Abstract

Although two-phase stratified flow pattern commonly occurs in petroleum industry the understanding of it, in terms of structure of the wave interface and the ensuing interaction between the two phases, is limited compared to single phase flow. Also, most of the studies have not taken the velocity profile across the pipe cross-section into account, rather just focused on the average velocity of each cross section. This paper will provide a unique insight into these velocity profiles, which are critical for frictional pressure drop calculations and prediction of phenomena such as wall effects of multiphase flow, erosion, corrosion, hydrate formation, wax deposition, etc. The objective of this paper is the analysis of this gas-liquid flow pattern in a horizontal pipe utilizing Computational Fluid Dynamics (CFD) simulation and comparison of the results with experimental data. The scope also includes the investigation of turbulent flow structure beneath gas-liquid interface by calculating the stream-wise velocity profile.

Experimental studies have been conducted to investigate gas-liquid stratified flow in horizontal pipe. A unique experimental facility was constructed with a 4-in ID PVC pipe and measurements were performed utilizing air-water. Liquid level at the center of the pipe is measured by ultrasonic proximity sensor at different superficial velocities of gas and liquid. Along with experimental tests, CFD simulations for the same test conditions have been performed using a commercial CFD code. For tracking the two-phase interface, Volume of Fluid (VOF) method was applied. The numerical simulation was obtained with the Realizable k-epsilon model of turbulence.

Comparing the CFD simulation results and liquid hold up measured experimentally revealed a good agreement (discrepancy<±15%). The experimental data also show that at constant superficial liquid velocities, the liquid level increases with decreasing superficial gas velocities. The validation of the CFD results with experimental data indicates that, CFD simulation has the potential to be used for facility design and scale-up processes in petroleum industry.

Introduction

Stratified flow is the simple multiphase flow regime, where the heavier phase flows below the lighter phase. This flow regime is the most desirable flow regime for oil and gas industry, since it creates low pressure drop, less erosion and equipment failure, and facilitates easier phase separation. Although the
stratified flow has a simple structure and is the best understood one, a lot of studies are devoted on it primarily to understand the effect of the interfacial wave on pressure drop and liquid entrainment along the pipe.

Estimation of pressure drop and liquid holdup are limited to one dimensional mechanistic modeling, where the shear stress in the interface are calculated based on the average velocity. There is a critical need to conduct CFD simulations to predict the liquid structure and gas-liquid interface in detail. The CFD simulation can help to study multiphase stratified flow at any location within the system reliably and in a cheaper way.

Shoham and Taitel (1984) applied the combination of 2-D momentum equation and eddy viscosity turbulence model to characterize the liquid phase behavior in stratified flow. For the gas phase, the gas and interfacial stresses are estimated based on the bulk flow calculations. Banerjee and Isaac (2003) applied CFD code to simulate stratified flow by using VOF model, and three different turbulent models, namely, Standard k-ε, RNG k-ε and Reynolds Stress. The results show that RNG k-ε turbulence model gives best results compared to the other models. Holmas et al. (2005) employed CFD to find the characteristics of stratified flow in horizontal channel by VOF model. Two turbulence models, namely, RNG k-ε and MSST k-ω predict poor results for interfacial turbulence. Bartosiewicz et al. (2008) applied the VOF to find wave celerity, critical wave number, and also the transition between stratified smooth and wavy. Good agreements have been achieved between the experimental data and the CFD predictions.

Desamala et al. (2014) studied the flow characterization of oil-water flow in horizontal flow with CFD simulation, and good results were obtained for slug, stratified and annular flow with VOF model. Al-Yaari and Abu-Sharkh (2011) simulated stratified oil-water flow for horizontal pipe using RNG k-ε turbulent and VOF models. Issa et al. (1988) combined the momentum equation for gas and liquid with k-ε model, and modeled the stratified flow applying wall function for solid boundaries. The results were in agreement with the Taitel and Dukler mechanistic model. De-Sampaio et al. (2007) applied finite element method to solve the Reynolds averaged Naveier Stokes equations with the k-ω model to model gas-liquid stratified model. Sidi-Ali and Gatingnol (2008) developed CFD simulation with k-ε model to model the stratified flow with and without gravity effect.

