VOID FRACTION AND WAKE ANALYSIS OF A GAS-LIQUID TWO-PHASE CROSS-FLOW

P. Hanafizadeh,¹,* M. Momenifar,¹ A. Nouri Geimassi,² & S. Ghanbarzadeh³

¹Center of Excellence in Design and Optimization of Energy Systems, School of Mechanical Engineering, College of Engineering, University of Tehran, Iran
²Department of Mechanical Engineering and Material Science, University of Pittsburgh, Pittsburgh, PA, 15261, USA
³Department of Petroleum and Geosystems Engineering, The University of Texas at Austin, Austin, TX, 78712, USA

*Address all correspondence to P. Hanafizadeh E-mail: hanafizadeh@ut.ac.ir

Two-phase cross-flow takes place in a wide range of industrial equipment, including heat exchangers and measurement devices. The aim of this paper is to establish a numerical model and experimental methodology for comprehensive study and visualization of void fraction and wake region in gas-liquid cross-flow over immersed bodies with various cross-section geometries. Conservation of mass and momentum for both-phase free streams, along with constitutive relationships, were used for modeling turbulence. The input parameters for the numerical simulations were two-phase Reynolds number, free-stream void fraction, bubble size in the inlet, and cross-section geometry of prisms inserted in the two-phase flow path. Because the wake region and phase distribution around an immersed object are time-dependent, we report time average values of drag, lift, and pressure coefficients. The results show that drag and lift coefficients are strongly dependent on the two-phase Reynolds number; this dependency is a more moderate function of void fraction. The results are in good agreement with available empirical correlations and experimental work. Furthermore, experiments were conducted to visualize phase distribution and wake region in two-phase cross-flow. Comparison of the experimental and numerical results verifies the developed numerical model.

KEY WORDS: gas-liquid cross-flow, drag coefficient, flow regime, void fraction

1. INTRODUCTION

Multiphase flow occurs in a wide range of natural and industrial situations. Examples include boiling heat transfer, bubble columns, and reactors in the chemical industry; power plant cooling circuits; liquid fuel and paint spraying; emulsions; and rain, bubbles, and volcanic eruptions. Gas-liquid two-phase cross-flow occurs in many devices, such as boilers, condensers, and heat exchangers. Vortex shedding induced by immersed bodies...
causes vibration in such equipment. This phenomenon can have a significant effect on the long-term reliability and safety of industrial systems.

Two-phase cross-flow also occurs in most heat-exchanging compartments of power plants. In some cases, pressure drop in each compartment serves as a design parameter to increase efficiency and overall plant output. An extensive study of gas-liquid cross-flow seems essential to provide a better estimation of pressure loss in different of a plant compartments for different operating conditions. Although complex conduits with flow obstacles are widely employed in many industrial devices and their flow is often of the gas-liquid two-phase type, only a few studies of two-phase cross-flow have been reported in the literature (e.g., Ghanbarzadeh et al., 2009). More effort should be made to study two-phase flow around immersed bodies and to investigate the forces applied to external objects.

Yokosawa et al. (1986) measured the drag force on a single cylindrical tube in a two-phase cross-flow in the Reynolds number (Re) range of 4,000–300,000 and the low void fractions (0–0.1). It was shown that drag coefficient decreases with increasing void fraction when two-phase Re is sufficiently below the single-phase critical value (transition from laminar to turbulent flow around a cylinder: \( \text{Re}_c = 2 \times 10^5 - 5 \times 10^5 \)). For
Re above the critical value, the drag coefficient was found to gradually increase with increasing void fraction, although it remains significantly lower than the value of the subcritical single-phase flow. Yokosawa et al. also classified the flow patterns of two-phase wake flow behind a cylinder and quantitatively investigated the change in drag coefficient corresponding to the flow pattern transition.

Inoue et al. (1986) studied the characteristics of flow around an immersed cylinder located in a vertical upward air-water bubbly flow. This work shows how vortex flow and changes in static pressure and liquid velocity distributions around the cylinder distort void fraction distribution around the cylinder. In this work, void fraction distribution around the cylinder was measured with an electrical impedance probe inserted in the flow in the Re range of 5000–80,000. It was found that high void fraction regions (peaks or wakes), with the local void fraction approximately 3–4 times higher than the free stream, are generated in the vicinity of the cylinder surface near the separation point. It was also shown that as the mean velocity in the main flow increases, peak void fraction also increases and comes closer to the cylinder. Liquid layers were observed to become thicker in front and thinner at the rear of the cylinder as the mean flow velocity increases.

