Analysis of bubble distribution characteristics in a multiphase rotodynamic pump

Yongjiang Li, Zhiyi Yu*, Wenwu Zhang

School of Mechanical Engineering, Beijing Institute of Technology, Beijing, China
* Correspondence: yuzhiyi@bit.edu.cn; Tel.: +86-010-68914304-806

Abstract. Study of bubble distribution characteristics is the basis to understand the mechanism of gas-liquid two-phase flow in multiphase pumps. In the present work, a method based on Eulerian-Eulerian approach was used for the simulation of the flow in a multiphase rotodynamic pump. The bubble size was calculated through a bubble number density equation. The distribution characteristics of bubbles were analysed in conditions of different inlet gas volume fraction (IGVF). The results show that bubbles move to the hub and a homogeneous bubble size distribution can be observed due to the rotational effect and stirring action of the impeller blades. The bubble size increases in the vicinity of interaction region because of the rotor-stator interaction. Collision rate of bubbles increase and a large bubble size is shown in the diffuser, which is resulted from the decrease of bubble velocity. It is found that high IGVF cause small distance between bubbles so that collision rate of bubbles increase. Largest bubble size and gas volume fraction are obtained when IGVF is 15%.

1. Introduction

Gas-liquid multiphase pumps are widely used in petroleum, chemical, water conservancy and nuclear industry, and the improvement of the pump performance makes big profit for related industries. So, it is necessary to study the flow in the gas-liquid multiphase pump, such as gas-liquid separation, the coalescence and break-up of bubbles and fluctuation of phase volume fraction, and take measures in a targeted manner to improve the performance of the pump.

In order to understand the gas-liquid flow mechanism in both macro and micro scale, numerical simulation is an indispensable tool and the selection of two-phase models is the first step. Euler-Lagrange (E-L) [1,2] model and Euler-Euler (E-E) [3-5] model are always selected to simulate the gas-liquid flow. It is believed that the use of E-L model allows the introduction of coalescence, collisions and break-up easily, but the number of bubbles in simulation is limited. While E-E model is applicable to a wider range of volume fraction and it can be computationally economic. The size and distribution of bubbles in two-phase flow is a key issue and the drag force and the turbulent dispersion force are related to it. Kocamustafaogullari and Ishii [6] established a general form of the population balance equation and the number probability density function has been used to predict the bubble size. The population balance equation is comprehensive in describing the process of break-up and coalescence of bubbles, but it can be computationally expensive. F.Lehr [7] made a simplification and derived an equation fit for condition of large gas volume fraction. However, the coalescence and break-up terms of the equation depend heavily on empirical data. In order to pursue a simpler approach, Bakker [8,9] used a bubble number density equation to predict local average diameters and the bubbles tend towards an equilibrium average size. Based on Bakker's research, the bubble number density equation was applied to the flow simulation in a stirred tank [10], and the flow pattern, bubble
distribution and bubble diameter were well predicted. Similar to the stirred tank, gas-liquid multiphase pumps also have rotational impellers, but most scholars took the bubble diameter as a constant [11-13], which is far from the actual situation.

In this paper, the bubble number density equation is introduced in the simulation of the flow in a multiphase pump. The size and distribution of bubbles in the pump is analyzed in conditions of different IGVF. Validation of the proposed model is achieved by qualitative and quantitative comparisons between experiment and simulation.

2. Numerical method

2.1. Governing equations

Two fluid model based on E-E model is used. The model considers the gas and liquid phases as two interpenetrating fluids [14]. The mass and momentum equations are as follows

\[
\nabla \cdot (\alpha_i \rho_i \mathbf{w}_i) = 0
\]

\[
\nabla \cdot (\alpha_i \rho_i \mathbf{w}_i \mathbf{w}_i) = -\alpha_i \nabla p + \mathbf{M}_i + \alpha_i \rho_i \mathbf{f}_i + \alpha_i \mathbf{\tau}_i
\]

Where the subscript \(i = l\) or \(g\) denotes the liquid or gas phase; \(\alpha_i\) is the phase volume fraction; \(\rho_i\) is the density; \(\mathbf{w}_i\) is the velocity vector; \(\mathbf{M}_i\) is the interphase force; \(\mathbf{f}_i\) is the mass force relevant to the rotation of the impeller; \(\mathbf{\tau}_i\) denotes the viscous stress tensor concerning the fluid viscosity and turbulence viscosity.

