Modeling and simulation of downward vertical two-phase flow with pipe rotation

Farhad Raeiszadeh a, b, Ebrahim Hajidavalloo a, b, *, Morteza Behbahaniejad a, Pedram Hanafizadeh c

a Mechanical Engineering Department, Shahid Chamran University of Ahvaz, Ahvaz, Iran
b Drilling Research Center, Shahid Chamran University of Ahvaz, Ahvaz, Iran
c School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran, Iran

A R T I C L E   I N F O

Article history:
Received 15 March 2018
Received in revised form 4 June 2018
Accepted 2 July 2018
Available online 10 July 2018

Keywords:
Flow regime
Downward two-phase flow
Ansly-Fluent
Pipe rotation

A B S T R A C T

One of the major problems in two-phase flow research is prediction of flow pattern at different configurations without requiring expensive experimental tests. Numerical modeling and simulation is a suitable and emerging approach for this purpose which has many benefits including saving in time and cost. In this study, for the first time, the effect of pipe rotation on the flow patterns of air–water two-phase flow in downward direction was numerically studied. For this reason, Eulerian–Eulerian multi-fluid approach was utilized in Ansys-Fluent software. At first, apparent conditions of each regime in various revolution were recorded from experimental tests and then the simulation was carried out to find if the results are compatible. It was shown that pipe rotation has important effects on the flow patterns map and shift transitions boundaries of slug and annular flow toward lower gas superficial velocity. Comparison of numerical results with experimental data show acceptable match for all regimes. Good agreement was observed for falling film, bubbly and slug regimes but the least agreement was observed for froth regime due to high turbulence and perturbation of flow.

© 2018 Institution of Chemical Engineers. Published by Elsevier B.V. All rights reserved.

1. Introduction

Gas-liquid two-phase flow has several applications in oil and gas industries, chemical processes, nuclear technology etc. One of the major industrial applications of two-phase flows is in pipes, which may happen in different slopes from horizontal to vertical situations. An important problem in the two-phase analysis of flow in the pipe is determination of exact flow pattern and gas and liquid distribution inside the pipes (Chorai and Nigam, 2006).

Since many of important design and engineering parameters such as pressure drop, mass transfer, heat transfer etc. are closely related to the phase distribution, therefore, two-phase flow patterns identification is an essential requirement in two-phase flow analysis (Bratland, 2009). Two-phase flow with pipe rotating condition can be found in many industries including drilling industry specially during underbalance operation in which exact prediction of pressure at bottom hole plays a crucial role in successful operation. Other application of this type of flow can also be found in plasma arc welding processes (off-shore industry) (Steinkamp and Mewes, 1994) and nuclear industry (impeller of cooling pump) (Wang et al., 2015). Although some studies have already investigated gas-liquid downward two-phase flow behavior, little study has examined the effect of pipe rotation on such flow conditions (Bratland, 2009). A few studies have examined flow patterns in annular space with a rotating inner tube (Ozbayoglu and Ozbayoglu, 2007; Ettehadi et al., 2013; Shioomi et al., 1993).

Flow patterns prediction and frictional pressure loss in two-phase flow through a horizontal annulus with a rotating inner pipe were studied by Ozbayoglu and Ozbayoglu (2007). They used an artificial neural network (ANN) approach to develop a mechanistic model and compared it with experimental data. Ettehadi et al. (2013) studied three-phase flow in inclined eccentric annuli and discussed on the characteristics of two-phase drilling fluid with cuttings as the third phase where inner pipe was in rotation. They showed that flow patterns...
are influenced by geometry and the presence of cuttings. Shiomi et al. (1993) studied two-phase flow in a concentric annulus with a rotating inner cylinder. They showed that increasing the rotating inner cylinder caused the bubbles to agglomerate and form a spiral flow. Flow visualization of a cross-section of the annulus showed that spirals or rings of bubbles formed very close to the rotating inner cylinder.

