A simplified CFD model for air-lift artificial upwelling

Qicheng Meng\textsuperscript{a,c}, Chizhong Wang\textsuperscript{b}, Ying Chen\textsuperscript{a,b,}\textsuperscript{*}, Jiawang Chen\textsuperscript{b}

\textsuperscript{a} The State Key Lab of Fluid Power Transmission and Control, Zhejiang University, Hangzhou, The People's Republic of China
\textsuperscript{b} Department of Ocean Science and Engineering, Zhejiang University, Hangzhou, The People's Republic of China
\textsuperscript{c} Department of Mechanical Engineering, University College London, Torrington Place, London WC1E 7JE, UK

\textbf{A R T I C L E   I N F O}

Article history:
Received 9 August 2012
Accepted 13 July 2013

Keywords:
DOW
Artificial upwelling
Air-lift
CFD
Two-phase flow
Euler–Lagrange

\textbf{A B S T R A C T}

Air-lift artificial upwelling was simulated through a simplified computational fluid dynamics (CFD) model of two-phase flow in an upward pipe. This model is based on the momentum transfer between the two phases. An algebraic turbulence model was employed to enclose the equations. The volumetric flow rates of the liquid and the gas phases, the void fraction distribution, and the pressure and velocity profiles along the axial coordinate were calculated in the stationary state. Numerical tests were conducted to verify the model. The good agreement obtained between the computational results and the field test measurements with cross-shaped air injectors demonstrates the capability of this model for dispersed bubbly flow conditions. Comparisons with the analytical theory were also made. The results indicated that the proposed engineering-oriented model is efficient for the optimization of the system.

© 2013 Elsevier Ltd. All rights reserved.

1. Introduction

Overfishing represents a global problem, as approximately 75% of the world’s fish stocks are consumed beyond sustainable levels (Souter, 2002). Climate change is also recognised as a severe threat to the global environment (Solomon et al., 2007). Both overfishing and global climate change have significant impacts on the ocean environment, and both of these processes are accelerating.

Artificial upwelling is considered a promising approach for the stimulation of Earth’s capacity to cure itself (Lovelock and Rapley, 2007). The deep ocean water (DOW) is rich in inorganic nutrient salts, which are insufficient in the euphotic layer. Nutrient salts are essential for phytoplankton, which are the foundation of the marine food chain. The areas where natural upwelling is induced are generally significant fishing grounds. Thus, it is natural to hypothesize that artificial upwelling would also promote primary productivities in the ocean. And some recent evidence supports this hypothesis (Maruyama et al., 2011).

There are many creative ways to induce artificial upwelling. In fact, artificial upwelling induced by ocean currents was studied by Hsieh et al. (1978). Isaacs et al. (1976) designed a wave-driven artificial upwelling device that could pump DOW utilising wave energy with a one-way valve. Liu et al. (Liu and Jin, 1995; Liu et al., 2003) studied on the wave pump in regular and random waves and successfully conducted field trials. Another approach, which is denoted perpetual salt fountain, was proposed by Stommel (1956).

In this approach, artificial upwelling is stimulated by the difference in the salinity and temperature between the DOW and the surface water. Large-scale field trials were conducted in the Pacific Ocean and analysed by Maruyama et al. (2004, 2011). Another ambitious project named Ocean Nutrient Enhancer (ONE), which directly pumps deep seawater, was promoted by Marino-Forum 21 (MF21) of Japan. The first ONE device, which is denoted TAKUMI, was deployed in 2003 (Ouchi, 2003).

However, for the objective is to enhance the primary productivity of a large area in the ocean, it is necessary to achieve a large deep seawater flow rate and a relatively very low lift head compared with those required to pump water on land. Liang (1996) utilised bubbles to lift the DOW, which is similar to the mechanism used by air-lift pumps. Liang and Peng (2005) improved the energy flux balance equations developed by Rautenberg (1972). Because air is compressible, a high air flow rate under standard conditions can be obtained by increasing the air pressure. In addition, at a certain scope, the lifted water flow rate increases with a higher air flow rate. Therefore, the objective can be achieved. Liang and Peng (2005) indicated that the ratio of the seawater flow rate to the standard air flow rate is higher with a larger diameter of the upwelling pipe. Thus, this technology is considered promising to achieve a marked increase in the deep seawater flow rate. A recent field test reported by Fan et al. (2013) has enhanced the feasibility of this hypothesis.

Differently from other upwelling concepts, two-phase flow that consists of liquid and gas is found in the upwelling pipe. It is necessary to obtain a better understanding of the material transport process and to establish an appropriate model to control the...
fluid and optimize the apparatus. The engineers are mostly concerned with the sensitivities of the key parameters, e.g., the injection location, the shape of the tube, the air injector shape, and the bubble size.

Simple analytical models are widely used, especially by the industrial community and can be quite successful. The models developed by Rautenberg (1972) and Liang and Peng (2005) originate from the homogeneous model. Empirical correlations for energy losses take into account the overall energy balance. Liang and Peng (2005) highlighted that the power demand associated with the density difference head and the sea surface rise are crucial parameters in the air-lift upwelling system. Fan et al. (2013) improved the analytical theory by including the effects of the gas void fraction and the local head loss. Another successful model, which was developed by Zubler and Findlay (1965), is the drift-flux model that focuses on the relative motion. In addition, the influence of the bubble relative velocity and the radial profile can be incorporated into this model (Gu et al., 2006). Various semiempirical drift-flux parameter correlations have also been investigated (Hibiki and Ishii, 2003).