2.2. Interphase force

The interphase force can be taken to be a sum of several forces, which mainly contain drag force \(D\), added mass force \(A\), lift force \(L\), and turbulent dispersion force \(T\) [13], given by

\[
\mathbf{M}_i = \mathbf{D}_i + \mathbf{A}_i + \mathbf{L}_i + \mathbf{T}_i
\]

Where the added mass force is induced by the relative acceleration between the two phases; the lift force is significant when there is strong vorticity or a large velocity gradient in the continuous phase; The turbulent dispersion force arises due to non-zero correlations between the fluctuating variables. The interphase force between liquid and gas is equal in value but opposite in direction. The expressions are given by

\[
\mathbf{A}_l = \alpha_l \rho_l \mathbf{C}_A (\frac{d \mathbf{w}_l}{dt} - \frac{d \mathbf{w}_g}{dt})
\]

\[
\mathbf{L}_l = -\mathbf{L}_g = \alpha_l \rho_l \mathbf{C}_L (\mathbf{w}_l - \mathbf{w}_g) \times (\nabla \times \mathbf{w}_l)
\]

\[
\mathbf{T}_l = -\mathbf{T}_g = -C_{ad} \rho_l k_l \nabla \alpha_l
\]

Where \(\mathbf{C}_A\) and \(\mathbf{C}_L\) are respectively added mass coefficient and lift coefficient, whose values are both taken as 0.5 [15]. \(C_{ad}\) is the turbulent dispersion coefficient and its value is 0.1 [16]. \(k_l\) is the turbulent kinetic energy.

Drag force is the resistance of bubbles in the flow field, given by

\[
\mathbf{D}_l = -\mathbf{D}_g = -\left(\frac{1}{2} \rho_l \pi d^2 \mathbf{C}_D \left| \mathbf{w}_l - \mathbf{w}_g \right| \right)
\]

Where \(\mathbf{C}_D\) is the drag coefficient, under the research contributed by [17], it can be given by

\[
\mathbf{C}_D = \max(C_{D0}, C_{D\infty})
\]
\[ C_{d1} = \frac{24(1 + 0.1Re_s^{0.75})}{Re_s} \] (9)

\[ C_{d2} = \frac{2d}{3\alpha_i^{0.5}} \left( \frac{\rho_i - \rho_s}{\sigma} \right)^{1/2} \] (10)

\[ Re_s = (1 - \alpha_s) \frac{\rho_d |w_i - w_s|}{\mu_i} \] (11)

Where \( \mu_i \) is water dynamic viscosity; \( \sigma \) is surface tension coefficient; \( Re_s \) is relative Reynolds number.

2.3. Prediction of bubble size

A model is proposed based on a bubble number density equation [8], which allows the prediction of the bubble size and bubble distribution at each point in the calculation. The Bubble number density is calculated as

\[ n = \frac{\alpha_s}{\pi / 6d^3} \] (12)

Where \( d \) is taken as the mean bubble diameter at each point in the calculation. The conservation equation for bubble number density is given by [7]

\[ \nabla \cdot (\mathbf{n} \phi) = \phi_{br} - \phi_{co} + \phi_{ph} \] (13)

Where \( \phi_{br}, \phi_{co} \) and \( \phi_{ph} \) are source terms specifying rates of break-up, coalescence, and phase change.

