

CFD Simulation for Stratified Oil-Water Two-Phase Flow in a Horizontal Pipe

Adib Zulhilmi Mohd Alias,^a, Jaswar Koto,^{a,b,*} and Yasser Mohamed Ahmed,^a

^aDepartment of Aeronautics, Automotive and Ocean Engineering, Universiti Teknologi Malaysia, Malaysia

^bOcean and Aerospace Research Institute, Indonesia

*Corresponding author: jaswar.koto@gmail.com

Paper History

Received: 10-October-2014

Accepted: 18-November-2014

ABSTRACT

Oil-water two-phase flow in 0.0254m horizontal pipe is simulated using FLUENT 6.2. The stratified flow regime is modeled using Volume of Fluid (VOF) with turbulent model RNG k- ϵ . Grid independent study has been conducted to decide mesh size for solution accuracy and optimum computational cost. The simulation is performed in time-dependent simulation where oil and water are initially separated by patching the region base on difference in density. Observation on the effect of velocity to the pressure gradient was also simulated. Flow velocity at 0.2, 0.5, 0.8 and 1.1 m/s with same volume fraction for each phase with appropriate multiphase model and turbulence model are presented.

Keywords: *Stratified oil-water flow; Turbulence flow; CFD*

1.0 INTRODUCTION

Immiscible liquid-liquid flow is a common occurrence encountered in a variety of industrial processes. In oil and gas industry, oil transportation either from reservoir to processing facilities or to onshore refinery are usually transported in multiphase flow condition since water and oil are normally produced together. Fractions of water are usually influenced by its existence within the stratum and also through oil recovery method which used water to enhance the remaining oil in the reservoir.

The presence of water, during the transportation of oil has a significant effect because the flow is no longer can be treated as a single-phase flow. Oil-water has complex interfacial structure which complicates the hydrodynamic prediction of the fluid flow. Changes in water fraction may influence the power required to pump the fluid due to corresponding changes in pipeline pressure drop. Either water-in-oil or oil-in-water dispersions, both can influence the pressure gradient dramatically.

Computational fluid dynamics (CFD) techniques have been used to simulate the stratified pipe flow. One of the early CFD models of turbulent stratified flow in a horizontal pipe was presented by Shoham and Taitel^[1] where a 2D simulation for liquid-gas flow was simulated by adopting zero-equation models for the liquid region flow field while the gas region was treated as a bulk flow. Issa^[2] numerically simulated the stratified gas-liquid pipe flow, using standard k- ϵ turbulence model with wall functions for each phase. Newton and Behnia^[3] obtained more satisfactory solutions for stratified pipe flow by employing a low Reynolds number turbulent model instead of wall functions.

Hui et al^[4] simulated stratified oil-water two-phase turbulent flow in a horizontal tube by applying 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. Their simulation results was compared with Elseth et al^[5] who simulated the turbulent stratified flow, however their numerical results are not acceptable when compared with their measured data.

Stratified oil-water two-phase pipe flow was investigated using different type of multiphase model. Awal et al^[6] achieved CFD simulation tool to investigate inline oil and water separation characteristics under downhole conditions. 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. Carlos F.^[7] developed a 2D model for fully-developed, turbulent-turbulent oil-water stratified flow using finite-volume method in a bipolar

coordinate system and applying a simple mixing-length turbulence model. Hui et al [4] and Al-Yaari et al [8] simulated stratified oil-water two-phase turbulent flow in a horizontal tube numerically using a volume of fluid (VOF) model. They applied RNG k-ε model with enhanced wall function combined with optimum meshes through grid independent study to obtain clearly separated oil layer and optimum computational cost.

In the present paper multiphase model of Volume of Fluid (VOF) is used to model the stratified oil-water flow. Optimum number of elements for simulation accuracy has been conducted through grid independent study. Observation on the effect of velocity to the pressure gradient was also simulated at flow velocity 0.2, 0.5, 0.8 and 1.1 m/s with same volume fraction for each phase.

2.0 NUMERICAL SIMULATION

2.1 Geometry and mesh

The domain and the meshes were created using ANSYS Design Modeler. A sketch of the geometry of the calculation domain is shown in Figure 1. The geometry consists of semicircular inlet for oil and water with 1 meter length of the flow domain. The inlet for both phases is at the same inlet face where oil on top and water at the bottom region. This will initially made the flow in stratified condition. In addition, as both inlets also flew with a same velocity with direction almost parallel to each phase makes fewer disturbances to maintain stratified flow. The diameter of the pipe for the present work is 0.0254 m. In order to keep the volume of oil and water are flowing continuously throughout the domain until the outlet, patch file and adapt region is used to declare the top and bottom regions for oil and water. This will avoid insufficient volume of either phase.

