Numerical simulation of two-phase flow in airlift pumps using the Physical Influence Scheme

Pedram Hanafizadeh and M.H. Saidi*

Center of Excellence in Energy Conversion,
School of Mechanical Engineering,
Sharif University of Technology,
P.O. Box 11155-9567, Tehran, Iran
E-mail: hanafizadeh@mech.sharif.edu
E-mail: saman@sharif.edu
*Corresponding author

Masoud Darbandi

Centre of Excellence in Energy Conversion,
Department of Aerospace Engineering,
Sharif University of Technology,
P.O. Box 11155-9567, Tehran, Iran
E-mail: darbandi@sharif.edu

Azade Kebriaee

Centre of Excellence in Energy Conversion,
School of Mechanical Engineering,
Sharif University of Technology,
P.O. Box 11155-9567, Tehran, Iran
E-mail: kebriaee@mech.sharif.edu

Abstract: A new approach has been presented to solve gas-liquid flow numerically in vertical pipes of air-lift pumps. To improve modelling, a new strategy has been employed with the capability of coupling the continuity and momentum equations and enforcing the role of pressure directly in the continuity equation. This is achieved via applying a novel scheme called the Physical Influence Scheme (PIS). The current finite volume solution is compared with other available numerical solutions. Indeed, they are in fair agreement. However, the present predictions are far superior to those obtained from an existing simple method, which is widely used in airlift pump modelling.

Keywords: airlift pump; two-phase flow; finite-volume method; PIS; physical influence scheme; two-fluid approach.


Biographical notes: P. Hanafizadeh is pursuing his PhD candidate at the Centre of Excellence in Energy Conversion at the School of Mechanical Engineering, Sharif University of Technology.

M.H. Saidi is a Professor and Chairman of the School of Mechanical Engineering, Sharif University of Technology. His current research interests include heat transfer enhancement in boiling and condensation, modelling of pulse refrigeration, vortex tube refrigerators, indoor air quality and clean room technology and energy efficiency in home appliances, desiccant cooling systems.

M. Darbandi is a Professor in the Department of Aerospace Engineering and a Faculty Member of the Centre of Excellence in Energy Conversion. His special interest is in Designing, Simulating, Optimising, and improving the energy conversion systems from the macroscales to micro/nanoscales.
1 Introduction

The airlift pump lifts liquid, using from the buoyancy force created by air bubbles. This pump was invented by Carl Loscher at the end of 18th century, and has been numerically modelled to elaborate its characteristics over the last several years (Giot, 1982). The idea of using airlift pumps has been practiced to lift solid particles as well. This type of pump involves a vertical pipe consisting of two parts. A suction pipe is between the bottom end and the port air injection and an upriser pipe between the air port and discharge port and it is partially submerged at the bottom, see Figure 1. The flow is in liquid phase in the suction part; while it is two-phase in the upriser. Although this pump has low efficiency in comparison with other types of pumping, it provides several important advantages compared to ordinary pumping. One main advantage is that the cavitation is avoided in airlift pumps. Therefore, it is possible to place the pump in a pressurised capsule and leave it under the water surface. Another advantage is its simplicity in manufacturing because of the absence of moving mechanical parts. This benefit makes the pump ideal for some special applications such as those needed for working in corrosive and viscous flow applications. Although the external airline systems are more efficient, the internal airline pumps are used more because of their versatility and the ease of assembling. Therefore, the simple and accurate simulation of airlift pumps is essential to determine their optimum operational working conditions.

The literature shows that various correlations (e.g., Zenz, 1993) and simple mathematical models (e.g., Kato et al., 1975; Casey, 1992; Nenes et al., 1996a; Hanafizadeh and Saidi, 2008) have been developed to achieve this important purpose. Additionally, there are experimental and analytical approaches to model these types of pumps in three phase flow (Kassab et al., 2007). In recent decade with improving computers, numerical methods are widely used to determine the characteristic of two phase flow (Zhou et al., 2006). All these methods are based on mixture (Cocchi and Saurel, 1997) or two-fluid approaches (Karni et al., 2004).