According to reference [7], the rate of bubble break-up is assumed to be related to the collision rate with turbulent eddies of similar size, the turbulent velocity and the ratio of turbulent shear force to restored surface tension force. Additionally, the rate of bubble coalescence is related to the rate of binary collisions between bubbles moving with a random velocity. Expressions are as follows

\[ \phi_{br} = C_{br} (1 - \alpha_s) \left( \frac{\varepsilon}{d^2} \right)^{1/2} \exp\left( -\frac{We_{br}}{We} \right) \] (14)

\[ \phi_{co} = C_{co} (1 - \alpha_s) \eta_{co} \left( \frac{d^5}{n^2} \right) \] (15)

Here, \( We \) is the bubble Weber number, \( \eta_{co} \) is the coalescence efficiency which takes into account the finite time for bubble deformation and film drainage, given by

\[ We = \frac{2\rho_d \varepsilon \frac{d^5}{\sigma}}{5} \] (16)

\[ \eta_{co} = \exp\left( -\left( \frac{We}{4} \right)^{1/2} \right) \exp\left( -K_{co} \frac{d^3}{\varepsilon} \right) \] (17)

Where \( \varepsilon \) is turbulence dissipation rate; \( C_{br} \) and \( C_{co} \) are constants fitted to the experimental data and taken as 0.3 and 6.0 respectively; \( We_{br} \) represents a critical value of bubble Weber number with the value of 1.5; \( \sigma \) is the surface tension coefficient; \( K_{co} \) is a constant obtained by fitting the model, and taken as \( 5.49 \times 10^6/(s^3 \cdot m) \).
2.4. Model verification
To verify the reliability and accuracy of the numerical method, the software CFX is used for the flow simulation in a bubble column. As is shown in Figure 1, there is a certain amount of water in the column and it is static initially. A hole of 1 cm in diameter is located at the center of the bottom plate. The temperature can be considered constant in this process.

![Experimental device, 3D representation, Dimensions](image)

**Figure 1.** Bubble column geometry.

According to the literature [18], numerical setting of the domain is shown in table 1.

| Component               | Feature     | Details                      |
|-------------------------|-------------|------------------------------|
| Turbulence models       | Water       | Homogeneous Model k-ε        |
|                         | Air at 25°C |                              |
| Boundary conditions     | Inlet       | Normal speed 0.17m/s         |
|                         | Outlet      | Degassing condition          |
|                         | Wall        | No slip wall                 |

The qualitative and quantitative comparisons between experiments and numerical simulations are shown in figure 2 and figure 3. As is shown in figure 2, high gas hold-up area mainly concentrates on the inlet and gas accumulates on the top of the column both in experiment and simulation. In both cases, the water presents an upward flow trend in the column center and a downward flow trend near the wall, which denotes axisymmetric flow in the column. Quantitative comparison is shown in figure 3, where three different bubble column heights (H) in the center plane are selected, and the time averaged vertical liquid velocity of simulation agrees well with the experiment.
3. Boundary conditions and convergence settings

As is shown in figure 4, the geometric model is composed of the inlet pipe, the impeller, the diffuser and the outlet pipe. Some flow parameters, such as total flow rate $Q$, rotational speed $n$ and IGVF, affect the size and distribution of bubbles in the pump. Note that IGVF is the most influential parameters for the multiphase pump [19]. In the simulation, IGVF is from 5% to 15%, while the $n$ and $Q$ are kept unchanged as 2950r/min and 50 m$^3$/h. According to the observation with a high speed camera, the bubble size near the outlet of the diffuser is mostly 0.3mm-1.2mm when IGVF is 10% [20]. Based on the results, Yu [13] found that for condition of small volume flow rate, the numerical prediction agree well with the experiment when bubble diameter is 0.4mm, so the initial bubble diameter is selected as 0.4mm in this simulation.

**Figure 4.** Geometric model of multiphase pump.
The bulk mass flow rate and the average static pressure are set at the inlet and the outlet of the computational domain, respectively. No slip wall is used in the wall. The frozen rotor model is applied for the interface between the impeller and the diffuser, and this model can solve the multiple frame of reference problem well [21]. A high-resolution scheme with second–order accuracy is used for the discretizing, and the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) method is set to solve for the pressure and velocity.

4. Results and discussion

4.1. Comparison between experiment and simulation

The sketch of the experiment rig is shown in figure 5. The mixture of air-water is transported into the buffer tank to be mixed uniformly, then pumped by the multiphase pump, and finally carried into the water tank. The shell of the pump is made of organic glass, one high-speed camera and two light sources are applied for the observation of flow field.

