Application of two-fluid model in the unsteady flow simulation for a multiphase rotodynamic pump

Z Y Yu¹, B S Zhu², S L Cao² and G Y Wang¹

¹ School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100081, China
² State Key Laboratory of Hydrosience and Engineering, Tsinghua University, Beijing 100084, China

E-mail: yuzhiyi@bit.edu.cn

Abstract. Based on the assumption of tiny bubbly flow, the gas-liquid two-phase unsteady flow in a multiphase rotodynamic pump was numerically simulated with two-fluid model. The two-phase transport process and the evolution characteristic of the pump head were analyzed. In the working conditions, the liquid flow rate was constant, and the IGVF (inlet gas volume fraction) was 0.05, 0.15 and 0.25, respectively. The $k \omega$ based SST model was used for turbulence; the drag force and the added mass force were accounted for in the interfacial momentum transfer terms. Because the wrap angle of the blade was large, the hybrid mesh was adopted to guarantee high mesh quality. The simulation results demonstrate that two-fluid model can more reasonably capture the transport process than the homogeneous model; and the drag law should be corrected based on the mixture viscosity in high gas volume fraction conditions. If the liquid flow rate is constant, the increase of IGVF can raise the pressure in the inlet extended region, while the pressure in the outlet extended region will not be affected much, thus the pump head will go down. In addition, due to the fluctuation of gas volume fraction field, the pump head will also fluctuate around a stable value in the transport process.

1. Introduction

As the key equipment of multiphase mixed transport technique, multiphase pumps are widely used in many industries such as petroleum, chemical engineering, food, municipal water supply and nuclear engineering. Because of the huge benefits brought by this technique, great efforts are made by many research institutes and oil companies to the study of multiphase pump. Among the pumps developed for gas-liquid multiphase transport, rotodynamic pump receives now a widespread acceptance because it can provide a large flow rate and it is applicable to multiphase flow containing small sand particles [1]. To improve the design of multiphase pump and meet the need of complicated two-phase flow transport, the two-phase flow process in the pump must be known through experiment or numerical simulation. Due to the financial and technical limitation, the former can now only be used in the observation of the flow in non-rotating parts or some special working conditions [2], while the latter has received more and more attention with the fast development of computer and CFD technology.

Usually, three multiphase models are used for gas-liquid two-phase flow simulation, i.e. homogeneous model, drift flux model and two-fluid model [3]. In two-fluid model, the gas and the liquid are both regarded as a continuous medium which occupies the whole flow field. The two phases
will couple with each other through the interaction of mass, momentum and energy according to the corresponding governing equations. Therefore, two-fluid model is more accurate relative to the other two models. With this model, people have already had some basic knowledge on the phase distribution in the pump when steady flow assumption is adopted, and they have acquired some basic strategies on the usage of two-fluid model\cite{4-6}. In the present paper, two-fluid model will be used in the analysis of the unsteady flow process in a multiphase rotodynamic pump based on the above studies. Also, the validity of the model will be evaluated.

2. CFD analysis method

2.1 Governing equations

Taking water and air as the two phases, assuming that the two-phase flow in the pump impeller is bubbly flow, the conservation equations of mass and momentum for unsteady incompressible turbulent flow can be written as follows in vector form:

\[
\frac{\partial}{\partial t}(\alpha_k \rho_k) + \nabla \cdot \left(\alpha_k \rho_k \mathbf{w}_k\right) = 0
\]  
\[
\frac{\partial}{\partial t}(\alpha_k \rho_k \mathbf{w}_k) + \nabla \cdot \left(\alpha_k \rho_k \mathbf{w}_k \mathbf{w}_k - \alpha_k \mathbf{\tau}\right) = -\alpha_k \nabla p + \mathbf{M}_k + \mathbf{f}_k
\]

where the subscript \(k = l\) or \(g\) denotes the liquid or gas phase. \(\rho_k\) is the density; \(\alpha_k\) is the volume fraction, which needs to satisfy the relation \(\alpha_g + \alpha_l = 1\); \(p\) is the pressure; \(\mathbf{w}_k\) is the relative velocity; \(\mathbf{M}_k\) is the interfacial force acting on phase \(k\); \(\mathbf{f}_k\) is the mass force relevant to the rotation of the impeller, which includes the centrifugal force and the Coriolis force; \(\mathbf{\tau}\) denotes the viscous stress tensor concerning fluid viscosity as well as turbulence viscosity.