With the development of computational fluid dynamics (CFD), numerical models have been established based on the understanding of the physical process. Although bubbles do not exhibit all of the attributes of a liquid, the two-fluid model, which is based on the Euler–Euler approach, is able to demonstrate the detailed distribution and profiles of the key parameters in the two-phase flow system. Many different models have also been derived from the two-fluid model. Cheung et al. (2007) compared different models of coalescence and breakage mechanisms using a population balance approach. Pougatch and Salcudean (2008) proposed a model for three-phase flow based on the Euler–Euler approach to simulate deep sea air-lift mining. The influences of the pipe diameter and other parameters were discussed in the optimization of the system. Obviously, two-fluid models are time consuming and quite dependent on the computational resources. Hence, it is difficult to actually model the whole system.

There have been many fundamental studies on bubble formation and bubble rise velocity (Kulkarni and Joshi, 2005), and these studies can provide the bubble-related input parameters for the whole model.

In the present paper, we propose a simplified CFD model to simulate the whole air-lift artificial upwelling system for engineering applications. The proposed model is expected to be efficient and accurate and is based on a clear physical background. Thus, this model is designed to be engineering-oriented and easily coded. The model has to be run repeatedly with a variety of parameters for the optimization of the parameters of the system. Numerical cases based on a lake trial were simulated using different time intervals, grid numbers, and initial conditions. The experimental results and the analytical solutions are provided to validate the proposed numerical model. The analysis of different types of air injectors was undertaken to specify the applicable scope of the model.

2. Mathematical models

2.1. Assumptions

In this study, the models are based on the following assumptions: because the diametre of the upwelling tube is generally very large (1–3 m in actual deployment and 0.4 m in the field test), the air volumetric flow rate is relatively low, and the void fraction is less than 10% in most cases, the applicative regime of this model is restricted to dispersed bubbly flow, and the bubble distribution along the radial direction was assumed to be uniform. In addition, the flow was assumed to be stationary and fully developed. It was also assumed that there is no phase change, no mixing, and no mass transfer between the two phases under any circumstances. Assuming that the flow in the tube is adiabatic, energy equations were not considered in the present study. The liquid phase was assumed to be an incompressible fluid. The bubbles were assumed to be rigid spherical balls and to follow a rectilinear path. Because the gas in the bubbles is compressible and experiences the isothermal inflation process, it obeys the ideal gas law. In addition, the collision, coalescence and breakage of bubbles were not considered. We also assumed that the forces on the bubble are at the equilibrium at each time step.

It should be known that the bubbles are not always spherical and do not always follow a rectilinear path (Wu and Gharib, 2002). However, the study generally measures only one bubble without the disturbance caused by other bubbles. At present, there are no convincing conclusions on the shape and path of a bubble among a large number of bubbles. Based on adequate references on spherical bubbles, we chose the assumptions above and calibrated the coefficients using experimental data.

2.2. Bubble formation

Many studies have investigated bubble formation. Almost 30 approaches and expressions were presented by Kulkarni and Joshi (2005). However, most of these are based on bubble formation at a single submerged orifice. Studies on bubble formation at perforated plates and sieve trays will likely be conducted.

The simplified correlation for the bubble volume estimation called the detached bubble volume, which was developed by Shyu et al. (2006), was used in this study. This correlation was developed from the classic formula provided by Davidson and Schuler (1960). The detached bubble volume is expressed as

\[
V_b = D_h^3 \pi h + \frac{1.31}{\sqrt{D_h g}} \mu_{\text{crit}} \sqrt{D_h / L} \frac{D_h}{R},
\]

where \(D_h\) is the diametre of the air injection hole, \(\rho_l\) the density of the liquid, \(g\) is the acceleration due to gravity, \(\sigma\) is the surface tension, \(L\) is the length of air injection hole, and \(B_0\) is the Bond number, which is calculated by \(B_0 = \rho_l D_h^2 / \sigma\). Both \(R\) and \(u_{\text{crit}}\) are empirical constants.

It was assumed that the pressure inside and outside the bubble equalize as soon as the bubble detaches from the orifice and that bubbles are simultaneously formed at all of the orifices. Thus, the frequency of bubble formation is

\[
f_b = \frac{Q_h / \rho_h}{n_{\text{orifice}} V_b}.
\]

where \(Q_h\) and \(p_h\) are the air volumetric flow rate and the air pressure in the injection pipe, respectively, \(p_h\) the air pressure inside the bubble, and \(n_{\text{orifice}}\) is the number of orifices.

The mean initial velocity of detached bubbles is identified as

\[
u_{\text{initial}} = \frac{Q_h}{A_h}.
\]

where \(A_h\) is the cross-sectional area of the injection pipe.