![Figure 5. Test bench of multiphase pump.](image)

The head comparison between experiment and simulation is shown in Table 2. Here, the pump head is calculated by the formula given by [14]. It can be seen that the pump head decreases with the increased IGVF. The largest relative error lies in the case of IGVF=15% and it is below 5%, which demonstrates that head calculated by simulation agrees well with the head obtained by experiment.

| Component              | IGVF 5% | IGVF 10% | IGVF 15% |
|------------------------|---------|----------|----------|
| Head from Experiment   | 16.23   | 15.76    | 14.96    |
| Head from simulation   | 16.06   | 15.92    | 15.20    |

The bubble number density in simulation and experiment are similar under the condition where \( n = 2950 \text{r/min} \), \( Q = 50 \text{m}^3/\text{h} \) and IGVF is 15%. In the vicinity of interaction region, a large bubble number density appears because of the rotor-stator interaction. A small bubble number density exists in the blade suction side at leading edge of the diffuser and this can be observed both in simulation and experiment.
4.2. Analysis of simulation results

4.2.1. Bubble size and bubble number density in flow field

IGVF of 15% is selected to show the distribution characteristics of bubbles because similar flow field is obtained in other IGVF conditions.

Due to the rotational action of the impeller, the liquid with high density moves to the shroud of the impeller while the gas with low density moves to the hub of the impeller. As is shown in figure 7(a) and figure 7(b), the bubble number density and gas volume fraction near the hub is obviously larger than that near the shroud, so that the collision rate of bubbles increases and bubbles tend to show a large size. However, big bubbles break-up into small bubbles owing to the stirring action of the impeller blades. With a dynamic balance between the coalescence and the break-up of bubbles, the bubble size keeps almost constant in the whole impeller.

![Flow field of experiment](image1)
![Flow field of simulation](image2)

**Figure 6.** Bubble number density in simulation and experiment.

**Figure 7.** Contours of distribution characteristics in meridional plane

In the vicinity of interaction region, as is shown in figure 8, the mixture flows disorderly between the impeller and the diffuser. High collision rate of bubbles, which is produced by the rotor-stator
interaction [22, 23], causes a large bubble size (seen from figure 7(c)). It can also be observed from figure 7(a) and figure 7(b) that the bubble number density and gas volume fraction is still high in hub but low in shroud, which can be attributed to the influence of rotational action from the upstream impeller.

![Figure 8. Water velocity vector at the hub in the vicinity of interaction region.](image)

Bubbles spread gradually from hub to shroud of the diffuser as the cross-sectional area increases. Bubble diameter reaches a largest value in the diffuser. This can be analyzed by figure 9. On one hand, increased pressure along the streamwise direction enables a decreased bubble velocity; on the other hand, the change of the flow direction results in the formation of vortices. These effects contribute the collision rate of bubbles and large bubble diameter form (as is shown in figure 7).

![Figure 9. Contours of pressure and air velocity in meridional plane.](image)

In summary, when the mixture flows into the impeller, bubbles move to the hub of the impeller because of the rotational effect, while a homogeneous bubble size distribution can be obtained due to the stirring action of the impeller blades. In the vicinity of interaction region, high collision rate of bubbles is produced by the rotor-stator interaction and the bubble size increases. As the mixture reaches the diffuser, bubbles spread gradually from the hub to the shroud as the cross-sectional area increases, meanwhile with a decreased bubble relative velocity, the collision rate increases and a high
bubble diameter can be observed.

4.2.2. Distribution characteristics of bubbles in different IGVF

At the inlet of the inlet pipe, the highest IGVF means largest bubble number density in the condition of a same initial diameter, but at the outlet of the inlet pipe, as is shown in figure 10(a), the smallest bubble number density and the largest bubble diameter are observed at the highest IGVF. It can be concluded that a high IGVF means small distance between bubbles and a high collision rate of bubbles occur. This results in a large bubble diameter and a small bubble number density in the impeller inlet, as shown in figure 10(a) and figure 10(c).

In the diffuser, bubbles of high IGVF move slowly and thus the coalescence efficiency of bubbles increases. The bubble diameter and gas volume fraction of high IGVF is larger than that of low IGVF, which is verified by figure 10(a) and figure 10(b).