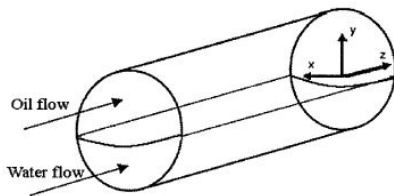


Figure 1: Schematic representation of pipe flow

A block-structured meshing approach was used to create meshes with only tri/tet cells. To obtain fine meshing scheme, sizing was setup with curvature normal angle 11 degree, 0.0001 minimum size and 3.0 m maximum size. While to improve the flow near the wall region, two layer inflation with growth rate 1.2 is adapted

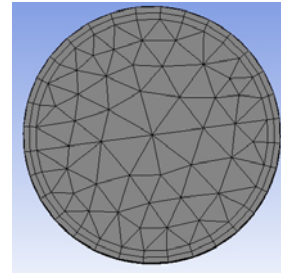


Figure 2: Tri/Tet meshes

2.2 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 at the end 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: Fluid phases physical properties

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	0.17 m @ 20°C	

2.3 Solution strategy and convergence

Pressure-based solver is chose since it was applicable for wide range of flow regimes from low speed incompressible flow to high speed compressible flow. This solver also requires less memory (storage) and allows flexibility in the solution procedure. Green-gauss Node-Based is elected for higher order discretization scheme since it is more accurate for tri/tet meshes. For pressure, PRESTO! discretization scheme was used for pressure, second order upwind discretization scheme was used for the momentum equation, volume fraction, turbulent, kinetic and turbulent dissipation energy. Second-order upwind is chose rather than First-order upwind because it uses larger stencils for 2nd order accuracy and essential with tri/tet mesh even though the solution to converge may be slower but manageable. In addition, the simulation is time dependent (transient) with 1000 time steps, 0.01 time step size and 200 iterations at each time step size.

3.0 RESULTS

In this section one presents, use of Volume of Fluid multiphase model along with RNG k-ε for turbulent model, grid independent test and sample of pressure drop prediction using this simulation

3.1 Grid independent study

A grid independent study is conducted to obtain sufficient mesh density as it was necessary to resolve accurate flow. A grid independent solution exists when the solution does not change when the mesh is refined. The computational grid of 46631,

79488, 104584 and 142374 elements were tested for the mesh independent study to find out the optimum size of the mesh to be used for simulation. Figure 3 shows an oil volume fraction contours at plane $z = 0.5$ m which indicates the accuracy of the mesh to display the flow pattern. As shown in figure, system increased number of elements shows better prediction for stratified flow pattern with smoothness of the clearly oil and mixed layer. 46631 showing bad prediction on the oil and mixed layer since insufficient amount of elements could not give detail prediction especially on the mixed layer. Both meshes for 104584 and 142374 gave almost similar contours of oil fraction with slight differences in the smoothness of the clearly oil and mixed layer. Therefore, based on the oil volume fraction contours results, 142374 cells are the most optimum number of cells required to predict the oil-water stratified flow in the tested domain and such mesh is going to be used for simulation.

In addition, such decision has been tested by comparing the pressure profiles obtain for every meshes tested (Chart 1). At mesh size 46631, 68204 and 79488, the pressure plot is away from the other plots. The pressure profile starts to unchanged with mesh 92440 until 171393. Before deciding the best meshes size, simulation cost also is required to look at. Since increase num of meshes will increase the amount of time for simulation, the meshes size of 142374 is the most optimum number of elements could be chose.

3.2 Pressure prediction at different flow velocity

By using the simulated oil-water stratified flow, pressure prediction at different flow velocity have been conducted. Flow velocity of 0.2, 0.5, 0.8 and 1.1 m/s with (0.5 input water volume fraction) as a sample flow pattern has been simulated. Volume of fluid (VOF) multiphase model with RNG k- ϵ model was used for simulation the tested domain containing 142374 cells (the optimum mesh size) based on the decision mentioned earlier in this paper. At such condition, the oil-water flow pattern simulated is seen stratified (see figure 4) with multiple layers of phase density in the middle of the pipe where the oil and water phases met. Figure 5 shows the view of oil volume fraction contours at pipe length ($z = 0.5$ m) which located in the middle of the pipe length. (Chart 2). Different velocity indicates different inversion point. 0.2 and 0.5 m/s can be considered as slow speed which gives more time for both phases to dispersed within each other. On the view of oil production is not good since avoiding mixing phases will reduce time during separation processes. 0.8 and 1.1 m/s shows better oil and water mixture. From the contours seen the fraction of oil at the upper region shows high fraction of oil. It indicates less water inversion to its phase.

4.0 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 off the oil-water stratified flow pattern and the smoothness of the interface.