Joo and Dhir (1994) conducted a series of experiments to investigate drag coefficient in two-phase cross-flow based on the pressure distribution on a cylinder. The liquid Re was shown to be 430–21,900 for a single tube and the liquid gap Re to be 32,900–61,600 for a tube placed in a triangular array. The free-stream void fraction is 0–0.4. For all examined Re, the researchers observed a deficiency of voids in the region near the downstream stagnation point. At low Re, a gas-rich region was observed ahead of the forward stagnation point. However, this trend appears to reverse as Re increases. At low Re, the ratio of two-phase to single-phase drag coefficient was found to be a strong function of $\frac{Gr}{Re^2}$ ($2 \times 10^{-3} < \varepsilon \frac{Gr}{Re^2} < 4 \times 10^{-5}$ and $\varepsilon < 0.4$). However, at high Re void fraction is the most important parameter.

A limited number of numerical studies on two-phase cross-flow do not necessarily mean that there is a shortage of numerical models for simulation of two-phase flows. In fact, many interesting numerical studies have been carried out on two-phase flow simulation that mostly address vertical upward flow in two-phase gas-liquid systems. However, some interesting work has also been done on downward gas-liquid flow, including Artemiev and Kornienko (2002) and Zaichik et al. (2004). A numerical model developed by Artemiev and Kornienko (2002) takes into account the turbulence generated by the presence and motion of bubbles; it can be applied to both upward and downward vertical flows. Turbulent viscosity is represented as a linear combination of two terms, one due to liquid-phase turbulence, which can be calculated by the Reichardt formula (1943), and the other due to relative gas motion. The second term in turbulent viscosity has some empirical constants, which restrict the developed numerical model to special cases.

Zaichik et al. (2004) proposed a diffusion-inertia model for the transport of low-inertia particles of arbitrary density. The $k-\varepsilon$ turbulence model for single-phase flow was used to calculate turbulent viscosity. Originally developed for gas-dispersed flows, this
model is applicable to bubbly flows. Polydispersed two-phase flows were studied by Carrica et al. (1999) and Politano et al. (2003). The models proposed in these studies shift the maximum bubble concentration from the near-wall zone toward the flow core, observed when the dispersed-phase size increases above some critical value. Troshko and Hassan (2001) developed a numerical model involving the law of the wall for vertical monodispersed bubbly flow in a pipe.

For predicting liquid-phase turbulence, Lopez et al. (1994), Carrica et al. (1999), Politano et al. (2003), Troshko and Hassan (2001), and Lee et al. (1989) used a two-equation turbulence model that extends to two-phase flows. Lucas et al. (2009) established a detailed database on air-water flow in a vertical pipe for a wide range of flow rates by taking measurements using wire-mesh sensor technology. Despite intensive recent numerical investigation of downward gas-liquid flows, reported data are insufficient and cover only a narrow range of parameters; thus they can be applied only in particular conditions.

Image processing has become a powerful technique for studying two-phase flow phenomena and verifying numerical models. Hasanein et al. (1996) described a visualization technique based on video recordings of flow (with bubble length estimation achieved by direct comparison with on-site rulers). Polonsky et al. (1999) used image processing to study the motion of individual Taylor bubbles. BuiDinh and Choi (1999) investigated the application of digital image processing in two-phase flows, and Fore et al. (2002) measured droplet size in nitrogen-water two-phase annular flow using video images. Gmu and Heiskanen (2002) measured bubble size in flotation machines using digital imaging, which Van et al. (2002) applied to continuous slug flow (although only to a small number of bubbles). Sousa et al. (2006) developed a three-dimensional (3D) photographic method to measure phase distribution in dilute bubble flow. Hanafizadeh et al. (2011a) used image analysis to visualize the air-water vertical upward two-phase flow pattern in the riser pipe of an airlift pump. Hanafizadeh et al. (2011b) also used this technique for experimental detection of gas-liquid two-phase flow regimes in a vertical mini-pipe. Finally, Ghanbarzadeh et al. (2012) used image processing coupled with a fuzzy logic model to predict flow regime and pressure drop in vertical two-phase flow, particularly gas-liquid cross-flow.