The advantages of finite-volume-based methods have promoted many workers to practice this approach in computational fluid dynamics. Generally, there are two basic choices to treat the fluid flow governing equations. One choice is the type of primary dependent variables utilised in the computational algorithm and the other one is the relative locations of the dependent variables on the computational grids. The primitive finite-volume work of Patankar and Spalding (1972), which is known as SIMPLE, considers the continuity equation as a constraint equation to compute the pressure. However, SIMPLE and its early revisions suffered non-physical oscillatory pressure and velocity field solutions (e.g., Darbandi and Hosseinizadeh, 2003, 2006; Javadi et al., 2008). The staggered grid arrangement was assessed and utilised as a general remedy to overcome the encountered drawbacks. This type of grid stores the dependent variables at two different locations, which are displaced with respect to each other. Boundary condition implementation difficulty and excessive book-keeping are two major objections to staggered grid approaches. In addition, the velocities that satisfy mass do not necessarily conserve momentum in the same control volume. From two-phase flow applications, Markatos (1993) and Nenes et al. (1996b) developed the basic SIMPLE method to a suitable modified version to model to treat the single-phase and two-phase flow problems in the industrial cases. They reduced the primitive drawbacks encountered with SIMPLE algorithm. Also Galea and Markatos (1991) presented a three dimensional mathematical field model to simulate fire development in an aircraft cabin. He used a modified procedure in accordance with SIMPLEST algorithm of Spalding for solution and implemented a Body Fitted Coordinate (BFC) formulation to describe correctly the curvi-linear interior geometry of the aircraft.

This paper presents a novel collocated finite volume formulation to model two-phase flow in airlift pumps. The collocated grid has many benefits, see Darbandi and Bostandoost (2005). The formulation utilises the PIS to predict gas and liquid velocities along the pipe. The analysis is focused on the upriser part. In this regard, we solve the flow governing equations and apply suitable boundary and initial conditions. A physically acceptable solution is obtained when the absolute differences between the amount of pressures in two successive time steps are equal, and this represents the convergence criterion of the model.
2 The governing equation

As shown in Figure 1, the airlift pump includes a vertical pipe divided into suction and upriser parts. Hence, the analysis of flow is arranged in two steps and the governing equations are written for these two parts separately. The flow in the suction pipe is single phase and is simulated by applying the Bernoulli equation considering one extra term to include the pressure drop in the suction. In this paper, we concentrate on analysing the upriser part because of its great complexity and find the velocity and pressure profiles for a two-phase flow condition.

Flow in the upriser pipe is in two-phases, i.e., a mixture of water and air. It is customary to present the performance of the newly developed methods via developing a one-dimensional model. At least, it helps to avoid the entrance region close to the inlet at vertical pipe. The entrance region introduces two singularities at the inlet of pipe and makes the solution uncertain (Darbandi and Schneider, 1998c; Darbandi and Hosseinizadeh, 2004; Darbandi and Vakilipour, 2007). Similar to the work of Markatos and Singhal (1982) and Markatos (1986), we also treat the one-dimensional unsteady flow, considering the variation of properties by including the variation axis defined along the upriser. For simplicity, it is assumed that the operating conditions are steady and the flow in both phases is isothermal. By neglecting the effect of compressibility, the continuity equation is extended to (Markatos and Singhal, 1982)

\[
\frac{\partial R_i}{\partial t} + \frac{\partial}{\partial z}(RU_i) = 0
\]  