![Figure 10](image)

(a) Average bubble diameter  
(b) Average GVF  
(c) Average bubble number density

**Figure 10.** The bubble distribution characteristics in streamwise direction.

5. Conclusion

In this paper, bubble number density equation is introduced in the flow simulation of a multiphase pump, and the distribution of bubble size and bubble number density in different IGVF has been studied. The results can be summarized as follows

- Due to the rotational effect and the stirring action of the impeller blades, bubbles move to the hub and a homogeneous bubble size distribution can be obtained. Because of the rotor-stator interaction, collision rate of bubbles, as well as the bubble size, increase in the vicinity of interaction region. In the diffuser, bubbles spread gradually from the hub to the shroud as the
cross-sectional area increases. As the bubble velocity decreases, the collision rate increases and a high bubble diameter can be observed.

- In the inlet pipe, high IGVF means small distance between bubbles and large collision rate of bubbles, which results in a large bubble diameter and a small bubble number density at the inlet of the impeller.
- In the diffuser, big bubbles move slowly and thus the coalescence efficiency of bubbles increases in the condition of high IGVF. The bubble diameter and gas volume fraction reaches a largest value in the highest IGVF

Acknowledgments
This study was supported by the National Natural Science Foundation of China [grant no. 51579006], and Basic Research Foundation of Beijing Institute of Technology [grant no. 20150342012].

References
[1] Buwa V, Deo D and Ranade V 2006 Int. J. Multiphase Flow. 32 864–85.
[2] Darmana D, Deen N and Kuipers J 2005 Chem. Eng. Sci. 60 2556-75.
[3] Sokolichin A and Eigenberger G 1999 Chem. Eng. Sci. 54(13) 2273-84.
[4] Buwa V and Ranade V 2004 AIChE J. 50 2394-407
[5] Bech K 2005 Chem. Eng. Sci. 60 5294–304.
[6] Kocamustafaogullari G and Ishii M 1995 Int. J. Heat Mass Tran. 38(3) 481-93.
[7] Lehr F and Mewes D 2001 Chem. Eng. Sci. 56 1159–66.
[8] Bakker A and Akker H 1994 Chem. Eng. Res. Des. 72(A4) 594-606.
[9] Bakker A 1992 Thesis Technische Univ. 4-86
[10] Lane G, Schwarz M and Evans G 2005 Chem. Eng. Sci. 60(8) 2203-14.
[11] Kim J, Lee H, Kim J, Choi Y, Yoon J, Yoo I and Choi W 2015 J. Fluid Eng. 137(8) 081301.
[12] Zhang J, Li Y, Cai S, Zhu H and Zhang Y 2016 Adv. Mech. Eng. 8(4) 1-13.
[13] Yu Z, Zhu B and Cao S 2015 Eng. Computation. 32(7) 2166-80.
[14] Zhang J, Li Y, Vafai K and Zhang Y 2018 Int. J. Numer. Method H. 28(3) 763-784.
[15] Yu Z, Zhu B, Cao S and Liu Y 2014 Adv. Mech. Eng. 2014(2) 958352.
[16] Zhang W, Yu Z, Zahid M and Li Y 2018 Energies 11(5) 1069.
[17] Tabib M V and Schwarz P 2011 Chem. Eng. Sci. 66(14) 3071-86.
[18] Pfleger D, Gomes S, Gilbert N and Wagner H 1999 Chem. Eng. Sci. 54(21) 5091-99.
[19] Zhang J, Zhu H, Wei H and Peng J 2011 Appl. Mech. Material 52 399-404.
[20] Falcimaigne J, Brac J, Charron Y, Pagnier P and Vilagines R 2002 Oil Gas Sci. Technol. 57(1) 99-107.
[21] Zhang W, Zhu B, Yu Z and Yang C 2017 Adv. Mech. Eng. 9(5) 168781401770765.
[22] Zhang W, Yu Z and Zhu B 2017 Energies 10(2) 148.
[23] Zhang W, Yu Z and Zhu B 2017 Energies 10(5) 634.