The initial bubble velocity should be zero because it is attached to the injector before it starts to rise. However, the bubble detachment lasts a very short period. To capture the detachment, the time interval should be very small. Correspondingly, a large amount of computing time would be required. Because we are interested in the steady state in our simulations, we chose a relatively large time interval that skips the detachment and thus we assigned a nonzero initial velocity to the bubble.
2.3. Control equations

In addition to the Euler–Euler approach, the Euler–Lagrangian approach is a popular model for two-phase flow, particularly bubbly flow in which the gas phase is uniform. The research study conducted by Sokolichin et al. (1997) showed that the two methods can agree quantitatively. The gas phase's behaviour is more similar to that of a discontinues medium in the study of dispersed bubbly flow. Hence, Euler–Lagrangian method was employed in the present study.

For a single bubble, the control equation using the Lagrangian method, which is derived from Newton’s Second Law of Motion, may be expressed as

\[
\frac{\partial \rho_B \mathbf{u}_B}{\partial t} = -\nabla P - (\rho_B - \rho_l)\mathbf{V}_B g + F_{gl},
\]

where \( \rho_B \) is the gas density inside the bubble, which varies with the depth, \( \mathbf{V}_B \) the volume of the bubble, which varies with a change in \( \rho_B \), \( \rho_l \) is the density of the adjacent liquid, \( \mathbf{u}_B \) is the velocity of the bubble, \( \nabla P \) is the pressure gradient, \( g \) is the gravitational acceleration, and \( F_{gl} \) is the phase interfacial force acting on the bubble.

In the gas phase, the bubbles are divided into groups. In each group, the attributes (e.g., velocity, position, diametre and interfacial force) of the bubbles are regarded as the same. Thus, one bubble in a certain group can represent all of the attributes of the bubbles in that group. The number of bubbles in a group is denoted \( n_B \). In this study, \( n_B \) was obtained by the product of the bubble formation frequency \( f_B \) and the time interval \( \Delta t \).

For the liquid phase, the Euler method was used to model the fluid. The mass conservation of the liquid is written as

\[
\frac{\partial \rho_l \mathbf{u}_l}{\partial t} + \nabla \cdot (\rho_l \mathbf{u}_l \mathbf{u}_l) = 0,
\]

where \( \rho_l \) is the density of the liquid and \( \mathbf{u}_l \) is the velocity vector of the liquid.

In consideration of the incompressibility condition of water, the equation above was simplified to

\[
\nabla \cdot \mathbf{u}_l = 0.
\]

The momentum conservation equation for the liquid can be expressed as follows:

\[
\rho_l \frac{\partial \mathbf{u}_l}{\partial t} + \rho_l \nabla \cdot (\mathbf{u}_l \mathbf{u}_l) = - (1 - e) \nabla P + \rho_l g + \nabla \cdot \left[ \mu \nabla \mathbf{u}_l \right] + F_{wall} + F_{lg},
\]

where \( F_{wall} \) is the wall friction induced by the tube wall boundary, and \( F_{lg} \) is the counter acting force of all of the interfacial forces acting on the bubbles in the control volume.

\[
F_{lg} = - \sum n_B F_{gl},
\]

where \( n_B \) is the number of bubbles in a group and \( \sum \) represents the summation of all of the interfacial forces of the bubble groups in the control volume.

\[ F_{wall} \] follows Darcy’s model:

\[
F_{wall} = - \lambda \frac{H | \mathbf{u}_B \mathbf{u}_l | \mathbf{u}_l}{2D},
\]

where \( D \) is the diameter of the upwelling tube, \( H \) the length of the tube, \( A \) the cross-sectional area of the tube, and \( \lambda \) is the friction coefficient, which is an empirical coefficient that varies depending on the Reynolds number.

2.4. Interfacial momentum transfer

In general, there are four types of interfacial forces that act on the two-phase interface in the vertical direction. These are the drag force, lift force, virtual mass force, and liquid thrust, which are denoted \( F_{drag} \), \( F_{lift} \), \( F_{v} \), and \( F_{p} \), respectively (Johnson and Hsieh, 1966). The total interfacial force \( F_i \) may be expressed as

\[
F_i = F_{lg} = - \sum n_B F_{gl} = \sum n_B (F_{drag} + F_{lift} + F_{v} + F_{p}).
\]

Based on the above mentioned assumptions, the drag force is the dominant force, and the other forces are negligible.

The drag force was given by Johnson and Hsieh (1966) as

\[
F_{drag} = C_D \rho_l \frac{1}{2} \rho_B \mathbf{u}_B \mathbf{u}_l | | \mathbf{u}_B - \mathbf{u}_l | | (\mathbf{u}_B - \mathbf{u}_l),
\]

where \( a_{gl} \) is the surface area of the bubble, \( C_D \) is the drag coefficient, and \( \mathbf{u}_l \) is the mean velocity of the liquid at the vicinity of the bubble.

The drag coefficient followed Pougatch and Salcedoane (2008), which is

\[
C_D = \begin{cases} 
2 \frac{1 + 0.158 Re^{1/2}}{Re} & \text{if} \ Re < 1000, \\
0.44, & \text{if} \ Re > 1000, 
\end{cases}
\]

and

\[
Re_B = \frac{\rho_l | \mathbf{u}_B - \mathbf{u}_l | | \mathbf{u}_B |}{\mu},
\]

where \( Re_B \) is defined as the bubble-related Reynolds number, \( \mu \) the diametre of the bubble, \( \mu_l \) the viscosity of the liquid, and \( | \mathbf{u}_l | \) is the adjacent liquid velocity. The variable \( \mathbf{u}_l \) is replaced by the mean liquid velocity of the control volume in the following discretisation procedure:

When we solve the equations, both \( \mathbf{u}_l \) and \( \mathbf{u}_B \) are unknown variables. Based on the assumption that the forces on the bubble are at equilibrium, iteration was performed at each time step to determine the bubble velocity that can achieve equilibrium.