(1)

where \( R \) and \( U \) are volumetric fraction and velocity, respectively. The subscript \( i \) denotes either \( g \) as the gaseous phase or \( l \) as the liquid phase. In the momentum equation, we consider the exchange of momentum between the two phases only via interphase friction process assuming that pressure is the same for both phases. Considering the above, the momentum equation can be written as

\[
\frac{\partial}{\partial t}(R \rho U_i) + \frac{\partial}{\partial z}(R \rho U_i^2) = -R \frac{\partial \rho}{\partial z} + f_i^{\text{wall}} + f_i^{\text{int}} - R \rho g
\]

where \( P \) and \( g \) are pressure and gravity acceleration. Additionally, \( f_i^{\text{wall}} \) is the phase wall friction and \( f_i^{\text{int}} \) is the interphase friction. The latter is replaced with \( f_i^{\text{int}} \) for the gaseous phase and \( f_i^{\text{int}} \) for the liquid phase. The interphase friction source terms, i.e., \( f_i^{\text{wall}} \) and \( f_i^{\text{int}} \) should always satisfy the following relation:

\[
f_i^{\text{wall}} = -f_i^{\text{int}}.
\]

(3)

In the absence of any solid obstacle, the volume fractions at every point must satisfy one constraint as follows:

\[ R_g + R_l = 1. \]

(4)

At the injection port \( i \), the flow is only in the liquid phase, so the boundary conditions at this point are defined as

\[
\begin{align*}
R_g &= 0.75 \quad U_g = 0.8 \\
R_l &= 0.25 \quad U_l = 0.8
\end{align*}
\]

(5)

At the discharge, the pressure \( p_2 \) is equal to the atmospheric pressure and a free outflow boundary condition is implemented there as follows:

\[
\frac{\partial \Phi}{\partial z} = 0 \quad \text{where} \quad \Phi = R, U \quad \text{and} \quad i = l, g.
\]

(6)

To close the governing equations, we need to calculate the friction terms. The friction coefficient depends on the flow regime and the fluid properties. In this work, we employ the experimental relation of bubbly and slug flow to calculate the friction terms.

3 Domain discretisation

Figure 2 illustrates the grid distribution along the upriser. The control volumes are located between the crosses, which are called integration points. The grid nodes, shown by circles, are located at the geometric centre of the control volumes. The subscripts \( N \) and \( S \) are used to denote the nodal quantities associated with the control volume to the north and south of the control volume centred at node \( P \). Similarly, \( n \) and \( s \) indicate the north and south faces of the control volume centred at point \( P \). For more details, see Darbandi and Bostandoost (2005) and Darbandi et al. (2008).

Figure 2  The nomenclature used for the cell faces and their neighbouring cells

4 Computational modelling

The continuity equation is integrated over an arbitrary control volume, shown in Figure 2, over a time interval of \( \Delta t \). Applying the Gauss divergence theorem, the integration yields

\[
\int_{C.V} \left( \int_{t}^{t+\Delta t} \frac{\partial R_i}{\partial t} \, dt \right) dV + \int_{f_i} \left( \int n_i(R_i U_i) \, dA \right) dt = 0
\]

(7)
where $A$ and $\forall$ represent the cell face and volume, respectively. The outward unit normal vector to each cell face is shown by $n$. The current method is fully implicit in time. Contrary to the SIMPLE algorithm, which results in a few sets of tridiagonal, linear algebraic equations, this method results in one block tridiagonal system of linear algebraic equations for all the two-phase governing equations, and they are solved simultaneously. Considering the current implicit method, the steady term is evaluated at the advanced time and the transient term is approximated using a backward difference in time. If considering a mass lumped approach in each volume the transient term is reduced to

$$\int_{C_{i,j}}^{t+\Delta t} \left( \int n. (R, \rho U_j^z) \, dA \right) \, dt = \left( \int n. (R, \rho U_j^z) \, dA \right)_{t+\Delta t} - \left( \int n. (R, \rho U_j^z) \, dA \right)_{t} \Delta t \, A_{cell}. \quad (8)$$

where the superscripts ‘old’ and ‘new’, respectively, refer to the magnitudes at time $t$ and time $t + \Delta t$. $R_j^{old}$ indicates the volume fraction of phase $j$ in cell $P$ where magnitude is approximated from the preceding iteration and $z$ is the length of control volume. The integration of the second term in Equation (7) yields:

$$\int_{C_{i,j}}^{t+\Delta t} \left( \int n. (R, \rho U_j^z) \, dA \right) \, dt = \left[ \left( R, \rho U_j^z \right)_{t+\Delta t} - \left( R, \rho U_j^z \right)_{t} \right] \Delta t \, A_{cell}. \quad (9)$$