In this study, CFD simulations are developed for a 4-in horizontal pipeline, where air is selected as the gas phase, water in chosen as the liquid phase. The CFD simulation is developed with a commercial CFD code using VOF model to capture gas-liquid interface, and Realizable k-ε model is used for turbulence modeling. Water is introduced at the bottom of the inlet section, while air is introduced at the top to evaluate two phase flow developing. The main goals of this paper are calculating liquid hold up, wall shear stresses, streamline velocity profile and liquid height in the pipe centerline.

**Experimental Program**

An experimental facility was designed and constructed at The University of Tulsa’s North Campus Facility with a horizontal 4 in. PVC pipe, which measures liquid velocity, liquid hold up, and pressure drop. The designed test loop utilizes water as its liquid phase, and air as its gas phase. The liquid is injected from a water tank using a centrifugal pump equipped with a 2-hp electric motor. The liquid flow rate is measured by an electromagnetic flow meter, which measures water flow rate with high accuracy. The water flow rate is controlled by a manual valve at the downstream of the centrifugal pump. The tank outlet line is connected to a diaphragm pump with a flow rate capacity of 35 GPM, which is powered by compressed air.

The air is provided by a 60 hp electric compressor, model LS100, with a maximum flow rate capacity of 1,200 scfm and maximum operating pressure of 120 psig. To measure the air flow rate, a Coriolis mass flow meter is used, which has a maximum flow rate capacity of 1,000 lbm/min. A Fisher pneumatic valve is used to automatically control the air flow rate. The figure 1 presents the schematic of the flow loop.
Figure 2 shows the upstream of the test section, which is a 0.0965 m Schedule-80 PVC pipe of 11 m length. The mixture of air and water is injected at the upstream of the test section. The air and water are injected from a 45° angle from the top of the pipe in order to create a smooth mixing and to avoid high turbulence and unexpected slugs. The test section is equipped with ultrasonic proximity sensor with a sensing range between 100 mm and 600 mm in order to measure the liquid level in the center of the 4-in. pipe. At the downstream of the test section, a GLCC© separator is utilized to separate the air and water. Air is released to the atmosphere, and the water is continuously recirculated to the water tank.

Figure 1—Schematic of the flow loop

1 GLCC© - Gas-Liquid Cylindrical Cyclone - Copyright, The University of Tulsa, 1994
Liquid Holdup

An ultrasonic sensor is installed at the top of the pipe (as shown in Fig. 2) to measure the liquid height in the center of the pipe. If the liquid holdup calculation is based on the liquid height alone it will not be accurate. In order to calculate liquid holdup accurately, the double circle model, as proposed by Yongqian (2005), is applied in this study. The model consists of two circles, the inner pipe wall circle and an imaginary eccentric circle. The gas-liquid interface is represented by the imaginary circle. When the diameter of the imaginary circle is very large, the gas-liquid interface approaches a flat configuration (Fig. 3), otherwise, the interface has a concave configuration.

Figure 2—Schematic of the test Section

Figure 3—Gas- Liquid interface with flat configuration

For the case of the flat gas-liquid interface, the relationship between the central angle, $\theta_o$ (radians), and the liquid holdup $H_L$, can be written as,

$$H_L = \frac{1}{\pi} \left[ \frac{\theta_o - \frac{1}{2} \sin(2\theta_o)}{2} \right]$$  (1)

When the gas-liquid interface has a concave shape (Fig. 4) the relationship between, $H_L$, $\theta_1$ and $\theta_2$ is calculated by

$$H_L = \frac{1}{\pi} \left[ \frac{\theta_1 - \frac{1}{2} \sin(2\theta_1)}{\sin^2 \theta_1} - \frac{\sin^2 \theta_2}{\sin^2 \theta_2} \left[ \theta_2 - \pi - \frac{1}{2} \sin(2\theta_2) \right] \right]$$  (2)
By measuring the liquid height at the center and the side of pipe, liquid holdup can be calculated easily based on the double circle model given by Equations (1) and (2) above.