It should be noted that most of the classic formulas for the drag coefficient are based on the assumption of a single rigid spherical bubble in fresh water. The effects of the interactions between bubbles and different media, such as seawater and contaminated water, need to be addressed in further studies.

2.5. Turbulence models

When the effect of turbulence is considered, the liquid velocity can be decomposed into two parts, which are the mean value and the fluctuation value

\[
\mathbf{u} = \bar{\mathbf{u}} + \mathbf{u}',
\]

Thus, Eq. (7) can be derived in the form of Reynolds-Averaged Navier–Stokes equations (RANS)

\[
\rho_l \frac{\partial \bar{\mathbf{u}}}{\partial t} + \rho_l \nabla \cdot (\bar{\mathbf{u}} \bar{\mathbf{u}}) = - \nabla P + \rho_l g
\]

\[
+ \nabla \cdot \left[ \mu \nabla \bar{\mathbf{u}} - \rho_l \mathbf{u}' \mathbf{u}' \right] + F_{wall} + nF_{lg},
\]

The Reynolds stress is defined as

\[
\tau_{ij} = - \rho_l \bar{\mathbf{u}}_i \bar{\mathbf{u}}_j.
\]

In the empirical formula of \( F_{wall} \), the turbulence effect is partly considered in the vicinity of the tube wall. However, because there is a relative velocity between the gas and the liquid phases, the velocity gradient is very large in the vicinity of the two-phase interface. According to the turbulence theory, the turbulence intensity is large where the velocity gradient is large. Thus, it is necessary to model the turbulence dissipation effect in this area.

Among the algebraic turbulence models, the Prandtl mixing-length hypothesis is the most famous model. Inspired by Prandtl's
model, the term $\phi$, which represents the turbulence and viscosity dissipation around the bubble, was added to the right-hand side of Eq. (15)

$$\Phi = -\sum n_{\text{fl}}\rho_{\text{l}}|\mathbf{u}_\text{l} - \mathbf{u}_\text{b}|(\mathbf{u}_\text{l} - \mathbf{u}_\text{b}),$$  

where $n_{\text{fl}}$ is the viscosity dissipation length of the bubble.

If the liquid phase is restricted to one dimension, Eq. (15) becomes

$$\rho_l \frac{\partial \mathbf{u}_\text{l}}{\partial t} + \rho_l \nabla \cdot (\mathbf{u}_\text{l} \mathbf{u}_\text{l}) = -\nabla P + \rho_l g + \phi + F_{\text{wall}} + nF_{\text{bg}}.$$

For one-dimensional discretisation, the definition of the viscosity dissipation length is different. This coefficient is an empirical constant that takes into account all of the turbulence dissipation in the vicinity of the bubbles. Thus, the expression is converted into

$$\phi = -\sum n_{\text{fl}}\rho_{\text{l}}C_{\text{bd}}A_B|\mathbf{u}_\text{l} - \mathbf{u}_\text{b}|(\mathbf{u}_\text{l} - \mathbf{u}_\text{b}),$$

where $A_B$ is the cross-sectional area of the bubble and $C_{\text{bd}}$ is the viscosity dissipation coefficient of the bubble.

The coefficient $C_{\text{bd}}$ is a closure coefficient in the turbulence model. The definition of $C_{\text{bd}}$ is the ratio of the dissipation layer thickness to the diameter of the bubble

$$C_{\text{bd}} = \frac{l_{\text{sh}}}{d_{\text{sh}}}.$$  

Similarly to the mixing-length hypothesis (Wilcox, 2006), the model must be calibrated with experimental data. After calibration with one of the experiments, it can be applied to the other cases with the same regime. To complete the solution, we used data from field tests, which will be described in detail in Section 4.2. We have to restate that the scope of the model is dispersed bubbly flow. The flow regime might no longer be dispersed bubbly flow if circular air passage injectors are used. In the following calibration, the experimental data obtained with crossing air passage injectors were adopted.

To assess the sensitivity of the water flow rate to changes in $C_{\text{bd}}$, a non-dimensional coefficient $\gamma$ was introduced

$$\gamma = \begin{cases} \frac{|\mathbf{u}_\text{inlet} - \mathbf{u}_\text{exp}|}{\mathbf{u}_\text{max}}, & \text{if } \mathbf{u}_\text{inlet} > \mathbf{u}_\text{exp} \\ |\mathbf{u}_\text{inlet}|, & \text{if } \mathbf{u}_\text{inlet} > \mathbf{u}_\text{exp}. \end{cases}$$

where $\mathbf{u}_\text{inlet}$ is the variable that represents the mean liquid velocity at the inlet of the upwelling pipe, $\mathbf{u}_\text{max}$ the maximum mean liquid velocity at the inlet of the upwelling pipe calculated by the model when $C_{\text{bd}}$ is zero, and $\mathbf{u}_\text{exp}$ is the average of the experimental mean liquid velocity measured at a certain air injection volumetric flow rate. It was noted that the air flow rate has a minimum value. If the air flow rate is less than the minimum value, no water can be lifted.