Assuming a constant cell face, if Equations (8) and (9) are substituted into Equation (7) we get

$$\Delta \rho R_j^{new} + \left[ \left( R, \rho U_j^z \right)_{t+\Delta t} - \left( R, \rho U_j^z \right)_{t} \right] \Delta t = \bar{R}_j^{old} \Delta \rho. \quad (10)$$

We apply a simple linearisation $\bar{R}_j^{old} U_j^{new}$ to discretise the continuity equation. The bar over $R$ means that it is approximated from the preceding iteration. Therefore, the linearised mass conservative statement is given by:

$$\Delta \rho R_j^{new} + \left[ \bar{R}_j^{old} U_j^{new} - \bar{R}_j^{old} U_j^{old} \right] \Delta t = \bar{R}_j^{old} \Delta \rho. \quad (11)$$

Similarly, the momentum equation can be integrated over the same control volume and the time interval of $\Delta t$. By replacing the volume integral of the convective term with the surface integral, we obtain

$$\int_{C_{i,j}}^{t+\Delta t} \frac{\partial}{\partial t} (R, \rho U_j^z) \, dA \, dt + \int_{C_{i,j}}^{t+\Delta t} n. (R, \rho U_j^z) \, dA \, dt = \int_{C_{i,j}}^{t+\Delta t} -\frac{\partial p}{\partial x} \, dA \, dt + \int_{C_{i,j}}^{t+\Delta t} \bar{f}^j \, dA \, dt + \int_{C_{i,j}}^{t+\Delta t} \bar{f}^j \, dA \, dt - \int_{C_{i,j}}^{t+\Delta t} \bar{f}^j \, dA \, dt \quad (12)$$

where $\bar{f}^j$ is the force on phase $i$ due to phase $j$. The use of backward difference for the transient term yields

$$\int_{C_{i,j}}^{t+\Delta t} \frac{\partial}{\partial t} (R, \rho U_j^z) \, dA \, dt = \left[ (R, \rho U_j^z)_{t+\Delta t} - (R, \rho U_j^z)_{t} \right] \Delta A \, A_{cell}. \quad (13)$$

Additionally, the integration of the convection term over the cell faces yields

$$\int_{C_{i,j}}^{t+\Delta t} \left( \int n. (R, \rho U_j^z) \, dA \right) \, dt = \left[ \left( R, \rho U_j^z \right)_{t+\Delta t} - \left( R, \rho U_j^z \right)_{t} \right] \Delta t \, A_{cell}. \quad (14)$$

Using the divergence theorem, the pressure term can be calculated from

$$\int_{C_{i,j}}^{t+\Delta t} \left( \int -\frac{\partial p}{\partial A} \, dA \right) \, dt = \int_{C_{i,j}}^{t+\Delta t} \left[ (P)_{t+\Delta t} - (P)_{t} \right] \Delta t \, A_{cell}. \quad (15)$$

