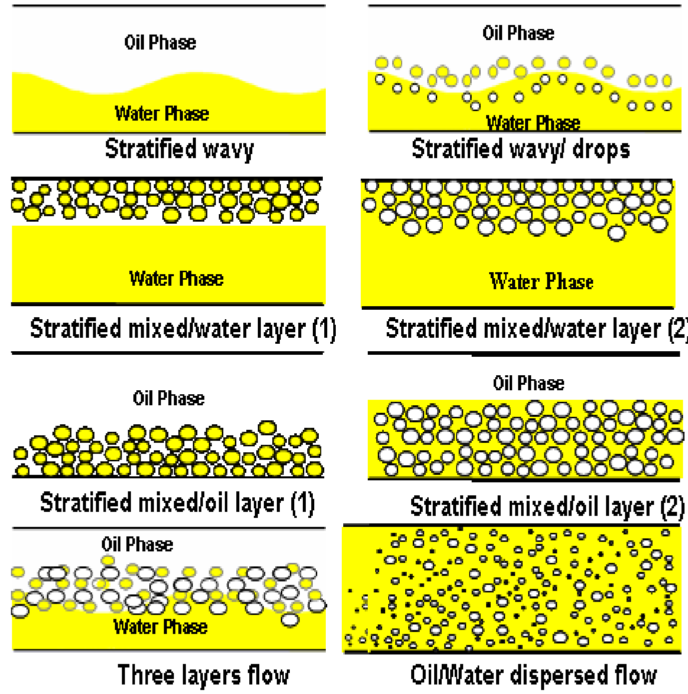

Figure 4 Oil-water flow patterns in a horizontal pipe with 0.0254 m ID

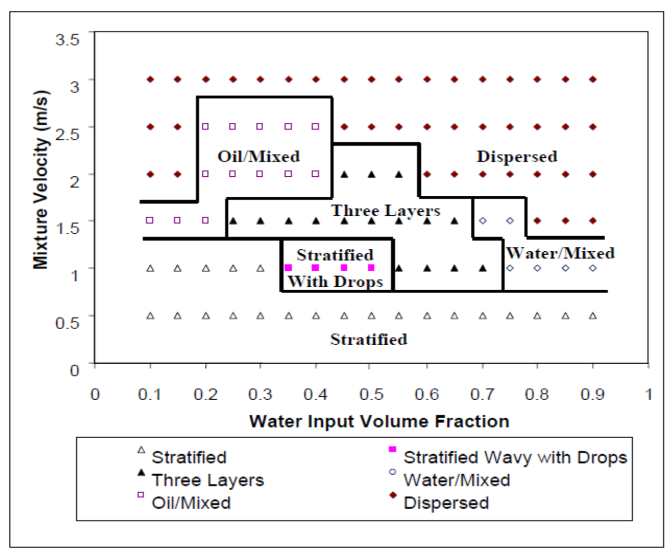

Figure 5 Flow pattern map of oil-water flow

B. Model Selection

Different multiphase models (VOF, Mixture and Eularian) were tested to model the oil water stratified flow pattern. In addition different turbulent models (RNG-k-ε, Releazable- k-ε, RSM, and SA) have been adopted to figure out the best selection to be used to simulate oil-water stratified flow in a horizontal pipe. RSM (1) and RSM (2), in Figure 7 below, represent the turbulent model without and with defining the interfacial tension between oil and water respectively. RNG (1) and RNG (2), in Figure 8 below, represent the turbulent k-ε model with standard and enhanced wall function respectively.

Before making selection, all models oil volume fraction results should be at an x-section where the flow is fully developed. Static pressure plot at the centerline of the test section (in x-direction) is presented in Figure 6 below. Based on that, x=-1.4 m section was made.

For the coarse mesh, all of the models fail to predict the stratified flow pattern as presented for some of them in Figure 7 below. This can be attributed to that the mesh was not fine enough to predict the oil-water stratified flow pattern. However, for finer meshes, while almost all the models predict the clearly separated oil layer, they do fail to predict clearly water separated layer and this problem should be figured out before simulating other flow patterns presented in Figure 5. Based on the time required for convergence to occur and literature and Fluent recommendations, VOF multiphase model with RNG-k-ε model has been selected to achieve the simulation.