At a certain air injection volumetric flow rate, an array of $\mathbf{u}_\text{inlet}$ is obtained for changing values of $C_{\text{bd}}$. There is an upper limit for $C_{\text{bd}}$ when the liquid velocity is zero. The value of $\gamma$ moves toward 1 when $\mathbf{u}_\text{inlet}$ is closer to the experimental data and tends to 0 when $\mathbf{u}_\text{inlet}$ is farther from the experimental data. In other words, this variable acts as the indicator of the best value of $C_{\text{bd}}$.

As mentioned previously, two sets of experimental data were normalised. As shown in Fig. 1, each solid line represents the normalisation of one of the experimental results using air injector type 1. Each dotted line represents the normalisation of one of the experimental results using air injector type 2. Almost every poly-line has a single crest, which means that we can choose an optimal value of $C_{\text{bd}}$ for each experiment. However, due to the complicated nature of two-phase flow and experimental errors, the best values of $C_{\text{bd}}$ do not converge to a certain constant. These vary within a certain scope, from approximately 0.18–0.195. Because we only calibrate it with two set of data, which almost totally overlap each other within this scope, it is not adequate to develop an algebraic model for $C_{\text{bd}}$. Thus, we chose a intermediate value of 0.185 for $C_{\text{bd}}$, which is marked by the dotted dashed line in Fig. 1.

3. Numerical discretisation and procedures

3.1. Discretisation and grid generation

As mentioned above, those bubbles with the same attributes form a group. In the programming section, a structure is allocated to a bubble group. The attributes of the bubble group are attached to the structure. The structure is allocated once the bubbles form and is destroyed when the bubble group escapes the upwelling tube outlet. During the life cycle of the structure, the parameters in it change with the state of the bubbles and take part in the coupling and iterative calculation of the algorithm. This is a compromise to the numerous variables included in the Lagrange method.

The Finite Volume Method (FVM) was employed to discretise the liquid phase. We substituted $\phi$ for $\mathbf{u}_\text{l}$. Eq. (18) can then be expressed as

$$\rho_l \frac{\partial \mathbf{u}_\text{l}}{\partial t} + \rho_l \nabla (\mathbf{u}_\text{l} \mathbf{u}_\text{l}) = -\nabla P + \rho_l g + \phi + F_{\text{wall}} + nF_{\text{bg}}.$$

where $\mathbf{u}_\text{l}$ is the vertical component of the liquid velocity, $\Gamma$ is the diffusion coefficient, and $S$ is the source term.

The convective and dissipative terms were discretised by the first-order upwind scheme and the central differencing scheme, respectively. The temporal terms were discretised by the explicit scheme. The slip wall condition was used. The boundary conditions are the pressure inlet and the pressure outlet at the bottom and the top of the upwelling tube, respectively. The air injection location is at the lower bound of a certain control volume in the middle of the tube. The volume of the air injector was not considered.

The source term $S$ was decomposed into

$$S = S_u + S_p \phi_p.$$  

In the momentum equation, this term can be expressed as

$$S_u = S_{uc} + S_{wp} \mathbf{u}_l,$$

$$S_{uc} = -\nabla P + \rho_l g - \frac{\delta k_{el} |\mathbf{u}_l| |\mathbf{u}_l|}{2D} A.$$
is distributed uniformly. In the two-phase flow, the tube. There is single-phase injection technique, the air injector is located at a certain depth in the tube. In the two-phase flow region, the void fraction might be larger than one if the grids are uniform. This problem usually occurs with an unresolved model, in which the cell size is larger than the bubble size. At each time step, the bubbles in one group are assumed to be within a certain grid, which results in both temporal and spatial discontinuity. These discontinuities are worsened if the bubble expands three-dimensionally as it rises. To solve this problem, the grids were self-adapted and thus extend one-dimensionally as the depth decreases. This technique ensures calculation stability.

We assumed that the uniform grid interval at the single-phase flow region is \(\delta x_0\). This is also the interval of the first control volume above the air injector. Those bubbles that form within the same time interval are assumed to have the same velocity. Therefore, there is a minimum value for \(\delta x_0\)

\[
\delta x_{\text{min}} \geq V_{\text{initial}} \Delta t.
\]

where \(\delta x_{\text{min}}\) is the minimum spatial interval, \(V_{\text{initial}}\) is the initial velocity of the bubbles, and \(\Delta t\) is the time interval.

In the two-phase flow region, the spatial interval \(\Delta x\) is a function of the depth

\[
\Delta x(m) = \Delta x(m-1) \sqrt{\frac{\rho_A h_B + p_0}{\rho_A h_B - \Delta x(m-1) + p_0}} \Delta x(0) = \Delta x_0, m = 1, 2, 3, ...
\]

where the spatial interval of the first grid above the air injector is \(\Delta x_0\), which is equal to \(\Delta x(0)\), \(\Delta x(m)\) is a sequence of spatial intervals that starts from the air injector and ends at the outlet of the upwelling tube, \(h_B\) is the depth of the control volume, and \(p_0\) is the atmospheric pressure.