![Figure 4—Gas-Liquid interface with concave configuration](image)

To ensure stratified flow, superficial liquid velocity of 0.1 m/s and superficial gas velocities ranging from 5 m/s to 13 m/s have been selected. The flow conditions of all experimental runs are plotted on a Taitel and Dukler (1976) flow pattern map. For all runs stratified wavy flow regime is observed.

![Figure 5—Taitel and Dukler (1976) flow pattern map](image)

**CFD Simulation**

The main objective of the study is to develop a CFD simulation to predict the liquid holdup for different flow conditions, and compare the results with experimental data. Since there is a symmetry in the tangential direction, half section of the pipe with the inner diameter of 0.096 m and a length of 3.4 m is
considered. To reduce two phase flow development length, the inlet plate is separated in two regions, where gas is introduced from the top region and liquid is introduced from the bottom region. Figure 6 presents the schematic of the inlet plate.

\[ V_{L,Inlet} = \frac{V_{SL} A_p}{2 A_{L,Inlet}} \]  \hspace{1cm} (3)

\[ V_{G,Inlet} = \frac{V_{SG} A_p}{2 A_{G,Inlet}} \]  \hspace{1cm} (4)

Water with default fluid properties is selected as the liquid phase, and air is chosen as the gas phase. The pressure boundary is selected as an outlet boundary and it is assigned as 0 psig. Volume of Fluid (VOF) with two Eulerian phases is applied as a multiphase flow model with explicit scheme in time. Realizable \( k-\epsilon \) (2-equation) is applied as a turbulent model with standard wall functions. The y-direction is used for gravitational acceleration given the default value.

The fluid flow is simulated with a transient method over a time period between 5 to 11 seconds for all cases, and run time depends on when the developing flow is achieved. The time step is specified based on Courant number, which is the most usual method to consider the stability of an explicit scheme. The initial value for the liquid holdup is selected to be 1.0 in order to shorten the fluid flow development. SIMPLE scheme is selected for pressure-velocity coupling.

Different results can be generated by using different discretization methods. The pressure, momentum, volume fraction and turbulent kinetic energy equations are discretized with PRESTO, second order upwind, geo-reconstruct and second order upwind respectively. The pipe meshing is developed with a commercial CFD code, and very fine mesh with the total number of 1,100,000 grid blocks is implemented. In Fig. 7, part (a) shows the schematic of inlet pipe meshing, and part (b) shows the schematic of pipe side meshing.
Validating the CFD Results

Several variables such as liquid holdup, liquid height, velocity profile in the gas and liquid regions, wall shear stress, interfacial shape, wave structure, wave amplitude and droplet entrainment can be obtained as the results of a CFD simulation. To simulate wave structure, wave amplitude and liquid droplet with CFD simulation, very fine mesh is required, and it requires high simulation run time. In this study, the variables acquired from CFD simulation are considered separately, and they are compared with the experimental data, where applicable. These variables include liquid holdup, liquid height, velocity profile, wall shear stress.

Liquid Hold up

The liquid holdup is defined as the ratio of the volume occupied by liquid phase to the total pipe volume. In order to obtain liquid holdup values from CFD simulation, a plate is created at the distance 0.7 m from the outlet, and due to waves created at gas-liquid interface, the liquid hold up on this plane is averaged over the time run.