Literature review shows that most studies examining the effect of pipe rotation on the flow inside pipe investigated only in single-phase flow condition. Numerical study of viscous flow in rotating rectangular ducts was conducted by Speziale (1982), Reich and Beer (1989) examined the effects of pipe rotation on the characteristics of turbulent flow. They found that rotation has a marked influence on the suppression of turbulent motion from the radially-increasing centrifugal force. Imo et al. (1996) examined the effects of tube rotation on turbulent flow experimentally by using single-component Laser-Doppler velocimetry. They showed that the intensity of turbulence in the rotating pipe decreased gradually as pipe rotation increased due to stabilizing effect of centrifugal force and that momentum transfer caused by turbulence is strongly suppressed in a rotating pipe.

Review of previous research shows that flow patterns simulation of two-phase flow in a rotational pipe with downward direction has not been reported yet. Raeiszadeh et al. (2016) experimentally studied the effect of pipe rotation on the flow pattern of downward two-phase flow and found that rotational speed of the pipe has an important effect on the flow map and change the boundaries of different regimes considerably. They studied different flow pattern including bubbly, slug, froth and falling film regimes.

It seems, due to the complex nature of rotational two-phase flow and problems with stability and convergence of the solution, numerical simulation of this type of flows has not been developed so far. However, through the steady development of modeling approaches and computational capabilities, numerical simulation can be used in the study of rotational two-phase (Koobus et al., 2007).

One of the major weaknesses of the experimental methods used in two-phase flow studies is that flow-pattern maps obtained by this method are usually valid for a certain range of flow parameters and generalizing them to other situations or configurations are not always reliable and accurate and may raise possibility of error in the results, therefore, it needs many experimental set-ups to study different situations which is very expensive. So, by using validated numerical method, this problem can be overcome as different flow situations can be easily simulated using numerical method without requiring much time and cost (De Schepper et al., 2008). Therefore, for the analysis of two-phase flow in actual operating condition, reliable numerical simulation is much demanded requirement.

As mentioned, despite the considerable number of numerical research on two-phase flow, little work has been carried out on the rotational two-phase flow patterns prediction using computational fluid dynamics. In this research, numerical simulation of two-phase flow in downward vertical pipe with rotation was studied. This situation which may be found in many industrial applications has not been addressed yet. For this purpose, experimental data of two-phase flow in this situation were used and then by selecting the similar conditions of each flow pattern, numerical simulation was carried out for the same operating condition using Ansys-Fluent software. Finally, numerical results were compared with experimental results to validate the approach.

2. Geometrical configuration

To study the behavior of two-phase flow in downward vertical pipe, an experimental set-up, which already had been built, were used consisting a Plexiglas pipe with 50 mm ID and 4 m length (Fig. 1). Water and air were continuously fed into the mixer and then the mixture of air and water was entered into the vertical pipe from top (Raeiszadeh et al., 2016). The set-up had the ability to rotate the pipe about its axis at different speeds using an electric gearbox, a shaft and two gears. A high speed digital camcorder with 1/40,000 shutter speed was equipped for visualization, flow regime identification and flow recording. More information can be found in Raeiszadeh et al. (2016).

3. Mathematical modeling

There are various models for numerical simulation of multiphase flows. Since in Eulerian–Eulerian viewpoint, two phases are examined as a continuous phase, useful information can be obtained to analyze the results as well as the lower computational cost than the Eulerian–Lagrangian approach. Widely used multi-phase models of Eulerian–Eulerian viewpoint include the multi-fluid or Eulerian model, mixture model and VOF model (ANSYSFLUENT, 2015).

Numerical experiences obtained from this study shows that the best choice for downward two-phase flow modeling is multi-fluid model. This model was also used to simulate many multiphase flows (Stenmark, 2013).

3.1. Governing equations

In multi-fluid approach, the equations of continuity and momentum for the phase i are as follows, respectively (ANSYSFLUENT, 2015):

$$\frac{\partial}{\partial t}(\rho_i \mathbf{v}_i) + \nabla \cdot (\rho_i \mathbf{v}_i \mathbf{v}_i) = 0$$

(1)
Fig. 1 – Schematic of experimental set up.