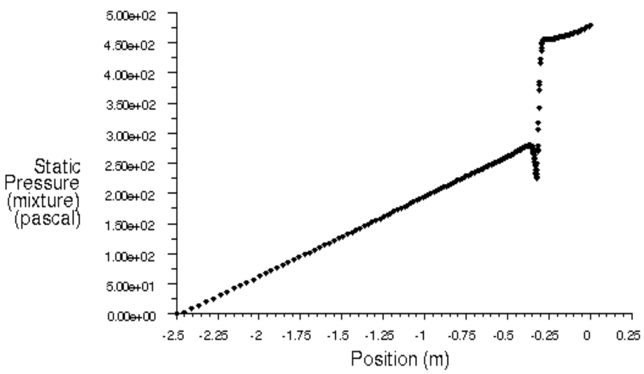

Figure 6 Pressure plot at the test section centerline in the flow direction

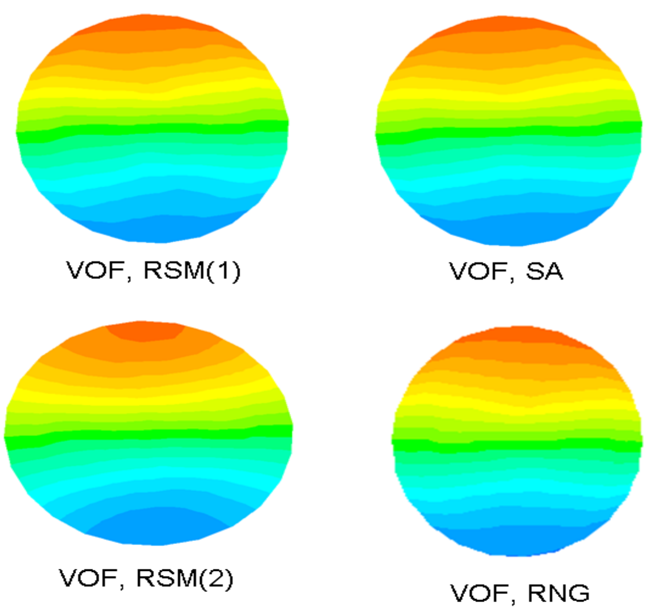

Figure 7 Oil volume fraction contours at x=-1.4 m (for the coarse mesh)

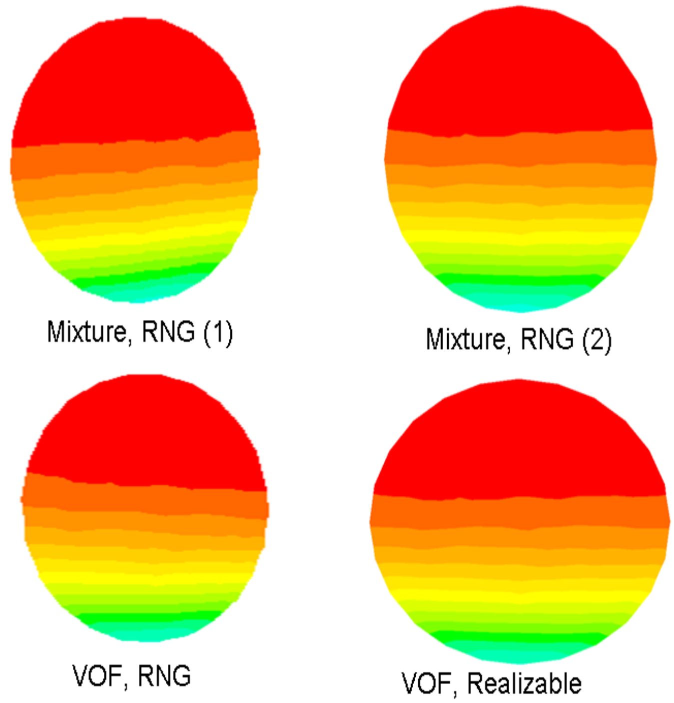

Figure 8 Oil volume fraction contours at x=-1.4 m (for finer mesh)

C. Mesh Independent Study

The computational grid of 63,493, 82,933, 104,533 and 147,733 cells were tested for the mesh independent study to find out the optimum size of the mesh to be used for simulation. Oil volume fraction contours at plane x=-1.4 m are presented in Figure 9 below.

As shown in this figure, system with 63,493 and 82,933 cells almost predict the same oil fractions with differences in the smoothness of the clearly oil and mixed layer. However, since 63,493 cells was bad while predicting the stratified flow pattern at other water volume fraction, extra number of cells are needed to be tested. Fortunately, system with 104,533 and 147,733 cells give the same oil volume fraction contours at x=-1.4 m. Therefore, based on the oil volume fraction contours results, 104,533 cells are the optimum number of cells required to predict the oil-water stratified flow in the tested domain and such mesh is going to be used while simulation.