Figure 8 presents the contours of volume fractions for CFD and experiment with five $V_{SG}$’s, where the $V_{SG}$ increases from left to right. The value of $V_{SL}$ for all cases is equal to 0.1 m/s. Part (a) presents the CFD result and nearly flat interface are observed for all cases, while contours in part (b) are scaled down to real experimental contour, where the interface shape plotted is based on measuring liquid height in pipe center and pipe side. Concave interface are observed for 5 cases, which is in disagreement with the CFD results. As can be seen, higher gas velocity creates higher drag force, reducing liquid level in the pipe, and it results in lowering the liquid holdup.
Taitel and Dukler (1976) model is the most applicable model used for characterizing stratified wavy flow in industry. The model is developed based on the combination of gas and liquid momentums, and the combined momentum is an implicit equation for the liquid level in the pipe. The model is applicable for steady state, fully developed, and Newtonian flow, and wettability effect is ignored. Gas-liquid interface is assumed to be flat, and the liquid holdup is calculated based on the liquid thickness. Figure 9 presents liquid holdup obtained from experiments, CFD simulation and Taitel & Dukler (1976) model, where the liquid holdup increases with lowering gas velocity. As can be seen, the CFD simulation predicts the experimental data well especially in the higher gas velocities while Taitel & Dukler (1976) model over predicts liquid holdup for all cases.
Centerline Liquid Height

In the experiment, the liquid in the center of the pipe is measured by ultrasonic sensor, which is installed at the top on the pipe. Taitel and Dukler model assumed that the gas-liquid interface is flat, and liquid height is calculated based on the combined momentum equation. Although this is a good assumption for low $V_{SG}$, but it has some uncertainties when the $V_{SG}$ value increases, and the flow pattern close to annular flow is observed where the interface has a concave shape.

To get liquid height values from CFD at the pipe centerline, a line is generated, which connects bottom of the pipe to top at the centerline. The time averaged liquid volume fraction on the line is considered as liquid height. Figure 10 shows the comparison between the experiment, CFD simulation and Taitel and Dukler (1976) model. There is good match between the experimental data and the CFD results especially at low gas velocities, but the Taitel and Dukler (1976) model over-predicts the liquid height.
Velocity Profile

The velocity profile for a single phase flow has a parabolic shape, while the velocity profile for two-phase flow is complex, and interfacial shear has a significant influence on the profile shape for each phases. The distribution of the gas and liquid stream-wise velocities on the vertical symmetry plane in the centerline are shown in Fig. 11 and Fig. 12 and the stream-wise velocity contours in test plane is presented in Fig. 13. Large equivalent roughness at interface causes high shear stress, thus as a result the maximum velocity of the gas phase is shifted to the region farther away from the gas-liquid interface and closer to the top pipe wall. With increasing gas velocity, the computed maximum velocity in the gas phase is shifted slightly downward, this can be the result of increasing the area occupied by the gas phase. In the liquid phase, the maximum velocity is observed at the interface, and the velocity profile has an S-shaped due to the effect of interfacial shear, which increases the liquid phase velocity.

Figure 10—Centerline liquid height results from experiments, CFD, and Taitel & Dukler model for \( V_{sl} = 0.1 \) m/s and \( V_{sg} = 5, 7, 9, 11, 13 \) m/s
Figure 11—Stream-wise velocity profile for gas and liquid phases for $V_{sl} = 0.1 \text{ m/s}$ and $V_{sg} = 5, 7, 9, 11, 13 \text{ m/s}$

Figure 12—Stream-wise velocity profile for liquid phase for $V_{sl} = 0.1 \text{ m/s}$ and $V_{sg} = 5, 7, 9, 11, 13 \text{ m/s}$
Figure 13 shows the stream-wise velocity gradient with respect to vertical position in the centerline for the highest and lowest gas velocities. For two cases, the velocity gradient near the top wall is steeper than near the gas-liquid interface. At $V_{SG} = 13$ m/s, the velocity gradient sharply changes at the interface due to higher interfacial shear. The maximum velocity gradient for $V_{SG} = 13$ m/s occurs at the lower height from the pipe bottom because of the lower liquid height compared to $V_{SG} = 5$ m/s.

![Streamline velocity contours for gas and liquid phases at the test plane](image13)

**Figure 14** shows the stream-wise velocity gradient with respect to vertical position in the centerline for the highest and lowest gas velocities. For two cases, the velocity gradient near the top wall is steeper than near the gas-liquid interface. At $V_{SG} = 13$ m/s, the velocity gradient sharply changes at the interface due to higher interfacial shear. The maximum velocity gradient for $V_{SG} = 13$ m/s occurs at the lower height from the pipe bottom because of the lower liquid height compared to $V_{SG} = 5$ m/s.