The purpose of the present study was the numerical and experimental investigation of gas-liquid cross-flow over immersed bodies with different cross-sectional geometries. The numerical model was based on a Eulerian representation for both phases and was optimized to study the structure of flow fields and wake zones behind immersed objects in monodispersed gas-liquid flow. In addition to immersed body geometry, the effects of bubble size, flow stream void fraction, and two-phase Re on local flow quantities were evaluated, also, air-water cross-flow around rigid bodies was investigated experimentally. Flow images were captured by a high-speed camera in a vertical pipe. The image-processing technique included background removal, morphological operation, and adaptive segmentation (see Ghanbarzadeh et al. 2010a and 2010b for details).
2. METHODOLOGY

2.1 Numerical Approach

In the Euler/Euler approach, the different phases were treated mathematically as interpenetrating continua. Mass and momentum conservation equations for each phase were derived to obtain a set of equations having the same structure for each phase (Dhotre and Joshi, 2007; Lehr et al., 2002; Simonin, 1990). Because of an unequal number of unknowns, the resulting system of equations was not closed. Using empirical equations as constitutive relations enabled the system to be closed. Coupling was achieved through pressure and interphase momentum exchange (interphase mass exchange was not included in this study). The manner in which this coupling was handled depended on the phases involved. Momentum exchange between phases was also dependent on the type of mixture being modeled. There are extensive empirical correlations for different cases in the literature (e.g., Prosperetti and Tryggvason, 2009).

Volume fraction for each phase, \( \alpha_q \), represents the space that phase occupies, and the conservation of mass is satisfied once the sum of \( \alpha_q \) for all phases is constant. The phase volume fraction may not be constant because the gas phase is treated compressible. Hanafizadeh et al. (2013) derived mass conservation equations for each phase. For phase \( q \) the volume \( (V_q) \) is

\[
V_q = \int_V \alpha_q dV
\]

(1)

where

\[
\sum_{q=1}^{n} \alpha_q = 1
\]

(2)

The effective density is

\[
\hat{\rho}_q = \alpha_q \rho_q
\]

(3)

And the continuity equation is

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q \vec{v}_q) = \sum_{p=1}^{n} (\dot{m}_{pq} - \dot{m}_{qp}) + S_q
\]

(4)

where \( \dot{m}_{pq} \) is the volumetric mass transfer rate between phases and \( S_q \) is the source term. In this study both terms were considered zero.

The momentum balance for phase \( q \) yields

\[
\frac{\partial}{\partial t} (\alpha_q \rho_q \vec{v}_q) + \nabla \cdot (\alpha_q \rho_q \vec{v}_q \vec{v}_q) = -\alpha_q \nabla p + \nabla \cdot \vec{\tau}_q + \alpha_q \rho_q \vec{g}
\]

\[
+ \sum_{p=1}^{n} \left( \vec{R}_{pq} + \dot{m}_{pq} \vec{v}_p - \dot{m}_{qp} \vec{v}_q \right) + \vec{F}_q + \vec{F}_{lft,q} + \vec{F}_{vm,q}
\]

(5)

where \( \vec{g} \) is the acceleration due to gravity, \( \vec{\tau}_q \) is the phase stress-strain tensor, \( \vec{F}_q \) is an external body force, \( \vec{F}_{lft,q} \) is the lift force, \( \vec{F}_{vm,q} \) is a virtual mass force, \( p \) is the pressure
shared by all phases, $\vec{R}_{pq}$ is the interphase force, and $\vec{v}_{pq}$ is the interphase velocity. Comprehensive details and empirical correlations for these terms are beyond the scope of this work; readers are referred to Hanafizadeh et al. (2010) for more information.

Because of the continuous motion of the gas phase in the liquid phase and instantaneous mixing of the phases in two-phase flow, turbulence is present in all Re, despite the single-phase flow case. Additionally, as a result of the static pressure gradient near the immersed body, intensive turbulence is generated near stagnation points and the momentum transported from the main flow to the boundary layer becomes fairly large. In this study, we used the $k-\varepsilon$ turbulence model for determining the fluctuations in flow parameters due to turbulence. This model is appropriate when the concentrations of the secondary phases are dilute. In this case, interparticle collisions are negligible and the influence of primary-phase turbulence is dominant in the random motion of the secondary phases. Fluctuating quantities in the secondary phases can therefore be given in terms of the mean characteristics of the primary phase and the ratio of particle relaxation time and eddy-particle interaction time.

The most general case of the multiphase $k$-model solves a set of $k$ and $\varepsilon$ transport equations for each phase. This model is appropriate when momentum transfer among the phases plays a dominant role.