### 3.2. Numerical procedures

The SIMPLE method based on staggered grids was used to solve Eqs. (6) and (22). The variables for the pressure and the air void fraction are stored in the centre of a cell, and the variables for velocity are stored at the face centre of a cell. The cross-sectional area through the cell centre is obtained by \((1-\varepsilon)A\), where \(A\) is the original cross-sectional area though the cell centre without bubbles. The variable \(\varepsilon\) is the air void fraction of a control volume, which is defined as

\[
\varepsilon = \sum_{k=1}^{n} \frac{V_k}{\Delta V}.
\]

where \(\Delta V\) is the volume of the control element and \(\sum\) represents the summation of all of the bubbles in the control volume.

At the face centre of the cell, the void fraction is obtained by averaging the void fractions in the neighbouring cells.

As an example, a numerical case was calculated and compared with the experimental data, which is shown in Fig. 3. The input volumetric air flow rate of this case was 0.00456 m³/s at 2.37 × 10⁵ Pa in the field test conducted with air injector type 1. The measure water volumetric flow rate was 0.0858 m³/s. The estimated mean water velocity of the cross-sectional area at the inlet of the upwelling tube was 0.683 m/s. We found that the numerical solution ultimately converged to the experimental result and that the velocity residual increases with the time step at the beginning and then decreases to small quantity of the order \(O(10^{-4})\).

The programme was written in MATLAB®. As an example, the numerical case described above, which consists of 274 cells and converges in 500 time steps with a 0.1 s interval, requires 40 min to reach convergence with an Intel® core™ i7 CPU.

### 4. Numerical results and discussions

#### 4.1. Sensitivity tests

The time interval sensitivity, the grid number sensitivity and the initial condition sensitivity were tested using the case described in Section 3.2, and the results are illustrated in Figs. 4–6, respectively.

As shown in Fig. 4, the simulation was run using different time intervals, and the other input parameters were fixed. The residuals were restricted to the ranges of \(10^{-4}\) m/s and 1 Pa. All of the results converge to a certain value with a relative error bound of approximately 2.4%. The calculation convergence becomes faster with larger time intervals. However, at very large time interval, such as 0.15 s, the velocity oscillates slightly.

The case was also studied using different spatial intervals, as illustrated in Fig. 5. The residuals were restricted to the ranges of \(10^{-4}\) m/s and 1 Pa. All of the results converge to a certain value with a relative error bound of approximately 0.9%. There is little difference between the convergence speeds. At very small spatial intervals, there is a slight fluctuation.
The strategy is to solve the stationary question using the time-matching method. Thus, a good estimation of the initial bubble population distribution and the initial velocity field of the liquid are of benefit to make the calculation converge faster. To test this, different initial conditions were used in the above case, as shown in Fig. 6. The residuals were restricted to the ranges of 10\(^{-4}\) m/s and 1 Pa. All of the results converge to a certain value with a relative error bound of approximately 2.3%. The convergence speed was faster when the initial estimation was closer to the experimental result.

To summarise, when the time intervals, grid numbers, and initial conditions vary over a large range, the liquid could converge to a certain value, and the accuracy is satisfactory. The convergence speed strongly depends on the initial conditions and the time intervals. This model is robust, even with sparse grids and large time intervals. Thus, the time required to solve it can be significantly reduced.

### 4.2. Field test details

In October 2011, the artificial upwelling group in Zhejiang University conducted a field test at the Xin’anjiang Experimental Station in Thousand Island Lake, Zhejiang Province, China. The average depth around the location where the catamaran moored was approximately 50 m. A 28.3 m-long tube was deployed from the deck of the catamaran and submerged beneath the lake surface. The submerged depth of the tube outlet was approximately 2.1 m, and its internal diameter was 0.4 m. The air injector was 8 m below the outlet of the tube. An electromagnetic flow meter with the same diameter as the tube was chained to the upwelling pipeline. This metre was located beneath the air injector (i.e., where the single-phase flow region is) and measured the liquid volumetric flow rate. The pressure and volumetric flow rate of the compressed air, the static pressure inside and outside of the tube, and the temperatures were also measured using an integrated monitoring system. The schematic diagram of the air-lift upwelling test system with the measurement system is illustrated in Fig. 7.

There are four types of the air injectors, as shown in Table 1 and Fig. 8. The total injection areas of the air passages were designed to be the same to allow the investigation of the impacts of both the different shapes of the air passage and the different diameters of the orifices.

For each type of air injectors, the air injection volumetric flow rate was turned up in several steps. It was observed that both the air volumetric flow rate and the air pressure fluctuated significantly, whereas the water volumetric flow rate fluctuated periodically at certain air injection flow rate ranks.

A field test record segment of the water volumetric flow rate is shown in Fig. 9, and a time series of the air volumetric flow rate, which was converted to that obtained under standard conditions, is showed in Fig. 10.

A more detailed description of this field test can be found in the report by Fan et al. (2013).

![Fig. 7. Schematic diagram of the lake test system (Chen et al., 2011).](image)

**Table 1**

<table>
<thead>
<tr>
<th>Type</th>
<th>Shape of the air passage</th>
<th>No. of orifices</th>
<th>Diameter of a single orifice (mm)</th>
<th>Total injection area (mm(^2))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cross</td>
<td>24</td>
<td>0.5</td>
<td>75.36</td>
</tr>
<tr>
<td>2</td>
<td>Cross</td>
<td>384</td>
<td>2</td>
<td>75.36</td>
</tr>
<tr>
<td>3</td>
<td>Circular</td>
<td>24</td>
<td>0.5</td>
<td>75.36</td>
</tr>
<tr>
<td>4</td>
<td>Circular</td>
<td>384</td>
<td>0.5</td>
<td>75.36</td>
</tr>
</tbody>
</table>
4.3. Analysis of the results

The relationship between the air injection flow rate and the lifted liquid flow rate is our primary concern. In addition, this relationship may be affected by different types of air injectors.