In addition, such decision has been tested with comparing static pressure contours at x=-1.4 m for all the four tested number of cells. Static pressure contours of all tested cases are presented in Fig. 10 below. As shown in that figure, the static pressures at x=-1.4 m were 437, 437, 465 and 465 Pa for 63,493, 82,933, 104,533 and 147,733 cells respectively. Such results support the previous results and decision.

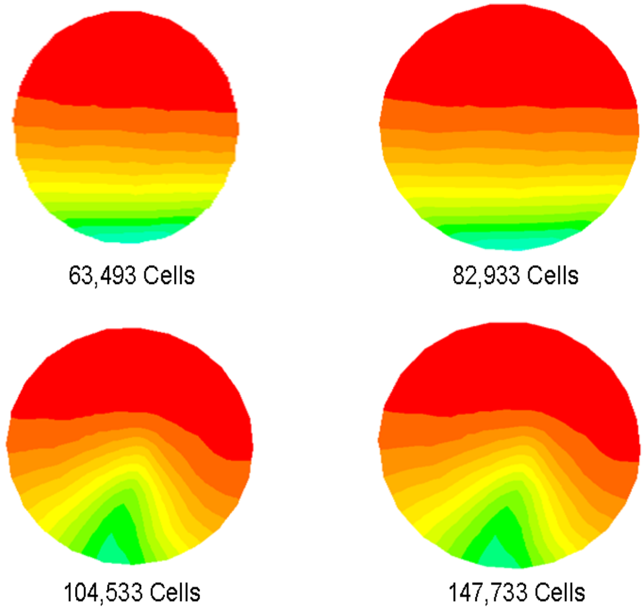

Figure 9 Oil volume fraction contours at x=-1.4 m for mesh sizes

D. Some CFD Results for One Case Study

The oil-water stratified flow pattern at oil-water mixture velocity of 1 m/s, 0.85 oil velocity and 0.15 water velocity (0.15 input water volume fraction) as a sample flow pattern has been simulated. At such condition, the oil-water flow pattern is stratified (See Figure 5).

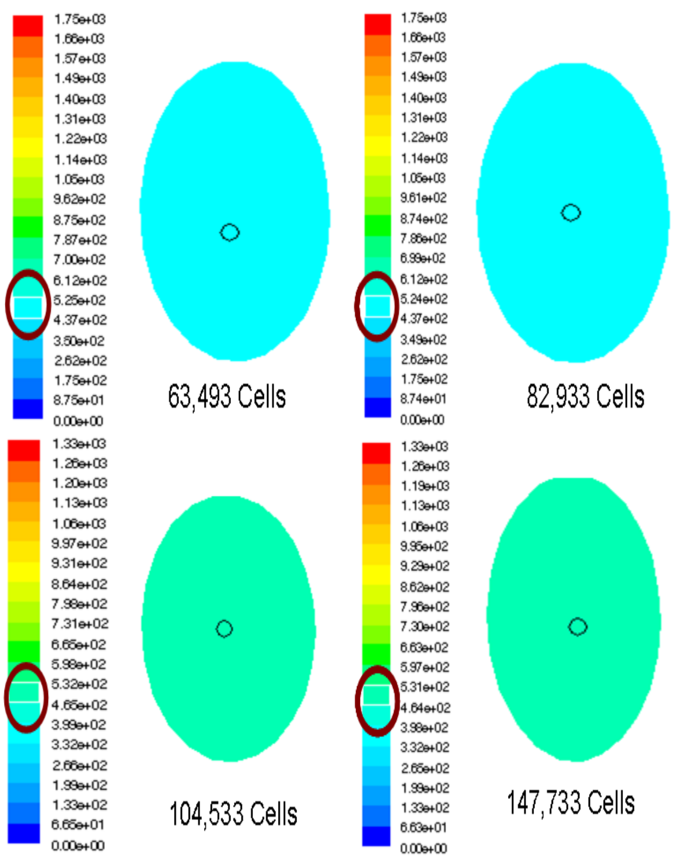

Figure 10 Static pressure contours at x=-1.4 m for mesh sizes

Volume of fluid (VOF) multiphase model with RNG-k-ε model was used for simulation the tested domain containing 104,533 cells (the optimum mesh size) based on the decision mentioned earlier in this paper. For this case, since water fraction is very low, initializing with 1 water fraction is the best to get convergence. In addition, 2% turbulence intensity was used since the flow if stratified way as observed experimentally (Al-Yaari, et al [1]).