The Reynolds stress tensor and turbulent viscosity are computed as follows:

$$\bar{\tau}_{pq}'' = -\frac{2}{3} \left( \rho_q k_q + \rho_q \mu_{t,q} \nabla \vec{U}_q \cdot \nabla \vec{U}_q + \rho_q \mu_{t,q} \left( \nabla \vec{U}_q + \nabla \vec{U}_q^T \right) \right)$$  \hspace{1cm} (6)

$$\mu_{t,q} = \rho_q C_{\mu} \frac{k_q^2}{\varepsilon_q}$$  \hspace{1cm} (7)

Turbulence parameter calculations are obtained from

$$\frac{\partial}{\partial t} \left( \alpha_q \rho_q k_q \right) + \nabla \cdot \left( \alpha_q \rho_q \vec{U}_q \cdot \nabla k_q \right) = \nabla \cdot \left( \alpha_q \frac{\mu_{t,q}}{\sigma_k} \nabla k_q \right) + \left( \alpha_q G_{k,q} - \alpha_q \rho_q \varepsilon_q \right)$$  \hspace{1cm} (8)

$$+ \sum_{l=1}^{N} K_{lq} \left( \alpha_q \rho_q \alpha_l \sigma_l \nabla \alpha_l \right) + \sum_{l=1}^{N} K_{lq} \left( \frac{\mu_{t,l}}{\alpha_l \sigma_l} \nabla \alpha_l \right)$$

and

$$\frac{\partial}{\partial t} \left( \alpha_q \rho_q \varepsilon_q \right) + \nabla \cdot \left( \alpha_q \rho_q \vec{U}_q \cdot \nabla \varepsilon_q \right) = \nabla \cdot \left( \alpha_q \frac{\mu_{t,q}}{\sigma_k} \nabla \varepsilon_q \right) + \frac{\varepsilon_q}{k_q} \left[ C_{1\varepsilon} \alpha_q G_{k,q} \right.$$  \hspace{1cm} (9)

$$- C_{2\varepsilon} \alpha_q \rho_q \varepsilon_q + C_{3\varepsilon} \left( \sum_{l=1}^{N} K_{lq} \left( \alpha_q \rho_q \alpha_l \sigma_l \nabla \alpha_l \right) - \sum_{l=1}^{N} K_{lq} \left( \frac{\mu_{t,l}}{\alpha_l \sigma_l} \nabla \alpha_l \right) \right)$$

$$+ \sum_{l=1}^{N} K_{lq} \left( \frac{\mu_{t,q}}{\alpha_q \sigma_q} \nabla \alpha_q \right) \bigg]$$

The terms $C_{lq}$ and $C_{ql}$ can be approximated by
\[ C_{lq} = 2, \quad C_{ql} = 2 \left( \frac{\eta_{lq}}{1 + \eta_{lq}} \right) \] (10)

Complete details for computing \( \eta_{lq} \) are available in Simonin et al. (1990) and Csanady (1963). The discretization of derivatives in the developed numerical model is based on the control volume framework proposed by Patankar (1980). In this method, a collocated grid is used and all variables are stored at the center of the control volume. The discretized governing equation is solved using the SIMPLE algorithm. The time-dependent equations are solved to increase the stability of the numerical solution. For any iteration, two continuity and two momentum equations with the transport equations of turbulent energy and dissipation are solved. Turbulent variables and velocity near the wall of the control volume were estimated from the wall laws. Velocities of both phases were calculated from the respective momentum equations (by means of actual velocity).

After velocity was determined, pressure was computed from the liquid continuity equation. Volume fraction was calculated from the continuity equation of the gas phase. Full details of the numerical algorithm and the discretization procedure can be found in Hanafizadeh et al. (2010).

### 2.2 Mesh Generation and Geometry

To simulate two-phase flow around an infinite-length immersed body, a two-dimensional (rectangular) computational domain was used. The boundary conditions on the edges of the computational domain were set as velocity inlet, two walls, and outflow. The wall boundaries fall on two opposite sides of the flow channel, and their distance is set far enough to eliminate zero velocities on walls and other boundary layer effects on the flow pattern around the immersed body. This consideration can minimize error and provide the fully developed flow condition around the prism. Figure 1(a) shows the channel geometry and the immersed body along with the generated mesh. The boundary mesh in the vicinity of the bluff body was magnified to investigate mesh details. Figure 1(b) depicts the domain, the distances between boundaries, and the location of the boundary conditions. Boundary conditions were set according to Table 1. The internal square was created in the domain to generate the structure mesh around the cylinder and to increase numerical stability and accuracy. A longer domain length behind the external object enabled capturing the wake region more accurately and avoiding the transfer of fully developed flow in the outlet to regions near the immersed body.

Grid independency of the results was verified using different grid sizes with different Re and immersed body cross-sectional shapes. For the circular cylinder with \( D = 10 \) mm, five meshes (35,000 quadrilateral cells, 50,000 triangular cells, 60,000 triangular cells, 75,000 quadrilateral cells, and 95,000 quadrilateral cells) were tested. Figure 2 compares drag coefficient versus number of nodes for different two-phase Re (Re\(_{TP}\)) while the inlet void fraction was kept constant in all cases. The figure shows that the drag coefficient in all examined Re is independent of the number of grids once there
are more than 750,000 nodes in the domain. The same procedure was carried out for different cylinder diameters and prism side lengths.