The turbulence is modelled by SST model, i.e. \(k-\omega\) based Shear Stress Transport model. This model combines the advantage of \(k-\omega\) model in simulating near-wall flow and that of \(k-\varepsilon\) model in simulating flow outside the boundary layer. As it accounts for the transport of the turbulent shear stress and thus gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradient\cite{7}. Under this model, the formulation of eddy viscosity is:

\[
\mu_t = \rho_{\text{mix}} \frac{a_k}{\max\left(a_k \alpha, SF_2\right)}
\]

where \(\rho_{\text{mix}}\) is the mixed density and is calculated by \(\rho_{\text{mix}} = \alpha_l \rho_l + \alpha_g \rho_g\); \(a_k\) is a model constant and \(a_i = 5/9\); \(S\) is an invariant measure of the strain rate; \(F_2\) is a blending function; \(k\) and \(\omega\) are the turbulent kinetic energy and the turbulent frequency, respectively.

2.2 Interphase force

For the fully-developed bubbly flow, the main terms of the interphase forces are the drag force and added mass force. The drag force per unit volume for spherical gas bubbles is modelled as\cite{8}:

\[
M_{l,D} = -M_{g,D} = 0.75 C_D \rho_l \frac{\alpha_g}{D_b} \mathbf{w}_k \mathbf{w}_k
\]

where \(\mathbf{w}_k\) is the relative velocity between the gas bubbles and the liquid; \(D_b\) is the diameter of gas bubbles, and \(C_D\) the drag coefficient. For sparsely distributed fluid particles, the Schiller-Naumann correlation described below is usually used\cite{9}:

\[
C_D = \begin{cases} 
44, & \text{Re}_b < 1000 \\
0.15 \text{Re}_b^{0.687}, & \text{Re}_b > 1000 
\end{cases}
\]
where \( Re_b \) is the bubble Reynolds number given by \( Re_b = \frac{\rho D_b |w|}{\mu} \).

For dense spherical particle regime, the Ishii Zuber drag law[9] makes a correction by using the mixture Reynolds number to account for the dense particle effects, where the drag coefficient in the viscous regime is given by

\[
C_D = \frac{24}{Re_m} \left( 1 + 0.15 Re_m^{0.687} \right)
\]

Here, the mixture Reynolds number \( Re_m = \frac{\rho D_m |w|}{\mu_m} \), \( \mu_m \) is the mixture viscosity which is a function of gas volume fraction.

The added mass force, induced by the relative acceleration between the two phases, is modelled as[10]

\[
M_{l,VM} = -M_{g,VM} = \rho_l C_{VM} a_{VM}
\]

where \( C_{VM} \) is the added mass coefficient that can be assigned as 0.5 for spherical gas bubbles; and \( a_{VM} \) is the added mass acceleration vector written as

\[
a_{VM} = \frac{\partial w}{\partial t} + w \cdot \nabla w - w_1 \cdot \nabla w_1
\]

2.3 Mesh and numerical settings

The flow in a single flow passage is numerically simulated assuming that the flow in the impeller is axisymmetric. To set the boundary conditions reasonably, the inlet and outlet of the passage is extended along the axial direction. The software ICEM_CFD 12.1 is used for mesh generation. For the inlet and outlet extended region, structured mesh is adopted, and the number of cells in the inlet extended region and outlet extended region are \( I \times J \times K = 30 \times 40 \times 20 \) and \( I \times J \times K = 50 \times 40 \times 20 \), respectively, where \( I, J, K \) are defined as the axial direction from the inlet to the outlet, the circumferential direction from the pressure surface to the suction surface, the radial direction from the hub to the shroud, respectively. For the impeller region, as the wrap angle of the blade is large, unstructured mesh is used to guarantee high mesh quality, and the total number of elements is 431143. Figure 1 shows the computation domain and the 3D hybrid mesh employed in the flow analysis.