Fig. 11 shows the comparison of the mean water velocities between the numerical simulations and the experimental data obtained using the air injector type 1. The experimental data originated from part of the field test record. The inputs used in the numerical calculation were determined based on the mean value of the air injection flow rate and pressure in the experimental data. The water volumetric flow rate was estimated by multiplying the mean water velocity by the cross-sectional area of the pipe.

Fig. 12 illustrates the comparisons of the volumetric flow rates under standard conditions between the numerical simulation and the experimental data using the four types of air injectors (the sub-figures (a–d) show the comparisons for air injectors type 1–4, respectively).

As illustrated in Fig. 12(a), the numerical simulations are in good agreement with the experimental data using air injector type 1 when the standard volumetric air flow rate is between 0.001 cms and 0.012 cms. In general, the upwelling system works within this range of air flow rate, which can also be referred to as operating interval. Within this interval, we discovered that the water flow rate increases with an increase in the air flow rate. However, the standard air flow rate is not the only factor that influences the water flow rate. Even with the same standard air flow rate, the water flow rate may not be the same. The model overestimates the water flow at lower air flow rates. At the maximum air flow rate, the water flow rate decreases. However, there is only one singular point at the maximum. It is difficult to identify the exact reason.

We then introduced a criterion denoted the lifting efficiency, which is the ratio of the lifted water volumetric flow rate against the standard air volumetric flow rate. According to the model, the lifting efficiency achieved with orifices with a smaller diametre is higher than that obtained with larger diameters. This finding is also supported by some ideal experiments. As shown in Fig. 12(b), the experimental lifting efficiency using air injector type 2 is slightly higher than that obtained with air injector type 1. However, it is overestimated by the numerical simulation. It is argued that the energy dissipation increases as the diametre of the orifice becomes larger. From the point of view of the whole system, this may neutralise the benefit achieved with smaller bubbles.

The circular air injector is quite different from the cross-shaped air injector. The orifices are arranged along the tube wall. As illustrated in Fig. 12(c and d), the model and the experimental results do not agree well. Due to the shape of the injector, the bubbles are most likely concentrated in the vicinity of the tube wall. Thus, it can be inferred that the bubble distribution along the radial direction varies widely with this unique air passage shape.
It can be argued that the assumptions used in the development of the model are not applicable in this case.

As mentioned above, Liang and Peng (2005) presented an elegant theory that allows us to recognize the advantage of air-lift artificial upwelling. The power balance equation may be expressed as

\[ N_I = N_R + N_{Ru} + N_b + N_E + N_x + N_{rise}. \]  

(28)

In this equation, \( N_I \) is the air input power derived from the power demand equation for the air compressor, and \( N_R \) and \( N_{Ru} \) are the frictional losses in power in the upper and lower pipe, respectively. The Darcy formula was used to calculate these variables, and the friction factor \( \lambda \) in the Darcy formula was given by Gu (1936)

\[ \lambda = 0.01227 + \frac{0.7543}{Re^{0.38}}, \]  

where \( Re \) is the Reynolds number of the water flow.

\( N_b \) is the velocity power of the mixing fluid, and \( N_E \) is the entrance frictional loss power. The entrance frictional loss coefficient \( \xi_e \) in the entrance frictional loss formula was set to 1. \( N_E \) is the frictional loss power of the relative sliding speed between the two phases and was calculated using an empirical formula, which includes a coefficient that varies with the sea surface rise height \( H_s \). \( N_b \) is the power demand of the density difference head and is not considered in this study because this term is very small in fresh water. \( N_{rise} \) is the power demand of the sea surface rise, which is denoted by \( H_s \), and this variable is correlated through an empirical formula that contains the water flow rate. The empirical coefficients in Eq. (28) are either constants or related to the water flow rate or standard air flow rate. Therefore, there are only two unknown variables in the equation: the water flow rate and the standard air flow rate. A quadratic equation is solved, and the relationship between the water flow rate and the standard air flow rate is expressed analytically. This model does not need to be calibrated. In addition, significant improvements can be made through a sound analysis on the power dissipation (Fan et al., 2013).

The theoretical results reported by Liang and Peng (2005) are given in Fig. 12(a–d) for comparisons. As illustrated in these figures, the trends of the theoretical predictions and the numerical results agree very well. Nevertheless, the theory overestimates the water flow rate compared to the numerical calculation and the field test results, especially at high air flow rates. No differences were obtained in the theoretical predictions with the different types of air injectors because the theory only takes the flow rate into account. As shown, the numerical simulations conducted in this study are closer to the experimental results than the theoretical results reported by Liang and Peng (2005).