The oil volume fraction contours are presented in Figure 11 below. As shown in this figure, clearly separated oil layer with a wavy interface (stratified wavy flow pattern) was predicted by CFD simulation and this matching results reported by Al-Yaari, et al [1]. On the other hand, separated water layer was not predicted completely. Therefore, much effort should be put to solve such problem.

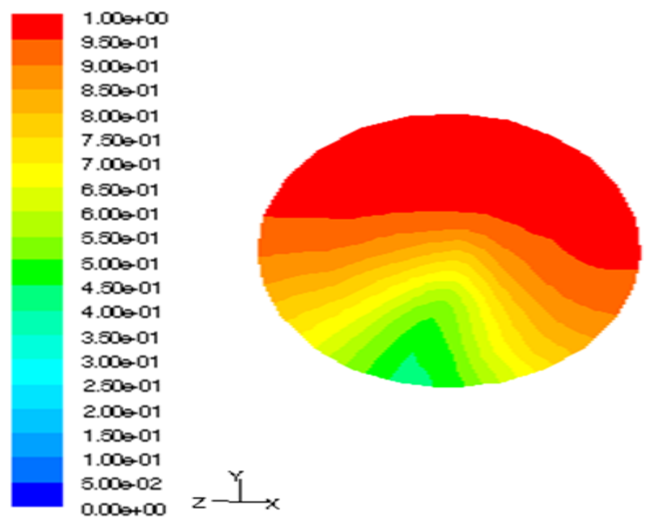

Figure 11 Oil volume fraction contours at x=-1.4 m for the optimum mesh size

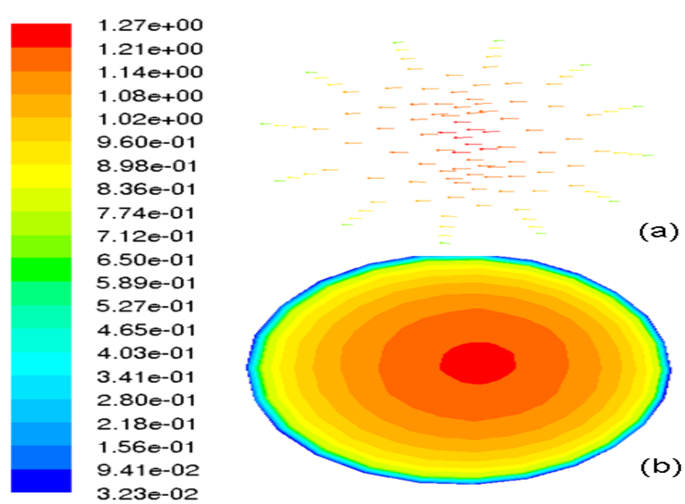

Figure 12 (a) Mixture velocity vectors at x=-1.4 m for the optimum mesh size
(b) Mixture velocity contours at x=-1.4 m for the optimum mesh size

Furthermore**,** the oil-water mixture vectors and contours at x=-1.4 m are presented in Figure 12 above. As shown in such figure, the no slip condition on wall was applied and mixture velocity increases with decreasing the radius of the flow area.

V. Conclusions

The following conclusive remarks result from our analysis. As far as the fluid dynamic analysis is concerned:

1. CFD calculations using Fluent 6.2 were performed to predict the oil-water stratified flow in 0.0254 m horizontal pipe.
2. Volume of fluid (VOF) multiphase model with RNG-k-ε two equations turbulent model was selected among other different multiphase and turbulent models based on the convergence, prediction of the oil-water stratified flow pattern and the smoothness of the interface.
3. Care should be taken while initializing the CFD solver to obtain convergence.
4. Mesh independent study has been achieved to decide on the optimum mesh size to be used in the simulation process.
5. Some results of the CFD simulation for one case study, where the stratified oil-water flow pattern was observed experimentally by Al-Yaari, et al [1].
6. While the clearly separated oil layer and the wavy interface of oil-water system were predicted well by CFD simulation, clearly separated water layer was not predicted completely. Therefore, such problem should be solved before simulate other stratified points.

Acknowledgment

Authors are also grateful for useful discussions with Dr. Rached Ben-Mansour of Mechanical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia.

References

[1] Al-Yaari, M., Abu-Sharkh, B., Soleimani, A., Al-Mubayeidh, U. and Al-Sarkhi, A. “Effect of Drag Reducing Polymers on Oil-Water Flow in a Horizontal Pipe”, *International Journal of Multiphase Flow*, **2009**, 35, 516–524.