The friction force is also integrated over the volume of the computational cell for phase $i$. It yields (Markatos and Singhal, 1982)

$$F_{iw} = \int f_{iw} \, dV = \int 0.5 f_{iw} (R, \rho U_j^z) A_{new} \Delta t \, A_{cell} \quad (16)$$

where $f_{iw}$ is the friction coefficient, and $U_j$ and $A_{new}$ are the area of contact between wall and the $i$th phase in the current cell. The friction coefficient $f_{iw}$ is calculated from the Blasius equation as follows:

$$f_{iw} = 0.079 Re^{0.25} \quad (17)$$

the magnitude of $A_{new}$ depends on the flow regime. The expression for the slug flow is given by:

$$F_{iw} = 4 R_{s}/D \quad (18)$$

where $D$ is pipe diameter. The interphase friction $f_{ij}^z$ is calculated using a linear expression of velocity as follows

$$F_{ij} = \int \bar{f}_{ij} \, dA = C_{ij} \left( U_i - U_j \right) \Delta A, \quad (19)$$

where $C_{ij}$ is interphase friction coefficient, which depends on the fluid and flow properties of the two phases. For the slug flow regime, the interphase friction coefficient is calculated from

$$C_{ij} = \frac{3}{8} C_D \rho \, R_s \, |U_i - U_j|. \quad (20)$$

The approach of taking the modulus of the relative velocity ensures that the force on phase $i$ always acts in the correct direction. For example, if phase $j$ moves faster than phase $i$, the friction force on phase $i$ would accelerate it. In above equation, $R_s$ is the average radius of bubbles, which is calculated from

$$R_s = \frac{0.06147}{2 \rho |U_i - U_j|^2} \quad (21)$$

where We is the critical Weber number and $\sigma$ is the surface tension. Additionally, the drag coefficient $C_D$ is calculated from an empirical correlation given by

$$F_{ij} = 3 \frac{3}{8} C_D \rho \, R_s \, |U_i - U_j|.$$
\[ C_p = \frac{24}{R_{ep}} \left(1 + 0.15R_{ep}^{0.657}\right) + \frac{0.42}{1 + 4.25 \times 10^5 R_{ep}^{1.16}} \]  
(22)

where \( R_{ep} \) is the particle Reynolds number, it is defined as

\[ R_{ep} = \frac{2\rho_m \rho_l}{\mu} \left| U_i - U_j \right| \]  
(23)

The last term in the momentum equation can be approximated by

\[ \int_{\Delta t} \left( \int_{L_i} R_{\rho g} g dV \right) dt = (R_{\rho g})^{\text{new}} \Delta Z \Delta t. \]  
(24)

The substitutions of Equations (13)–(24) into Equation (12) and considering a constant cell face area, it eventually results in

\[ \frac{1}{\Delta t} \left[ (R_{\rho g} U_i)_{\text{new}} \Delta z + \left[ (R_{\rho g} U_i^2)_{\text{new}} - (R_{\rho g} U_i^2)_{\text{new}} \right] \right] 
+ \frac{1}{\rho_p} \left[ (P^e)_{\text{new}} - (P^e)_{\text{new}} \right] = 0.5 \mu_{\eta} R_{\rho g} U_i^{\text{new}} A_{\text{in}} \Delta z 
+ C_{\eta} \left( U_{\text{new}} - U_{\text{old}} \right) \Delta z 
- (R_{\rho g})^{\text{new}} \Delta z + \frac{\Delta Z}{\Delta t} \left(R_{\rho g} U_i \right)^{\text{old}}. \]  
(25)

A fully implicit method in time is applied to solve the set of our non-linear equations. Contrary to explicit methods, the implicit methods are not restricted by the Courant number magnitude. However, to be able to use a linear algebraic solver in our implicit algorithm, the non-linear terms in the discretised momentum equation are required to be linearised properly. Since the liquid and gas phase densities, \( \rho_l \) and \( \rho_p \), are constant, the sophisticated Newton Raphson Linearisation Scheme (NRLS) (see Tannehill et al., 1997; Darbandi and Schneider, 1998b; Darbandi and Mokarizadeh, 2004; Darbandi and Bostandoost, 2005; Darbandi et al., 2008), leads to a simple linearisation scheme as follows:

\[ \rho u = \bar{\rho} u + \pi \rho - \bar{\pi} \bar{\rho} \]  
(NRLS)

\[ \rho u = \bar{\rho} u. \]  
(26)