For \( \vec{R}_{ji} \) modeling, Relation (3) is used:

\[
\frac{\partial}{\partial t} (\rho_i \vec{\phi}_i) + \nabla \cdot (\rho_i \vec{u}_i \vec{u}_i) = -\rho_i \vec{V} \cdot \vec{P} + \nabla \cdot F_i + \vec{F}_{\text{lift},i} + \vec{F}_{\text{VM},i} + \vec{F}_{\text{wl},i} + \vec{F}_{\text{td},i} + \vec{R}_{ji} \tag{2}
\]

\[
\sum_{p=1}^{n} \vec{R}_{ji} = \sum_{p=1}^{n} K_{ji} (\vec{v}_j - \vec{v}_i) \tag{3}
\]

In Eq. (2), \( \vec{n} \) is stress–strain tensor of phase i, and five terms, \( F_i, \vec{F}_{\text{lift},i}, \vec{F}_{\text{VM},i}, \vec{F}_{\text{wl},i}, \vec{F}_{\text{td},i} \) are external volume force, lift and virtual mass force, wall force and turbulence dispersion between two phases, respectively. \( \vec{R}_{ji} \) is the interaction force between phases and \( F \) is the pressure shared in all phases. Frank model is used for \( \vec{F}_{\text{wl},i} \) modeling and Bertodano model is used for \( \vec{F}_{\text{td},i} \) modeling.

In Eq. (3), \( K_{ji} = K_{ij} \) is momentum exchange coefficient between two phases which for fluid–fluid flows can be expressed by the following equation:

\[
K_{ij} = \frac{\alpha_i \rho_i f}{t_i} \tag{4}
\]
In Eq. (4) f is drag function. For drag function simulation is used from Schiller–Neuman model. In this model f function is defined as following:

\[
f = \frac{C_D \text{Re}}{24}
\]

(5)

\[C_D = \begin{cases} 24 \left(1 + 0.15 \text{Re}^{0.67} \right) & \text{Re} \leq 1000 \\ 0.44 & \text{Re} > 1000 \end{cases}
\]

(6)

Re is relative Reynolds number and for continues phase l (liquid) and dispersed phase g (gas) is expressed as:

\[\text{Re} = \frac{\rho_l |\vec{v}_l - \vec{v}_g| d_b}{\mu_l}
\]

(7)

In Eq. (4) \(\tau_i\) is particle relaxation time as:

\[\tau_i = \frac{\rho_i d_i^2}{18 \mu_i}
\]

(8)

In Eq. (8) \(d_i\), \(\rho_i\) and \(\mu_i\) are bubble diameter, gas density and liquid viscosity, respectively.

For multi-phase flows, Ansys-Fluent software considers the effect of lift forces on the particles (bubbles). Lift force operating on the second phase j in the initial phase of i, is calculated from the following equation:

\[\vec{F}_{ij} = -0.5 \rho_i \alpha_i (\vec{u}_i - \vec{u}_g) \times (\nabla \times \vec{u}_j)
\]

(9)

The effect of virtual mass force is important when the second phase j accelerates compared to the initial phase i. In other words, it is important when the density of the second phase is much smaller than the density of the initial phase. The initial mass of the initial phase due to the accelerated particles (bubbles) is applied as the virtual mass force:

\[\vec{F}_{i0} = -0.5 \rho_i \alpha_i \left( \frac{d \vec{u}_i}{dt} - \frac{d \vec{u}_g}{dt} \right)
\]

(10)

The term \(\frac{d \vec{u}}{dt}\) represents the time derivative of the phase. Stress–strain tensor of phase i is calculated from the following equation:

\[\tau_i = \alpha_i \left( \left[ \mu_i + \mu_{le} \right] \left( \nabla \vec{u}_i + (\nabla \vec{u}_i)^T \right) - \frac{2}{3} \alpha_i \left[ \mu_i + \mu_{le} \right] \nabla \delta - \rho_i \alpha_i \right]
\]

(11)

In Eq. (11), \(\delta\) is kronecker delta and \(\mu_{le}\) is turbulence viscosity of phase i as:

\[\mu_{le} = \rho_i C_{\mu} \frac{k^2}{\ell}
\]

(12)

In Eq. (12), \(C_{\mu} = 0.09.

In the case of external volume force \(\vec{F}_e\), there is only gravity force which is considered in operating condition.

To investigate the effects of turbulence in the flow, k-\(\varepsilon\) standard turbulence model was used. There are three models for modeling turbulence multiphase flows as follows: mixture, dispersed and per phase, for this study mixture model is used.