![Figure 1. Computation domain and mesh for flow analysis.](image)

As for the boundary conditions, the inlet velocity is given according to the flow rate, while an averaged static pressure is given to the outlet. At wall boundaries (the pressure and suction surfaces, the hub and shroud walls), the non-slip condition of viscous fluid is used for both phases. For the three pairs of circumferential boundaries, the rotational periodic condition is imposed.
For unsteady simulation, the steady solution of single water phase flow is specified as the initial flow field; the time step is 1e-3s and the maximum number of iterations per time step is 20; the total time of simulation is 0.6s.

2.4 Head prediction of the multiphase pump
In gas-liquid two-phase flow, the gas fraction must be taken into account for the calculation of external characteristics of a pump. If we denote $x$ as the mass flow rate ratio of gas to the two-phase mixture, the head of the pump operating under gas-liquid two-phase flow condition can be obtained by

$$H_{op} = (1 - x)H_i + xH_g$$  \hspace{1cm} (9)

The heads for the gas and liquid phases, $H_g$ and $H_l$, are expressed as

$$H_g = \frac{p_2 - p_1}{\rho_g} + \frac{(c_2^2 - c_1^2)}{2g}$$  \hspace{1cm} (10)

$$H_l = \frac{p_2 - p_1}{\rho_l} + \frac{(c_2^2 - c_1^2)}{2g}$$  \hspace{1cm} (11)

here, $p_1$, $c_1$, and $p_2$, $c_2$ are the static pressure and the absolute flow velocity at the impeller inlet and outlet, respectively.

3. Numerical results and discussions
With the two-fluid model and numerical settings described above, the 3D gas-liquid two-phase unsteady flow in the multiphase rotodynamic pump was simulated. The working conditions are listed in table 1. If we make a conversion, we can find that the liquid mass flow rates in table 1 are equal approximately, i.e. 8.35 kg/s, while the gas mass flow rates for the three cases are 0.0005 kg/s, 0.0018 kg/s and 0.0033 kg/s, respectively.

| Table 1. The working conditions in unsteady simulation with two-fluid model. |
|--------------------------|-----------------|-----------------|
| Rotation speed n (r/min)| Inlet gas volume fraction IGVF | Total volume flow rate Q (m³/h) |
| Case1                    | 1500            | 0.05            | 31.72          |
| Case2                    | 1500            | 0.15            | 35.45          |
| Case3                    | 1500            | 0.25            | 40.28          |

3.1 Effect of the drag law correction
The drag law plays an important role in the simulation of gas-liquid two-phase flow. For the three cases listed in table 1, the two drag models, i.e. Schiller-Naumann (S-N) and Ishii-Zuber (I-Z) drag models described in equation (5) and (6), are used to analyze the effect of the correction for the dense particle effects, where the bubble diameter is set as 1mm. The temporal evolutions of the pump head by the two drag laws are compared in figure 2.

The variation processes of the pump head are similar in different IGVF conditions. As the pure liquid flow field is specified as the initial flow field, the gas volume fraction in the flow passage in the earlier time is small and the head values are relative high. With the development of the gas volume fraction in the flow passage, the head value decreases quickly and then fluctuates around a stable value. In cases 1 and 2, the head fluctuations from the two drag laws are similar, while in case 3, the amplitude of the head wave from the I-Z model is much larger.
If the gas has not been completely transported to the outlet, the calculation of the head value by equation (9) does not make sense. From the simulation results, we know that the gas in the three conditions has all arrived at the domain outlet after the moment of \( t=0.32 \text{s} \), which will be shown in the next section. Therefore, to make a comparison with the experiment\(^\text{[11]}\), we can take an average over the head values after \( t=0.32 \text{s} \) as the pump head obtained from numerical simulation. Table 2 lists the head values in the three conditions from simulation as well as the experiment. For the cases with smaller IGVF (case1 and case2), the head predictions by the two drag laws are very close, while in case3 with higher IGVF, the I-Z model can greatly improve the head prediction. Therefore, the bubble swarm effect must be taken into account in simulation in higher IGVF conditions. In the following section, the I-Z model is used for higher reliability.