If we implement a simple linearisation scheme to treat the convection term, it yields

\[ \left[ \rho_{\text{in}} \bar{U}_{\text{in}}^{\text{old}} U_{\text{in}}^{\text{new}} - \rho_{\text{ip}} \bar{U}_{\text{ip}}^{\text{old}} U_{\text{ip}}^{\text{new}} \right] 
+ \frac{\rho_{\text{ip}}}{\Delta t} U_{\text{ip}}^{\text{new}} \Delta z + \bar{U}_{\text{ip}}^{\text{old}} \left[ \rho_{\text{ip}} - \rho_{\text{ip}} \right] 
= 0.5 \rho_{\text{ip}} \bar{U}_{\text{ip}}^{\text{old}} A_{\text{in}} U_{\text{ip}}^{\text{new}} + C_{\eta} \left( U_{\text{ip}}^{\text{new}} - U_{\text{ip}}^{\text{old}} \right) \Delta z 
- \rho_{\text{ip}} \rho_{\text{ip}} \Delta z + \frac{\rho_{\text{ip}}}{\Delta t} \bar{U}_{\text{ip}}^{\text{old}} \Delta z. \]  
(27)

By this statement, the discretisation of the governing equations is complete. Now the next step is to approximate the magnitudes at the cell faces in term of the magnitudes at the nodes. It is because the set of algebraic equation is not well-posed due to inconsistent number of unknowns appeared at the cell faces and cell centres.

5 The implementation of Physical Influence Scheme

To make the algebraic system of equations well-posed, this stage of our modelling requires to present the major dependent variables at cell faces in terms of nodal variables. Therefore, it is necessary to derive suitable expressions for the velocity component and pressure at the cell faces. Upwind, QUICK, and HYBRID are few schemes, which can be selected as suitable mathematical interpolations to achieve this goal (Versteeg and Malalasekera, 1995). Alternatively, there are more advanced schemes, which exhibit more physics of flow. Schneider and Raw (1987) employed PIS. They utilise the flow governing equations to derive the integration point expressions in incompressible flow simulations. Darbandi and Schneider (1997, 1998a, 2000) extended this model to compressible flow simulations. This scheme is further extended in this work, using the fundamental concepts of PIS to treat two-phase flow cases. The expressions for the velocity components can be derived by suitable discretisation of the momentum equation. In this regard, the momentum equation is expanded to

\[ \frac{\partial}{\partial t} (R_{\rho g} U_i) + \frac{\partial}{\partial z} \frac{\partial}{\partial z} (R_{\rho g} U_i^2) + \frac{\partial}{\partial z} \frac{\partial}{\partial z} = S_n \]  
(28)

where \( S_n \) is a source term, which includes the rest of terms in the momentum equation. Since the densities of two phases are constant, the above equation can be reduced to

\[ \frac{R_{\rho g}}{\partial t} U_i + R_{\rho g} U_i^2 + R_{\rho g} \frac{\partial p}{\partial z} = \frac{S_n}{\rho_i}. \]  
(29)

Following the essence of PIS, the terms in this equation are differenced in manners to respect the correct physics of flow. To achieve this purpose, they are approximated using either upwinding or central difference schemes as follows:

\[ \frac{R_{\rho g}}{\partial t} U_i \left|_{n+1} \right. = \frac{R_{\rho g}}{\partial t} U_i \left|_{n} \right. - \frac{U_i^{\text{mid}}}{\Delta t} \]  
(30)

\[ R_{\rho g} \frac{\partial U_i}{\partial z} \left|_{n+1} \right. = R_{\rho g} \frac{U_i}{\Delta z} \left|_{n} \right. - U_i^e \]  
(31)

\[ R_{\rho g} \frac{\partial p}{\partial z} \left|_{n+1} \right. = R_{\rho g} \frac{P_p - P_p}{\Delta Z} \]  
(32)