The governing equations for mixture turbulent kinetic energy (k) and dissipation of turbulence of mixture (\(\varepsilon\)) are shown in Relations (13) and (14):

\[\frac{\partial}{\partial t} (\rho_k k) + \nabla \cdot (\rho_k \vec{v}_m k) = \nabla \cdot \left( \frac{\mu_{le}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_k \varepsilon
\]

(13)

\[\frac{\partial}{\partial t} (\rho_k \varepsilon) + \nabla \cdot (\rho_k \vec{v}_m \varepsilon) = \nabla \cdot \left( \frac{\mu_{le}}{\sigma_\varepsilon} \nabla \varepsilon \right) + \frac{\varepsilon}{k} \left( C_1 \varepsilon G_{k,m} - C_2 \rho_k \varepsilon \right)
\]

(14)

In Eqs. (13) and (14), \(\rho_m\) and \(\vec{v}_m\) are density and velocity of mixture respectively as:

\[\rho_m = \sum_{i=1}^{N} \alpha_i \rho_i
\]

(15)

\[\vec{v}_m = \frac{\sum_{i=1}^{N} \alpha_i \vec{v}_i}{\sum_{i=1}^{N} \alpha_i}
\]

(16)

Production rate of turbulence kinetic energy, \(G_{k,m}\), was calculated as follows:

\[G_{k,m} = \mu_{le} \left( \nabla \vec{v}_m + (\nabla \vec{v}_m)^T \right) : \nabla \vec{v}_m
\]

(17)

Constants in these equations are similar to constants of single phase k-\(\varepsilon\) model and as follows:

\[C_1 = 1.44, C_2 = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3
\]

(18)

3.2. Numerical simulation approach

Three-dimensional flow simulations were performed on the same experimental geometry (50 mm diameter and 4 m length). According to the analysis of transient two-phase flow, time step was set at 0.001 s. It was assumed that there was no mass transfer between two phases. For pressure–velocity coupling, phase coupled algorithm was used and for discretizing momentum equation and two turbulence equations, first order upwind method were used and for discretizing
volume fraction equation, second-order upwind was used (Dakshinamoorthy et al., 2013; Parsi et al., 2016). Considering air and water, as working fluid, they entered in the pipe perpendicular to the inlet plane. Velocity inlet boundary condition and pressure outlet boundary condition were used for simulation. Air and water superficial velocities were considered as velocity input. The boundary conditions used in the simulation are the same as those were used in the experimental tests to facilitate validation process. Table 1 shows velocity boundary conditions which were used for experimental and

---

**Fig. 3** – Results of grid independence study.

**Fig. 4** – Effect of pipe rotation on the falling film regime ($V_{SG} = 0.04 \text{ m/s}$, $V_{SL} = 0.16 \text{ m/s}$).
Numerical simulations. Superficial phase velocities of water ($V_{SL}$) and air ($V_{SG}$) are defined as follows:

\[ V_{SL} = \frac{Q_L}{A_P} \]  
\[ V_{SG} = \frac{Q_G}{A_P} \]  

4. Results and discussion

For comparison of numerical and experimental results, a location of the pipe which is far from the entrance region was used for study which is shown as the “test section” in Fig. 2. At first, grid independency analysis was performed to ensure getting a proper solution. Volume fraction of flow versus radius of the pipe at a height of 1 m above the bottom of the tube were used for comparison at five grids with different number of cells. Considering the required accuracy and the costs of grid computing, it was found that grid with 800’000 cells is appropriate in the study as shown in Fig. 3.

The effect of pipe rotation on different flow regime are shown in Figs. 4–9. Fig. 4 compares experimental and numerical results for falling film regime at different pipe revolutions which shows good agreements. As seen, pipe rotation produced no special effect on the flow and as usual liquid phase moves close to the wall and air phase moves in the center. Fig. 5 compares experimental and numerical results in the bubbly flow regime. As seen, good agreement can be observed. In bubbly regime centrifugal force accumulates bubbles in the pipe center close to the tube axis. Increasing the revolution increases the centrifugal force so more accumulation of gas phase observed in the middle area. Increasing in liquid and gas flow rate, while keeping the bubble regime condition, decreases the accumulation of bubbles in the pipe axis as seen in Fig. 6. This means centrifugal force cannot overcome inertial force in this condition.