3. Mesh independent study has been achieved to decide on the optimum mesh size to be used in the simulation process.
4. Pressure prediction base on different flow velocity have been observed. It can be seen that as velocity increases, the pressure gradient also increases.
5. The pressure prediction will be extended to examine the effect from different water volume fraction.

ACKNOWLEDGEMENTS

The authors are very grateful to Universiti Teknologi Malaysia, Ocean and Aerospace Research Institute, Indonesia for supporting this study. Authors are also grateful for useful discussions with my family and all my friends.

REFERENCE

1. O.Shoham, Y.Taitel, "Stratified turbulent-turbulent gas-liquid flow in horizontal and inclined pipes", AICHE J. 30 (2) (1984) 377-385.
2. R.I, Issa, "Prediction of turbulent, stratified, two-phase flow in inclined pipes and channels", Int. J. Multiphase Flow 14 (1) (1988) 141-154.
3. C.H. Newton, M. Behnia, "Numerical calculation of turbulent stratified gas-liquid pipe flows", Int. J. Multiphase Flow 24 (5) (1998) 141-154.
4. Hui gao, Han-Yang Gu and Lie-Jin Guo, "Numerical study of stratified oil-water two-phase turbulent flow in a horizontal tube", Int. J. heat and mass transfer, (46) (2003) 749-754
5. G. Elseth, H.K. Kvandal, M.C. Melaaen, "Measurement of velocity and phase fraction in stratified oil-water flow", *International Symposium on Multiphase Flow and Transport Phenomena*, Antalya, Turkey, pp.206-210, 2000
6. 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.
7. Carlos F. "Modelling of oil-water flow in horizontal and near horizontal pipes", PhD Thesis, Tulsa University, 2006.
8. Al-Yaari, M., and Abu-Sharkh, "CFD Prediction of stratified oil-water flow in a horizontal pipe", Asian Transactions on Engineering Volume 01 Issue 05, November 2011.

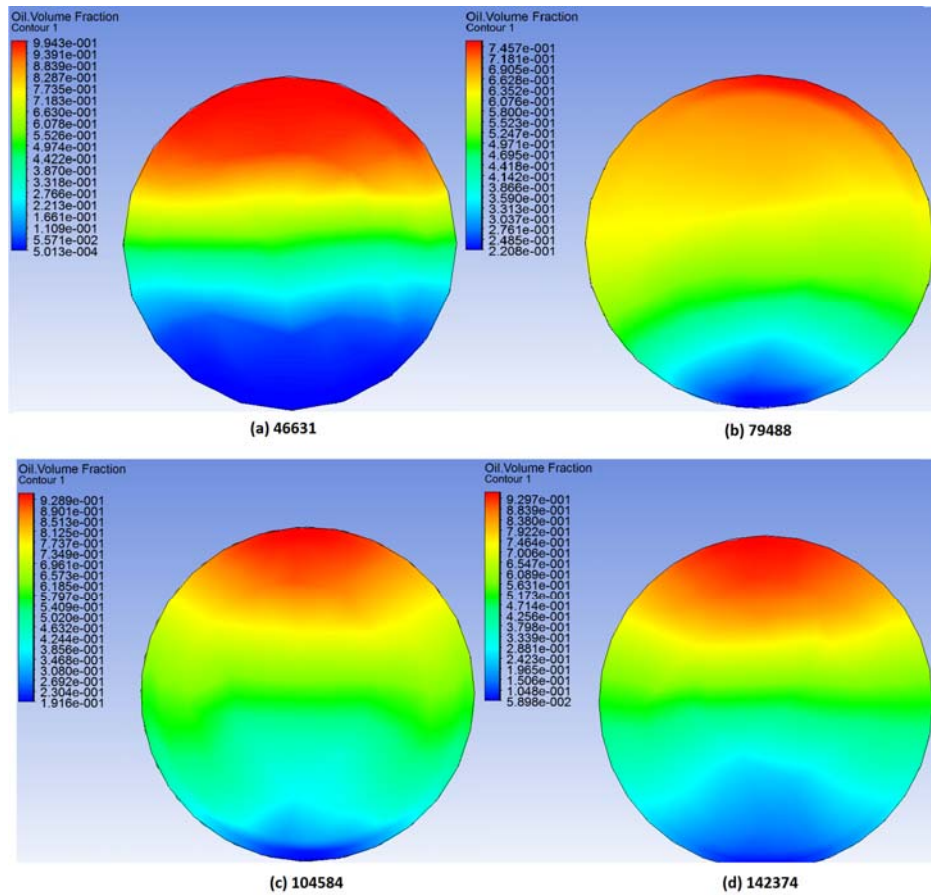


Figure 3: Oil volume fraction contours at pipe length ($z = 0.5$ m)