To explore the mechanisms of two-phase flow, a comparison of the pressure variation distribution between the numerical simulation and field test is presented in Fig. 13. The case described in Section 3.2 was chosen as the example. Because the variation in the pressure inside the tube is relatively small compared to the
As shown in Fig. 1, inside pressure recorded. In other words, the driving pressure required to lift DOW is very low. The pressure head is in the order of several tens of centimetres. The accuracy of the pressure measurement is relatively low in a harsh field test environment. To the best of our knowledge, this might be the first field test pressure measurement in an air-lift artificial upwelling area, and we include it to testify the hypotheses given by Liang and Peng (2005) and Pougatch and Salcudean (2008). Due to the decrease in the density of the air-water mixture, the pressure in the pipe is lower than that recorded at the same depth outside the pipe, and this difference becomes the driving force.

As shown in Fig. 13, two sets of pressure sensors are located beneath the air injector and the other set of two sensors is found above the air injector. The static pressure profile of the numerical simulation passes through the mean value of the measurement at the deepest measurement point, which is at the upper bound of the experimental error bar of the two measurement points. The trend of the pressure profile is the same as that simulated by Pougatch and Salcudean (2008). However, the pressure drop in the two-phase flow portion should be modelled more appropriately.

4.4. Discussions

(1) As illustrated in Fig. 12 (a and b), the numerical simulations agree well with the results of the field test when the regime is restricted. It was assumed that the bubbles are dispersed and distributed uniformly in the radial direction when the cross-shaped injector was adopted. However, the regime might change when the injector is circular (Fig. 12(c and d)). In this case, the bubbles may concentrate in the vicinity of the tube wall; thus, the assumptions of the numerical model are not satisfied. The scope of the application of this model needs to be verified through additional experiments. Because the lifting efficiency is much higher using the cross-shaped injector, it is reasonable to use the cross-shaped injector rather than the circular injector in practice.

(2) In connection with the above, the numerical prediction is not sufficiently satisfactory when the air flow rate is too large or too small. The conclusion at present is that the model works well in the operating interval. The model makes no sense when the system is operated at a very low air flow rate, and the capability of the air compressor limits the maximum value of the air flow rate.

(3) As shown in Fig. 1, $C_{bd}$ is obviously not a constant. Due to the complexity of two-phase flow and the lack of field test data, we assigned $C_{bd}$ an intermediate value. An empirical formula for $C_{bd}$ might be established with more experimental data.

(4) The analytical model and the numerical model were developed from different perspectives (energy conservation and momentum conservation, respectively). Both of these have their advantages and disadvantages. One of the advantages of the numerical model is that it considers more parameters. The theory only reveals that the water flow rate increases as the standard air flow rate increases, whereas the numerical results show that, even with the same standard air flow rate, the water flow rate can vary due to other parameters. The conclusion above is supported by experimental data.

(5) Section 1 mentions the parameters that the engineers are concerned with. The injection location and the cross-sectional areas along the upwelling tube can be changed in our model. Thus, the model will aid investigation of these parameters. The effect of the bubble size is not well captured by the constant vicinity dissipation coefficient $C_{bd}$. It might be more accurate if $C_{bd}$ is calibrated with experimental data obtained for orifices with different diameters. Because this model is confined to dispersed bubbly flow, the air injector shape is not addressed in the model.

(6) The proposed model is a model of the whole system, and progress in the fundamental research in this area can help improve the modules of the model. Previous studies have focused on bubble formation at a single orifice. However, bubble formation at perforated plates and sieve trays is quite different. The drag coefficients of bubbles in seawater and contaminated water are different from that in pure water. The bubble sizes and shapes are not uniform, and the radial distribution is not uniform. The collision, coalescence, and breakup of bubbles also exist. A method using a distribution parameter and probability models can be developed to take into account the above factors. Furthermore, when the actual project is implemented, flexible hoses will be adopted. These pipes deform in an ocean environment, and this process will affect the two-phase flow regime inside. At present these issues hinder us from applying this model, as well as other models, in practice.

5. Conclusions

A simplified numerical model of air-lift artificial upwelling is presented. This model robust performance and highly efficient computational capacity were confirmed through numerical experiments. A field test was conducted in Thousand Island Lake, China. A 28.3 m-long, 0.4 m in diameter upwelling pipe and four different types of air injectors were used in the field test, and the time-series data of various parameters were measured. Comparisons between the field test records, the numerical simulations, and the theoretical predictions were made. The good agreement obtained in this study demonstrates the capability of this model in the study of the dispersed bubbly flow regime. This regime is defined as the operating interval. However, with different types of air injectors and different air volumetric flow rates, the two-phase flow regime may change such that it is beyond the assumptions of the model. This model is engineering-oriented and integrates the whole system. In addition, algorithm is efficiency. To optimise the system, it is convenient to repeat the calculations with variable parameters. Additional study is needed to validate the model and determine its application range. The accuracy and universality of the model can also be improved by further studies examining the mechanisms of two-phase flow.

Acknowledgements

This work was supported by the National Natural Science Foundation of China (No. 51120195001) and the Program for Zhejiang Leading Team of S&T Innovation (No. 2010R50036). We would like to acknowledge Dr. Haocai Huang, Dr. Han Ge, Miss Shan Lin, Miss Jinjying Leng, and the crews for their help with the field test. Thanks are also given to Prof. Huachen Pan for his advice. The authors are particularly grateful to the anonymous reviewers for their inquiries and comments that shed light on the manuscript.
References