**Table 2.** Head values from simulation and experiment.

|                | Simulation (m) | Experiment (m) |
|----------------|---------------|----------------|
|                | S-N model     | I-Z model      |                |
| Case1          | 12.5          | 12.4           | 15.8           |
| Case2          | 11.1          | 11.0           | 13.2           |
| Case3          | 7.9           | 8.8            | 10.0           |

3.2 Distribution and evolution of gas volume fraction field

For the three cases listed in table 1, figure 3 to 5 show the contour distribution of gas volume fraction in the meridional surface as well as its temporal evolution. From these figures, we can see that the trajectories of gas in the flow passage are similar in the three cases. The gas will quickly accumulate to the shroud after it passes the domain inlet. When the gas flow near to the impeller region, it will move to the hub side in a short time and then go downstream to the outlet along the axial direction. The trajectories and distributions of the two phases are directly relevant to the forces acting on them. In the impeller region, both the liquid and the gas suffer centrifugal force because of the rotation of the impeller. However, the liquid phase (water) with larger density will suffer larger centrifugal force and thus mainly lies in the outer part of the passage, while the gas phase (air) is expelled by the liquid and most gas will gather near the hub with smaller diameter. After the two phases leave the impeller region, they will keep the characteristics of distribution because of the inertia, i.e. most gas will still lie near the hub. What is unclear is why the gas in the inlet extended region accumulates to the shroud, but the probable reason is the interphase force acting on the gas.
The gas volume fraction in the passage fluctuates in the mixed transport process, which can lead to the fluctuation of the pressure and velocity fields. From equation (9) to (11), it’s known that the fluctuations of inlet and outlet parameters will cause the corresponding change of the pump head, as is described in Sec. 3.1. From the results of the three cases, we can also know that with the increase of IGVF, the maximum value of gas volume fraction in the field also increase accordingly. At the moment of t=0.32s, this maximum value is 0.29, 0.59 and 0.76, respectively. At the same time, the stratification of flow becomes more evident. If IGVF is too high, then gas blocking phenomenon may occur, which is one of the main problems to be solve in the research and development of multiphase pump.

![Figure 3](image1.png)  
**Figure 3.** Temporal evolution of gas volume fraction field (IGVF=0.05).

![Figure 4](image2.png)  
**Figure 4.** Temporal evolution of gas volume fraction field (IGVF=0.15).

![Figure 5](image3.png)  
**Figure 5.** Temporal evolution of gas volume fraction field (IGVF=0.25).

Figure 6 shows the simulation result of the working condition of IGVF=0.25 with homogeneous model. It can be seen that the gas volume fraction field from this model is much more even than that from two-fluid model, as is shown in figure 5. From the homogeneous model, the phenomenon of gas accumulation to the shroud in the inlet extended region cannot be captured. Obviously, this is because the interphase forces are not considered in the homogeneous model and the velocity field is shared by the two phases. In the impeller region and the outlet extended region, we can still see that the gas will move to the hub and mainly lie near the hub surface, but this phenomenon is far from evident as that shown in figure 5. What’s more, with time elapsing, the two phases will gradually distribute in the whole domain. According to the actual operating conditions, the flow field attained by the homogeneous model is obviously not reasonable. Therefore, for the simulation of gas-liquid two-phase flow in the multiphase pump, two-fluid model is recommended.

![Figure 6](image4.png)  
**Figure 6.** Temporal evolution of gas volume fraction field attained by homogeneous model (IGVF=0.25).