2.3 Experimental Approach

To verify the numerical results and visualize the volume fraction distribution around immersed bodies, experiments were carried out in air-water vertical upward two-phase flow. Visualization of phases and the wake region provides insight on the fundamental characteristics of two-phase cross-flow. The examined external bodies in a two-phase cross-flow were a cylinder and a square prism with a diameter and side length of 10 mm.
TABLE 1: Implemented boundary conditions

<table>
<thead>
<tr>
<th>Number</th>
<th>Boundary condition</th>
<th>Type</th>
<th>Equation(s)</th>
</tr>
</thead>
</table>
| 1      | Inlet                  | Velocity inlet | $V_f^x = V_g^x = V_{in}$  
|        |                        |             | $V_f^y = V_g^y = 0$  
|        |                        |             | $P = 0$  |
| 2      | Outlet                 | Outflow     | $\frac{\partial V_f^x}{\partial x} = \frac{\partial V_g^x}{\partial x} = 0$  
|        |                        |             | $\frac{\partial V_f^y}{\partial y} = \frac{\partial V_g^y}{\partial y} = 0$  
|        |                        |             | $\frac{\partial P}{\partial x} = \frac{\partial P}{\partial y} = 0$  |
| 3      | Channel walls          | Wall        | No slip     |
| 4      | Immersed body wall     | Wall        | No slip     |

FIG. 2: Validation of mesh independency.

For visualization, a high-speed camera (1200 fps, 1/40,000-s shutter speed) was used to take instantaneous images from the flow which were then analyzed via image processing (Hanafizadeh et al., 2011a; Ghanbarzadeh et al., 2012).

The experiments on two-phase flow in this study were conducted in a large-scale test facility. The experimental loop with a corresponding apparatus is shown schematically in Fig. 3. The centrifugal pump, via the strainer, pumped water from the main tank. The water flow rates were regulated by two globe valves and were measured by a calibrated magnetic flowmeter with an accuracy of 0.5%. A compressor continuously
fed the compressed air (up to 6 bars) to the system. The airflow rate was set constant in each experiment and was measured by the calibrated gas turbine flowmeter with an accuracy of ±1%. Air and water were mixed in a plenum made of acrylic glass placed at the bottom of the riser pipe. Compressed air was injected at the plenum by a porous stainless steel plate with 108 holes of 0.5-mm diameter. The overall height and internal diameter of the riser pipe were 6 m and 50 mm, respectively. To facilitate observation of two-phase flow patterns, the riser pipe was made of a transparent acrylic glass, in which the water was moved upward by compressed air; the water and the air were then
separated in the separation tank at the top of the riser. The air was discharged to the atmosphere and the water was returned to the main tank or to the airlift tank depending on the test application, and was ultimately circulated back into the loop. Water temperature was kept constant at the ambient condition. Sixteen pressure transmitters measured the pressure distribution of flow with 0.3% accuracy at different positions along the riser.

All of the aforementioned instruments had their own specified signals that were scaled and fed to a rapid wideband PCI-6255 data acquisition card. The recorded data were stored for postprocessing. The test rig was equipped with a high-speed digital camcorder (1,200 fps) at a height of 5 m on the riser, to the rig provided water volumetric flux at 0.1–5 m/s and air volumetric flux at 0.01–10 m/s. Volumetric flux is the volumetric flow rate divided by the flow area; it is also called superficial velocity.

3. RESULTS AND DISCUSSION

The numerical simulations of two-phase cross-flow were carried out for the circular cylinder with 5-, 10-, and 20-mm diameters, aspect ratios of 0.5, 1, and 2 for the rectangular prism, and leading angles of 45, 60, and 75 degrees for the triangular prism. The liquid phase (primary) was water, the gas phase (secondary) was air, and the void fraction range varied 0.01–0.2 in the numerical experiments (as input). Another input parameter was bubble diameter at the inlet, which was considered constant and equal to 1 mm. The reason for this choice was that in experiments the average bubble size was approximately 1 mm.