Here, the convection terms are treated in an upwind manner. Evidently, if they are treated in central difference manner, it results in poor formulations (Versteeg and Malalasekera, 1995). Another possible form for the velocity component and pressure at the cell faces is

\[ R_{\rho g} \left( \frac{\partial U_i}{\partial z} \right) \left|_{n+1} \right. = \frac{R_{\rho g}}{\partial t} \left( \Delta z \right) U_i. \]  

The reason for avoiding this scheme is the possibility of low stability of the method. The substitutions of the discretised terms into Equation (29) and its rearrangement result in an expression for the momentum component at integration point. The compact form of this expression is written as
$U_w = C_1 U_{wp} + C_2 (P_p - P_n) + S'_w.$ \hfill (33)

The constants $C_1$ and $C_2$ and the source term $S'_w$ are defined as

$$C_1 = \frac{2 \overline{R}_m \overline{U}_m \Delta t}{\overline{R}_m \Delta Z + 2 \overline{R}_m \overline{U}_m \Delta t}$$

$$C_2 = \frac{\overline{R}_m / \rho \Delta t}{\overline{R}_m \Delta Z + 2 \overline{R}_m \overline{U}_m \Delta t} \hfill (34)$$

$$S'_w = \frac{\overline{R}_m \overline{U}_m \Delta Z + S_{uw} / \rho \Delta Z \Delta t}{\overline{R}_m \Delta Z + 2 \overline{R}_m \overline{U}_m \Delta t}.$$

Equation (33) indicates that the use of PIS produces a strong connection between the integration point variable at face $n$ and its neighbouring nodal variables, i.e., $P$ and $N$. Similarly, the cell face velocity at $s$ is written as

$$U_w = C'_1 U_{wp} + C'_2 (P_s - P_n) + S''_w. \hfill (35)$$

The constant and the source term are defined very similar to Equation (34). It can be shown that the substitutions of $U_{si}$ and $U_{di}$ in the mass Equation (11) and momentum Equation (27) provide reliable coupling between pressure and velocity fields. These treatments of the continuity and momentum equations eventually result in a set of blocked-tridiagonal linear algebraic equations. It can be presented by

$$A\Phi_{i-1} + B\Phi_i + C\Phi_{i+1} = D \hfill (36)$$

where $A$, $B$, $C$ are the matrix coefficients.

6 The solution procedure

The solution of the above set of equations is achieved via implementing an implicit approach in time. The procedure is given by

a. Define the fluid and gas properties and choose a suitable finite volume grid with an appropriate time step.

b. Define the initial condition of the flow variables in both phases.

c. Solve the system of mass and momentum equations with Equation (4) simultaneously.

d. Apply boundary condition given by Equations (5) and (6).

e. Solve the system of linear algebraic equation numerically using the Three Diagonal Matrix Algorithm (TDMA) and obtain the values of variables at the same time step.

f. Check the residuals. If they are not sufficiently small, go to step (c).

The residual calculation in step (f) can be defined on the change of pressure magnitude between two consequent time intervals.

7 Result

The simulations have been carried out for the sets of following data using both numerical and integral models. The equivalent diameter of pipe is $1.826 \times 10^{-2}$ (m) and the length of the pipe is 9.823 m. Moreover, the fluid properties of the liquid phase are $\rho_l = 733.212$ kg/m$^3$, $\mu_l = 9.193 \times 10^{-3}$ Ns/m$^2$ and $\sigma = 1.459 \times 10^{-2}$ N/m. The density and viscosity coefficients for the gas phase are $\rho_g = 38.75$ kg/m$^3$ and $\mu_g = 1.94 \times 10^{-5}$ Ns/m$^2$.

To start the solution, uniform flow properties are specified as the initial condition, considering a volumetric gas fraction of $R_g = 0.25$, a volumetric liquid fraction of $R_l = 0.75$, and liquid and gas velocities of $U_l = U_g = 0.8$ m/s. During the calculation, the inlet conditions are fixed while the outlet is allowed to vary, subject to the condition that the total volumetric rate is constant.