An interesting phenomenon was observed in Fig. 7, both in experimental and numerical results, in which accumulation of bubbles due to pipe rotation caused their coalescence and eventually the formation of a slug flow regime. This regime change from bubbly to slug regime usually happens where \( V_{SL} \leq 1 \text{ m/s} \). As seen, this flow pattern change due to the rotation can be predicted by numerical simulation which indicate the capability of numerical method for accurate prediction in two-phase flow.

Another interesting phenomenon can be observed in the slug flow regime in which pipe rotation caused the slug pock-
ets to coalesce, increasing their lengths to an extent which produce an annular regime as seen in Fig. 8a. This flow pattern change, due to pipe rotation, can also be predicted by numerical simulation as seen in Fig. 8b. Fig. 9 shows the effect of pipe rotation on the froth regime. The ratio of intense turbulence force compare to the centrifugal force decreases gradually as the rotational speed increases which cause the froth regime change to an annular regime as shown in Fig. 9. Good agreement can also be realized between experimental and numerical findings. Fig. 10 shows the cross-sectional view of air volume fraction at different flow regimes. As seen in Fig. 10a, in falling film water is located close to the wall and air is located in the middle of pipe which matches with Fig. 4. In bubbly regime (Fig. 10b) air and water contours are randomly distributed in the pipe cross-section which matches with Fig. 5. Fig. 10c shows volume fraction of phases in slug regime which indicates that slug is not located symmetrically in the pipe. In experimental photos, which was taken from the front view, however, slug bubble seems to be located in the middle of the pipe, whereas in the cross-sectional study using numerical simulation it was shown that is not the case. This is another advantage of numerical simulation compare to the experimental method where by showing different views of the flow exact results can be found. In froth regime, turbulence is high and for this reason, air and water contours are completely chaotic. There are air contours in front view, while these contours are not observed in cross sectional picture. It is for this reason that there was no air in cross section plot. This discrepancy is due to flow turbulence and perturbation.

Fig. 11 shows the cross-sectional view of air volume fraction for rotating pipe. In Fig. 11a falling film pattern is shown at 180 rpm. As previously mentioned (Fig. 4) pipe rotation has no effect on this configuration. In Fig. 11b rotation has caused the coalescence of bubbles and larger bubbles are formed. Rotation effect on slug pattern is caused that length of slug bubbles are increased and annular pattern is formed. This phenomenon is observed in Fig. 11c. In Fig. 11d pipe rotation decreases flow turbulence and froth regime is converted to annular regime.

As seen, good results can be observed using numerical approach, where those cannot be easily possible to obtain in experimental approach. This is another advantage of numerical approach which encourage more usage of this method in two-phase flow studies.

5. Conclusion

In this study the effect of pipe rotation on the air–water downward two-phase flow patterns was numerically examined by using Eulerian method in Ansys-Fluent software and was compared with experimental results. It was found that pipe rotation had a significant effect on two-phase flow patterns and numerical simulation can be used to predict this effect successfully. In bubbly regime, pipe rotation caused bubbles to accumulate around the axis of the pipe, which increased
the probability of their coalescence. At higher flow rates, water momentum overcame bubble tendency to gather in centerline and the bubbles remained dispersed, which created bubbly flow at lower air flow rates and froth flow at higher air flow rates. Depending on the air and water flow rate, the pipe revolutions change the flow regimes from bubbly to slug flow, from froth to annular and, at higher revolutions, from slug to annular flow pattern. As pipe rotation increases, the inten-
Fig. 9 – Effect of pipe rotation on the transition of froth to annular flow ($V_{SG} = 0.8$ m/s, $V_{SL} = 0.64$ m/s).

Fig. 10 – Air volume fraction contours from top view at test section for fixed pipe.
sity of turbulence force decreases compared to the centrifugal force which cause the froth zone decreases in size and in some revolutions the froth zone disappears between the slug and annular flow map.

In general, it was found that good agreement between simulation and experimental results indicates that the computational fluid dynamics is a capable tool in predicting downward two-phase flow patterns and can be used as an alternative approach instead of expensive experimental approach.

References

ANSYSFLUENT, 2015. 16.0 User's Guide. ANSYS Inc.