The two-phase Re (Re\textsubscript{TP}) is defined as

\[
\text{Re}_\text{TP} = \frac{\alpha \rho_g + (1 - \alpha) \rho_f}{\alpha \mu_g + (1 - \alpha) \mu_f} \frac{vd}{v_d} \quad (11)
\]

and changes from 10\textsuperscript{3} to 10\textsuperscript{5}. The pressure coefficient, \(C_P\), describes dimensionless pressure drop throughout a flow field and is defined by

\[
C_P = \frac{P - P_\infty}{(1/2) \rho_\infty V_\infty^2} \quad (12)
\]

where \(P\) is the pressure at the point where pressure coefficient is evaluated and \(P_\infty, \rho_\infty,\) and \(V_\infty\) are free-stream pressure, density, and velocity, respectively. The drag coefficient is equal to the sum of the friction drag coefficient and the pressure drag coefficient; it is defined as

\[
C_D = C_{D,friction} + C_{D,pressure} = \frac{F_{\text{Drag}}}{(1/2) \rho_\infty V_\infty^2} \quad (13)
\]

Because the local flow direction changes around a body, two-phase cross-flow is affected significantly by fluid inertia. Therefore, quite different flow behavior between the gas and liquid phases is induced because of their large density difference and so void
fraction distribution around the body changes markedly. Together, these changes result in a different trend in drag and lift forces around the immersed body. To verify the numerical results, drag coefficient was compared with experimental measurements reported by Yokosawa et al. (1986). As Fig. 4 shows, the predictions are in good agreement with experimental data, and it seems reasonable to conclude that the model can accurately solve the two-phase flow field around immersed bodies.

Figure 5 shows the lift coefficient, $C_L$, versus Re$_{TP}$ for the cylinder in air volume fractions of 0.1 and 0.2. As can be seen, $C_L$ increases with Re because of stronger phase mixing and phase pressure and velocity fluctuations. Higher Re essentially means a smaller wake region and a stronger Karman vortex street behind the cylinder. Simulation results also show that the increase in void fraction decreases the lift coefficient because of decreases in both effective flow density and buoyancy effects. As vortex shedding and pressure and velocity fluctuations in phases are time-dependent, time average values of drag, lift, and pressure coefficients were used to generate related plots.

Figures 6(a) and 6(b) show circular pipe drag coefficient versus Re$_{TP}$ for various diameters of the immersed cylinder (5 mm, 10 mm, and 20 mm, respectively). Increasing Re decreases drag. When Re is small, the flow behaves like a potential flow: there is no separation and drag is all due to skin friction. As Re increases, this drag decreases and the boundary layers separate; however, the wake is of a limited length and eddies behind the cylinder are fixed in place. As Re continues to increase, the eddies break off from the cylinder and continuously shed from the body and wash downstream. Two rows of vortices are formed, known as the Kármán vortex vortex street. Because of the smaller wake region with higher Re, drag coefficient continues to decrease.
FIG. 5: Variation in lift coefficient versus Reynolds number.

Figures 6(a)–6(c) also show the effect of inlet void fraction on drag coefficient. As expected, this effect is limited on lift coefficient. At low Re, drag coefficient is not a strong function of void fraction because the surface of the immersed body is covered with a layer of liquid and the skin friction remains same and independent of the amount of gas in the flow. However, as the Re increases, the void fraction effect starts to show up in effective flow density and pressure drag. A higher void fraction means both a lower effective density and a larger separation region with low-density gas that increases pressure drag on the immersed body.

To clarify the reasons for drag coefficient change with void fraction, a variation in pressure drag coefficient with Re is plotted in Fig. 7 for the same cases shown in Fig. 6. As observed, pressure drag coefficient decreases with Re, whereas the change with inlet void fraction is more complex. However, in the most cases an increase in void fraction increases pressure drag. As the increase continues, the high-gas region behind the cylinder grows in volume and pressure drag on the immersed body increases.

Variations in pressure coefficient around the cylinder are presented in Figs. 8(a) and 8(b) for two void fractions, 0.1 and 0.2. Each figure shows the changes in pressure coefficient for various $Re_{TP}$. It is clear that pressure coefficient decreases with increased Re. Two-phase flow pressure behavior is compatible with its single-phase counterpart. As can be seen, increasing Re moves the separation point from near 90° to approximately 70° and decreases the wake area behind the cylinder. A decrease in $C_p$ with Re behind the cylinder shows that pressure drag drops as Re increases. A closer comparison of Figs. 8(a) and 8(b) shows that $C_p$ behind the cylinder (at 180°) generally decreases as void fraction increases, which essentially increases pressure drag.
FIG. 6: Variation in drag coefficient with Reynolds for cylinder diameters (a) 5 mm, (b) 10 mm, and (c) 20 mm.
FIG. 7: Variation in pressure coefficient with Reynolds for cylinder diameters (a) 5 mm, (b) 10 mm, and (c) 20 mm.
FIG. 8: Variation in pressure coefficient with angle for cylinder diameter 10 mm and void fractions (a) $\alpha = 0.1$ and (b) $\alpha = 0.2$.