At the first stage, we verify our formulation by solving the inviscid flow through a simple converging-diverging nozzle. The nozzle area is given by

$$S = 1.398 + 0.347 \tanh(0.8x - 4). \hfill (37)$$

Figure 3 shows the pressure and velocity distributions in the nozzle. They are in full agreement with the analytical solutions. The rate of convergence is dramatically high because the PIS scheme provides strong coupling between the pressure and velocity variables in the set of discretised governing equations. Indeed, the case of the nozzle is converged just within one step, benefiting from this scheme. Figure 4 shows the amount of inaccuracy involved in our solution. Using the error of code, which is depicted, shows that the accuracy of code is the second-order. The mesh independency of the solution was down for 20, 50, 100, 200 and 300 nodes along the length. The results show that CFL value increases with mesh size reduction for constant accuracy and it remains constant after 200 nodes. So the 200 nodes were applied along the length with CFL = 1.1. The second step is to solve the bubbly and slug regimes. In this regard the transient flow calculations are continued until the steady state condition is reached. The steady state results are shown in Figure 5. As expected, considering the constant mass flow rate, the gas velocity increases and, consequently, the gas volumetric fraction decreases. The buoyant force affects the gas phase and causes increase of its velocity, while the gravity forces decreases the liquid phase velocity. The increase of gas velocity and decrease of liquid velocity, the interphase velocity slip $[U_g - U_l]$ increases and, thus, the interaction force between the two phases increase. This description justifies why the inertial term in the momentum equation vanishes and the role of this term is shifted to the interaction friction term. Furthermore, the pressure decreases along the pipe and performs a hydrostatic pressure distribution. In this work, the pressure gradient is also calculated using the integral method (Nenes et al., 1996a) and it is compared with the CFD simulation of Markatos and Singhal (1982), see Figure 6. The results show good agreement with the reference pressure distributions. Figures 7 and 8 compare the current
liquid and gas velocities with those of Markatos and Singhal (1982). These comparisons show that there are good behaviour agreements among them in most cases. In spite of being aware that the PIS scheme is a physical scheme, it is very important to have a good perception of the physics of each equation. This can help to implement a proper boundary condition and get correct result.

**Figure 3** Pressure and velocity distribution in converging-diverging nozzle problem

**Figure 4** Logarithm of error vs. logarithm of maximum cell number

**Figure 5** Axial variation of flow properties in the steady state solution for bubble flow

**Figure 6** Comparison of pressure distribution with integral method and numerical model of Markatos and Singhal

**Figure 7** Comparison of liquid phase velocity with that of Markatos and Singhal

**Figure 8** Comparison of gas phase velocity in pipe with that of Markatos and Singhal

**8 Conclusion**

A new PIS was properly developed to simulate two-phase flow and it was applied to the airlift pump problem. The current results were compared with another numerical solution and the results of analytical integral method. The results show that the extended PIS method is suitable
Numerical simulation of two-phase flow in airlift pumps using the Physical Influence Scheme

for treating incompressible flow; however, the volumetric fraction plays an important role instead of density in two-phase flow. Therefore, we were encouraged to use this method in incompressible two phase flow with good outcomes. The PIS method permits the inclusion of the role of pressure in the continuity equations in two phase flow applications and to couple these equations with the momentum equation. The current formulation performs correct overall behaviour consistent with another numerical method. However, it was shown that the current pressure predictions are better in comparison with the integral method. Indeed, the PIS scheme is more economical and generally gives more reliable results than the other numerical approaches. This scheme is fully implicit in time and all the variables are solved simultaneously. Therefore, unlike SIMPLE it does not need a staggered grid and a segregated algorithm with so many steps. Our experience also shows that more accurate predictions can be achieved by determining a proper interphase friction factor. Evidently, the results can be further improved if better correlations are used for the flow regime.

References