3.3 Pressure variation along the flow passage
The variation of area average pressure from the inlet to the outlet in the flow passage is shown in figure 7. In this figure, “z/L” is the relative position along the axial direction, “Case1, t=0.32s” denotes the result of the first case listed in table 1 at the moment of t=0.32s, while "Case1, init" is the initial flow field of the case, i.e. the working condition of pure liquid flow. For convenience, two vertical bars denoted by “A” and “B” are used to separate the impeller region from the inlet extended region and outlet extended region.

It can be seen from figure 7 that the pressure increases along the axial direction mainly in the impeller region, while in the inlet and outlet extended region without the impeller effect, the pressure keeps approximately constant. As the liquid mass flow rates are about the same in the three working conditions, the pressure differences between the inlet and outlet in the initial flow field are approximately the same, i.e. 77 kpa. However, when the gas is injected into the domain, the pressure in the inlet extended region will increase apparently, while the pressure in the outlet extended region remains unchanged on the whole, and thus the pressure difference through the pump becomes lower. According to this figure, the pressure differences in the three cases (from Case1 to Case3) are 67 kpa, 55 kpa and 34 kpa, respectively. Therefore, we can draw a conclusion that in the condition of constant liquid flow rate, the pump head will become lower with the increase of IGVF, which is also drawn in reference [11].

4. Conclusions

In this paper, the unsteady transport process in a multiphase rotodynamic pump is simulated with the two-fluid model. According to the results, the two-phase flow field and the pump head as well as their temporal evolution are analysed. Comparing with the homogeneous model, the two-fluid model can more accurately and reasonably capture the mixed transport process by taking into account the interphase forces. Due to the bubble swarm effect, the drag law should be corrected based on the mixture viscosity and the correction can greatly improve the head prediction in high gas volume fraction. In the mixed transport process, the gas will accumulate to the shroud in the inlet extended region, while in the impeller region and the outlet extended region, the gas will lie mainly near the hub surface due to the centrifugal force and the inertia. Through the simulation of three IGVF working conditions, it is found that in the condition of constant liquid flow rate, the increase of IGVF can raise the pressure in the inlet extended region while the pressure in the outlet extended region remains unchanged on the whole, thus the pump head becomes lower. In addition, the results demonstrate that the head of the multiphase pump will fluctuate around a stable value in the transport process, which is directly relevant to the fluctuation of the gas volume fraction field.
Acknowledgments
This work was supported by National Natural Science Foundation of China (51009003) and Open Research Fund Program of State key Laboratory of Hydroscience and Engineering (sklhse-2013-E-01).

References
[1] Falcimaigne J, Brac J, Charron Y, Pagnier P and Vilagines R 2002 Rev. IFP 57(1) 99-107
[2] Hajem M, Morel R and Champagne J 2001 Revue Internationale de l'Eau 3-4 34-39
[3] Lin Z H, Li Y G, Lu J C, Xie Z W and Su X J 2001 Characteristics of vortex shedding in gas-liquid two-phase flow and the engineering application (Beijing: Chemical Industry Press) (in Chinese)
[4] Tremante A, Moreno N, Rey R, and Noguera R 2002 Journal of Fluids Engineering 124(2) 371-376
[5] Lu J L., Xi G and Qi D T 2003 Journal of Engineering Thermophysics 24(2) 237-240 (in Chinese)
[6] Yu Z Y, Wang G Y and Cao S L 2009 Frontiers of Mechanical Engineering in China 4(1) 53-59
[7] Menter F 1994 AIAA-Journal 32(8) 1598-1605
[8] Mandar V T and Philip S 2011 Chemical Engineering Science 66 3071-3086
[9] Ansys Inc. Ansys CFX-Solver Theory Guide, Release 13.0. Ansys, Inc, Canonsburg, PA USA
[10] Abdullah A K. 2005 J. Applied Mechanics 72(5) 801-802
[11] Cao S L, Yu Z Y and Wang G Y 2002 10(Special ed.) 346-349 (in Chinese)