Figures 9(a)–9(c) show pressure coefficient around a cylinder in the various void fractions for $Re = 10^3$, $10^4$, and $10^5$, respectively. It is obvious that an increase in void fraction decreases pressure. This decrease can be attributed to the aggregation of bubbles behind the cylinder. Bubbles move to the lower-pressure region at the back of the cylinder and increase the pressure there. This also can be justified with the general trend
FIG. 9: Variation in pressure coefficient with angle for cylinder diameter 10 mm and (a) $Re = 10^3$, (b) $Re = 10^4$, and (c) $Re = 10^5$. 
of the pressure drag term as a function of void fraction. A decrease in pressure coefficient increases the pressure difference on opposite sides of the cylinder and results in higher pressure drag.

Variations in void fraction around the cylinder in air-water two-phase flow are presented in Fig. 10 for inlet void fractions of 0.05, 0.1, and 0.2, respectively. It is clear that most of the bubbles gather at the back of the cylinder because of the relative vacuum and negative pressure there (note the change in $C_p$). Increasing Re moves the separation point to the front of the body (around 70°) and brings the gas forward; it also diffuses to the wake at the back of the cylinder. The void fraction finds its minimum value very near the lower stagnation point and then grows along the tube circumference. It achieves the maximum value between 120° and 150°, and then decreases as far as the downstream stagnation point. Regions occupied by the liquid phase are thus located both upstream and downstream of the tube. The peak void fractions occur in the separation point regions by static pressure gradient, lift, and centrifugal forces.

Void fraction distribution around the cylinder is shown in Fig. 11 for various Re. As expected, an increase in inlet velocity and hence Re extends the gas spread in the wake region behind the cylinder. Two separate high-gas regions are easily detectable in all cases. Negative pressure around and behind the cylinder induces high-gas regions and decreases effective density in the wake region. This considerable gradient in density distribution exerts additional forces on the immersed body, in what is called the bubble-pumping effect.

The present study also analyzed two-phase cross-flow over a prism with rectangular and triangular cross sections. The respective variations in drag coefficient for these cases are shown in Figs. 12(a) and 12(b). In both cases, an increase in Re decreases drag, which is in accordance with the single-phase flow trend. As shown in Fig. 12(a), the rectangular cross section with the greater aspect ratio has the greater drag coefficient. In the case of the square cross section (aspect ratio = 1), friction drag is lower because of the smaller surface area exposed to flow. Thus, with low Re, total drag coefficient in the square body is less than that in the rectangular body with aspect ratios of 0.5 and 2. As the free-stream flow velocity increases, the effect of friction drag decreases and is replaced by that of pressure drag. In this case, the square has the largest wake region pressure drop around the surface and thus a larger drag coefficient. For the triangular cross section, drag coefficient is plotted as a function of Re for various leading angles in Fig. 12(b). The results show that an increase in leading angle increases drag coefficient; it also results in a sharper change in geometry behind the triangle and gives the immersed body a far from streamlined profile.

Figure 13 shows the variation in Strouhal number (St) versus Re in inlet volume fractions of 0.05 and 0.1. The results are obtained from flow simulation around a cylinder. The value of St is defined as

$$St = \frac{fL}{V}$$

(Multiphase Science and Technology)
FIG. 10: Variations in void fraction with angle for cylinder diameter 10 mm and inlet void fractions (a) inlet $\alpha = 0.05$, (b) inlet $\alpha = 0.1$, and (c) inlet $\alpha = 0.2$. 

Volume 26, Number 4, 2014
FIG. 11: Contour of void fraction around the cylinder for (a) $Re_{TP} = 1,000$, (b) $Re_{TP} = 3,162$, (c) $Re_{TP} = 10,000$, (d) $Re_{TP} = 31,623$, and (e) $Re_{TP} = 100,000$.

where $f$ is the frequency of vortex shedding, $L$ is the characteristic length (diameter of the cylinder), and $V$ is the velocity of the free-stream flow (two-phase velocity). To determine the frequency of vortex shedding, the Fast Fourier transform (FFT) of lift coefficient versus time was calculated and the dominant frequency was considered the shedding frequency. The result shows that $St$ increases with $Re$. Two regions of vortex shedding are seen in Fig. 13: one below a critical $Re$ near 150, where the speed of vortex generation increases with increasing $Re$; the other is after the critical $Re$, where the vortices are formed; here an increase in $Re$ does not much change the frequency but does increase vortex shedding and the size of the wake region behind the cylinder. Also, an increase in inlet void fraction slightly decreases cylinder $St$. As volume fraction increases in the inlet, the gas-dominant region behind the cylinder grows and decreases the chances that vortex shedding will form a wake region.