Chart 1: Optimum mesh size at unchanged pressure profile

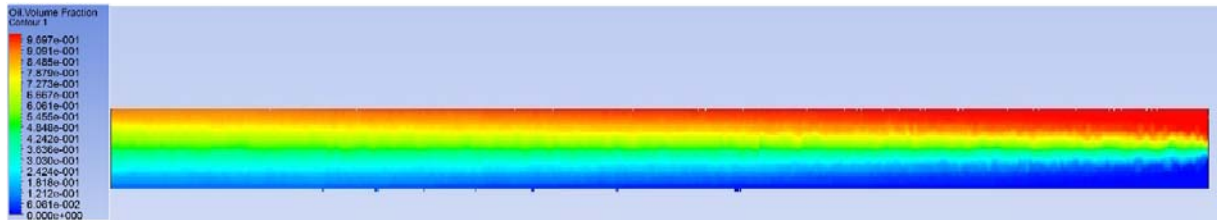


Figure 4: Stratified Oil-water flow simulation

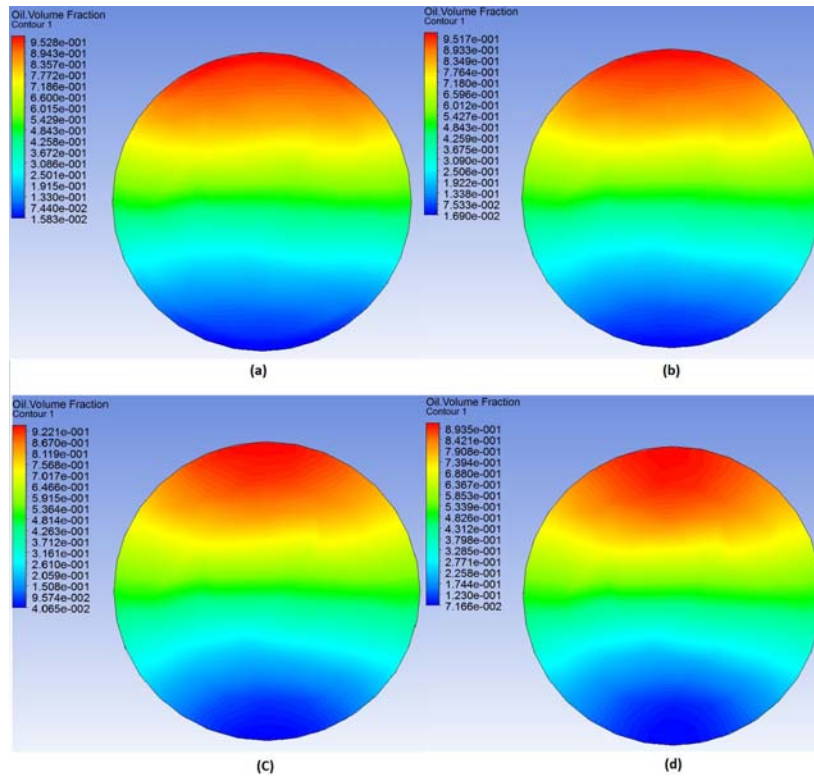


Figure 5: Oil volume fraction contours at pipe length ($z = 0.5$ m); (a) 1.1 m/s (b) 0.8 m/s (c) 0.5 m/s (d) 0.2 m/s

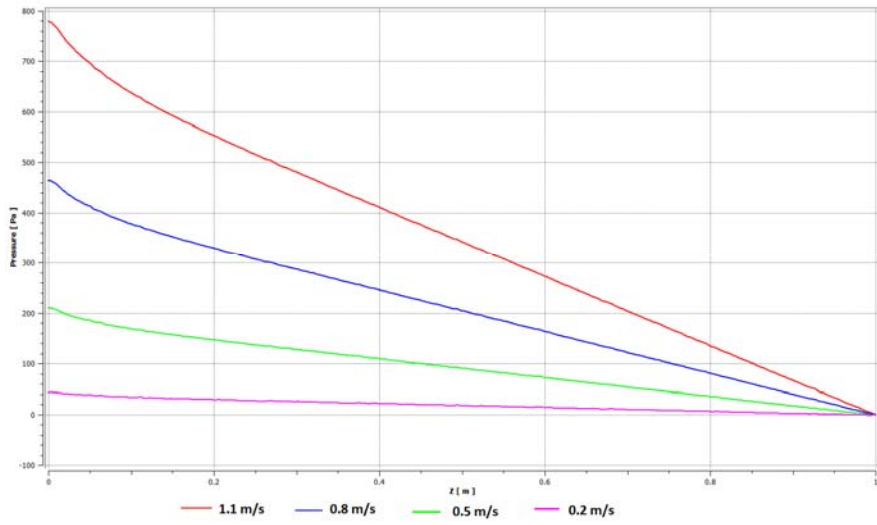


Chart 2: Pressure profile at each flow velocity