[2] Angeli, P., “Liquid-Liquid Dispersed Flows in Horizontal Pipes”, *PhD Thesis*, Imperial College, University of London, **1997**.

[3] Arirachakaran, S., Oglesby, K. D., Malinowsky, M. S., Shoham, O. and Brill, J. P. “An analysis of oil/water flow phenomena in horizontal pipes”, *SPE Production Operations Symposium, Oklahoma*, **1989,** March 13-14, *SPE* 18836.

[4] Awal, Mohammad R., Zughbi, Habib D., Razzak, Shaikh A., Al-Majed, Abdulaziz A., and Al-Yousef, Hasan Y., “Liquids phase holdup and separation characteristics as a function of well inclination and flowrate”, *SPE SA*, **2005**, 14-16 May.

[5] Brauner, N. and Maron, D.M. “Two-phase liquid-liquid stratified flow”, *PhysicoChem. Hydrodynamics,* **1989**, 11(4), 487-506*.*

[6] Brauner, N. and Moalem Maron. D., “Stability analysis of stratified liquid-liquid flow”, *Int. J. Multiphase Flow,* **1992,** 18(1), 103-121.

[7] Carlos F. “Modelling of oil-water flow in horizontal and near horizontal pipes”, *PhD Thesis*, Tulsa University, **2006.**

[8] Charles, M.E., Govier, G.W., Hodgson, G.W. “The horizontal pipeline flow of equal density oil-water mixtures”, *Can. J. Chemical Engineering*, **1961**, 39, 27-36.

[9] De Sampaio, Paulo, Faccini, Jose´ and Su, Jian, “Modelling of stratified gas–liquid two-phase flow in horizontal circular pipes”, International Journal of Heat and Mass Transfer, **2008,** 51, 2752–2761.

[10] Hui Gao, Han-Yang Gu and Lie-Jin Guo, “Numerical study of stratified oil–water two-phase turbulent flow in a horizontal tube”, *International Journal of Heat and Mass Transfer*, 2003, 46, 749–754.

[11] Kurban, A., Angeli, P., Mendes-Tatsis, M. and Hewitt, G. “Stratified and dispersed oil-water flows in horizontal pipes”, *7th International Conference on Multiphase Production, Cannes, France,* **1995**, 277-291.

[12] Oglesby, K. “An experimental study on the effects of oil viscosity, mixture velocity, and water fraction on horizontal oil-water flow”, *M.S. Thesis*, U. Tulsa, **1979**.

[13] Rashmi G. Walvekar, Thomas S.Y. Choong, S.A. Hussain, M. Khalid and T.G. Chuah, “Numerical study of dispersed oil–water turbulent flow in horizontal tube”, *Journal of Petroleum Science and Engineering,* **2009**, 65, 123–128.

[14] Russell, T., Hodgson, G. and Govier, G. “Horizontal pipeline flow of mixtures of oil and water”, *Canadian J. Chemical Engineering*, **February 1959**, 37, 9-17.

[15] Shi, H., Cai, J. and Jepson, W. “Oil-water two-phase flows in large-diameter pipelines”, ASME Transactions, *J. of Energy Resources Technology*, **2001,** 123(4), 270-276.

[16] Soleimani, A., “Phase distribution and associated phenomenon in oil-water flows in horizontal tube”, *PhD Thesis*, Imperial College, University of London, **1999**.

[17] Theron, B.E. and Unwin, T. “Stratified flow model and interpretation in horizontal wells”, *Proceedings of the 1996 SPE Annual Technical Conference and Exhibition*, Part Omega, Denver, CO, USA, **1996**, Oct. 6-9, 749-757.

[18] Trallero, J., Sarica, C. and Brill, J. “A study of oil/water flow patterns in horizontal pipes”, *SPE Production and Facilities*, **August 1997**, 12(3), 165-172.

Mohammed A. Al-Yaari was born in Sana'a, Yemen, on August 25, 1977. He received the B.Sc. degree in Chemical Engineering from Baghdad University, Baghdad, Iraq, in June 2000 and the M.Sc. degree in Chemical Engineering from King Fahd University of Petroleum & Minerals (KFUPM), Dhahran, Saudi Arabia, in June 2008. He is a PhD Candidate in the Chemical Engineering Department in KFUPM.

His research interest includes drag reduction by polymer additives, CFD simulation and emulsion technology.