3.1 Flow Visualization

Figures 14(a) and 14(b) visualize air-water two-phase flow around a cylinder with a diameter of 10 mm for $Re$ 2,400, 4,900 and 44,000, respectively. Bubbles accumulate at the separation point on the cylinder walls, and as $Re$ increases the wake region behind
the body grows. Figures 15(a) and 15(b) visualize the two-phase flow field around a square with an edge of 10 mm for Re 2,400, 4,900 and 44,000, respectively. They also show that increased bubble velocity results in an accumulation of bubbles in the backside wake region. It should be noted that void fraction differs between figures. Free-stream void fraction increases linearly with increased inlet gas velocity.

Figure 16 compares the two-phase flow field with the numerical simulation results of void fraction for a square cylinder. Although the numerical model is based on dispersed gas-phase distribution and cannot distinguish an explicit gas-liquid interface, the average gas-phase distribution of is in good agreement with experimental observation. The figure illustrates how the wake dissipates behind the body at a distance to downstream.
FIG. 13: Contour of void fraction around the cylinder for (a) $Re_{TP} = 1,000$, (b) $Re_{TP} = 3,162$, (c) $Re_{TP} = 10,000$, (d) $Re_{TP} = 31,623$, and (e) $Re_{TP} = 100,000$.

FIG. 14: Air-water two-phase flow around a cylinder for various (a) $Re = 2400$, (b) $Re = 4,900$, and (c) $Re = 24,000$.

Image processing was used to examine the details of the two-phase flow around the immersed bodies (Hanafizadeh et al. 2011a). The original RGB images were converted to grayscale to remove the background and then, with an adaptive thresholding method, the grayscale images were converted to binary (segmented). Morphological processes were applied to the segmented images to fill and smooth the bubble shapes. The result is shown in Fig. 17, where it can be seen that two-phase flow in the upstream of the cylinder has an almost uniform bubble distribution. Also, visible is a region with high bubble density at either side of the bodies and the formation of wakes and vortices behind them. Based on numerical simulations, a reverse flow might be induced in some parts of
FIG. 15: Air-water two-phase flow past a square cylinder for (a) $Re = 2,400$, (b) $Re = 4,900$, and (c) $Re = 24,000$.

FIG. 16: Comparison of numerical and experimental results for two-phase flow around a square cylinder.

FIG. 17: Final result of image processing of two-phase flow: (a) cylinder; (b) square cylinder.
the wake region. In the vicinity of the rear of the cylinder, reverse flow is limited by the wall effect and zero velocities on the cylinder surface. Thus, the bubbles cannot enter the rear of the cylinder and, as a result, the surface of the immersed body is covered with a liquid-rich layer.

4. CONCLUSIONS

Gas-liquid two-phase cross-flow is seen in many industrial applications, such as boilers, condensers, and shell-and-tube heat exchangers. In such equipment, performance, durability, and safety strongly depend on pressure drop, internal forces exerted on immersed pipes, and flow-induced vibrations. In this study, a previously developed numerical model was used to simulate the flow field around various immersed objects. Problem inputs were two-phase Reynolds number, gas void fraction at the inlet, size and geometry of the immersed objects, and gas bubble size at the inlet. The validity of the model and the results were verified by comparing observed drag coefficient values with reported experimental values. Parameters of interest were void fraction distribution, drag, lift and pressure coefficients (which characterize the flow pattern around a body), and Strouhal number (which characterizes the vortex-shedding pattern at the rear of the external body).

The results show that the overall behavior of drag, lift, and pressure coefficient versus Reynolds number is the same as in single-phase flow. Pressure coefficient decreases as void fraction increases, especially at the separation point. This decrease decreases pressure drag while the void fraction increases in free-stream. Also, increasing velocity, or two-phase Reynolds number, causes bubbles to disperse into the wake behind the immersed body. In the case of rectangular and triangular bodies, an increase in inlet void fraction decreases drag coefficient. Drag coefficient with a small Reynolds number is the smallest for the square prism; it decreases with decreasing leading angle for the triangular prism.

Phase distributions from the numerical simulations were compared with experimental visualization of two-phase cross-flow over the cylinder and the square prism. The accumulation of bubbles in low-pressure regions and the downstream dissipation of the wake region along with gas-phase volume fraction plumes were verified.

REFERENCES