![Stream-wise velocity gradient for $V_{SG} = 13$ m/s and $V_{SG} = 5$ m/s](image14)

**Wall Shear Stress**

Fluid velocity and phase density can play important roles on the shear stress values in pipeline. Shear stress for a single phase flow has a uniform behavior, while existence of another phase makes it non
uniform. As can be seen in Fig. 15, wall shear stress values for five cases are almost constant in the gas phase, and they slightly decrease near the top wall. With decrease in the vertical position, the wall shear stress suddenly increases to the maximum value at the interface due to increase in fluid density, and then reduces in the liquid phase because of velocity reduction.

According to Taitel and Dukler (1976) model, there are three averaged shear stresses, which are applied into combined momentum equation: gas wall shear stress ($\tau_{WG}$), liquid wall shear stress ($\tau_{WL}$) and interfacial shear stress ($\tau_I$). Since there are no experimental data for $\tau_{WG}$ and $\tau_{WL}$, the results from CFD simulation are compared with the results obtained from Taitel and Dukler (1976) model. Wall shear stress at the top pipe is applied as a gas shear stress ($\tau_{WG}$), and the value at the pipe bottom is used as liquid shear stress ($\tau_{WL}$). Figure 16 shows that good agreement has been achieved between the Taitel and Dukler (1976) model and CFD simulation for gas wall shear stress, while there is poor agreement between CFD and Taitel and Dukler model (Fig. 17) for liquid wall shear stress at different gas velocities, because the liquid holdup is over-predicted by Taitel and Dukler (1976) model.
Figure 16—Gas Wall Shear stress for $V_{sl} = 0.1 \text{ m/s}$ and $V_{sg} = 5, 7, 9, 11, 13 \text{ m/s}$

Figure 17—Liquid Wall Shear stress for $V_{sl} = 0.1 \text{ m/s}$ and $V_{sg} = 5, 7, 9, 11, 13 \text{ m/s}$
Conclusions

CFD simulations are conducted using a commercial CFD code for 4-in. air-water horizontal pipeline, and the results such as liquid holdup, liquid height, wall shear stress, and velocity profiles are presented. The CFD results are compared with the experimental data acquired from 4-in. test loop. Furthermore, Taitel and Dukler (1976) model is applied for comparison.

The liquid holdup and centerline liquid height obtained from CFD simulation are in good agreement with the experimental results, and it is in poor agreement with the Taitel and Dukler (1976) model. The parameters such as velocity profile, wall shear stress are also considered. The wall shear stress for gas and liquid from CFD are compared with Taitel and Dukler (1976) model, and good agreement is achieved for the gas phase.

Although extensive experimental data is not provided to compare with the CFD simulations, by developing confidence in this type of simulations, and with additional model development work, it will possible to capture other phenomena and to determine the effect of other variables.

Nomenclature

- \( H_L \) = Liquid Holdup
- \( \theta_0 \) = Central angle with a flat interface, Radian
- \( \theta_1 \) = Central Angle with a concave interface, Radian
- \( \theta_2 \) = Angle shown in figure 4, Radian
- \( \tau_{WL} \) = Liquid Wall Shear Stress (Pa)
- \( \tau_{WG} \) = Gas Wall Shear Stress (Pa)
- \( \tau_I \) = Interface Shear Stress (Pa)
- \( A_P \) = Pipe Cross Sectional Area (m²)
- \( A_{G, Inlet} \) = Gas Inlet Area (m²)
- \( A_{L, Inlet} \) = Liquid Inlet Area (m²)
- \( V_{SG} \) = Superficial Liquid Velocity, m/s
- \( V_{SL} \) = Superficial Liquid Velocity, m/s

Acknowledgments

The authors wish to acknowledge the financial support of the Tulsa University Separation Technology Projects (TUSTP).

References


