Influence of inlet gas volume fraction on performance of a gas-liquid centrifugal pump

S N Yan, X Q Luo, L F Zhang, J J Feng, G J Zhu, S L Chen, D H He, C H Li

Institute of Water Resources and Hydro-electric Engineering, Xi'an University of Technology, No.5 South Jinhua Road, Xi'an, 710048, China

Abstract. In order to study the influence of inlet gas volume fraction on performance of a gas-liquid centrifugal pump, the CFD commercial software ANSYS CFX 16.0 was used to solve the three-dimensional turbulent flow field in a gas-liquid two-phase flow centrifugal pump. The non-homogeneous two-fluid model was chosen for the purpose of obtaining phase distribution and its influence on the pressure and velocity fields, and the SST k-ω model was chosen as turbulence model. The result showed that the performance of the pump is stable under single-phase flow operating points. Under gas-liquid two phase flow, with the increase of inlet gas volume fraction, the pressure increment would be reduced, and the internal flow would be unstable. As the inlet gas volume fraction reaches a certain value, surge would occur under small flow rate conditions. At this time, a large amount of gas was concentrated in the impeller flow channel, and the channel of the impeller was blocked evenly, which lead to the pump’s head dropped suddenly. Under design flow and large flow rate operating points, with the inlet gas volume fraction increase, the performance of the pump was relatively stable, and the head of the pump decreases slowly.

1. Introduction

The gas-liquid centrifugal pump is one of the most commonly used artificial lift methods in the petroleum industry. In practical application, the presence of gas causes head degradation, low efficiency and unstable operation. Previous studies have revealed that as the inlet gas volume fraction reached a certain value, the ESP (electrical submersible pump) capability of boosting pressure is deteriorated suddenly. This critical condition termed as pump surge which significantly affects pump’s operational stability and run-life. The main reason for the pump performance suddenly dropped from the single-phase flow operation point to the gas-liquid two-phase flow operation point is that the gas volume fraction obtained from the positive pressure gradient regions are considerably higher and unevenly distributed which lead to the flow characteristics of the pump changed.

The objective of this work is to study the effects of inlet gas volume fraction on a gas-liquid centrifugal pump under different two-phase flow operation points. In order to complete this task, we did a comparative analysis on the two-phase flow patterns and bubble behaviour inside the pump channels under different operational conditions.

2. Numerical Methodology

The commercial CFD code ANSYS CFX-16 was implemented on a single-stage gas-liquid centrifugal pump to obtain the pump performance under single- and two-phase-flow conditions.

2.1. Geometry
The experimental section of the prototype pump is shown in figure 1.

![Prototype of the pump](image)

**Figure 1.** Prototype of the pump.

The prototype pump is a 25-stage electrical submersible pump. The geometry specifications of this pump are summarized in table 1. The parameter of the design operating conditions are rotational speed \( n=3500 \text{ r/min} \), volume flow rate \( Q_d=26 \text{ m}^3/\text{h} \) and head \( H_d=25 \text{ m} \).

| Impeller inner diameter \( D_1 \) | 62 mm |
|---------------------------------|-------|
| Impeller outer diameter \( D_2 \) | 127 mm |
| Impeller blade number \( Z \)    | 7     |
| Impeller outlet channel height \( b_2 \) | 15 mm |
| Diffuser inner diameter \( D_3 \) | 150 mm |
| Diffuser outer diameter \( D_4 \) | 62 mm |
| Diffuse vane number \( Z_d \)    | 10    |

**Table 1.** Main geometrical and operating variables.

Each stage of the ESP pump is exactly the same, and the single-stage of this pump was simulated in this study. Figure 2 shows the three-dimensional model of the calculation model pump which including the inner pipe, impeller, diffuser and outer pipe four parts. The full channel computational model was simulated because it takes into account the periodic effect. In order to eliminate the influence of the inlet and outlet flow pattern on the internal flow field of the pump, both the inlet and outlet pipe have been extended for some distances [1-2].

![Computational model](image)

**Figure 2.** Computational model.
2.2. Grid generation and mesh independence validation

The grid is the basis of transforming the geometric model into the computational domain of the flow field to carry out the CFD calculation. Grid quality has a very important influence on the reliability of numerical simulation. Thus each domain was meshed by commercial software ICEM to obtain high-quality structural hexahedrons. The surface mesh of the impeller and the diffuser is shown in figure 3. The near wall surface is refinement because of the existence of the viscous layer, and the thickness of the first layer of the near wall surface is 0.005 mm.

![Grid example](image1)

(a). Impeller. (b). Diffuser.

**Figure 3.** Surface mesh the pump

In order to eliminate the influence of discrete errors caused by grid factors on the simulation results, a sensibility analysis was performed to guarantee the grid independence. In this paper, six computational models with different grid numbers was selected for mesh independence validation. The scheme 1 with very few grid numbers for comparison, and the grid number in the latter schemes is 1.5 times higher than the previous one. Figure 4 shows the mesh validation on the basis of the six schemes. As illustrated in figure 4, the fluctuation of the simulated pump head and efficiency with grid number of 2.24 million is within 1%. Therefore, it is indicate that the grid number has no effect on the numerical simulation results if the grid number of the pump reaches 2.24 million. Considering the accuracy of the calculation and the calculation time, we choose the calculation model with the grid number of 2.24 million for numerical simulation research.

![Grid validation](image2)

**Figure 4.** Mesh independence validation.

2.3. Mathematical model.

The computational fluid dynamics (CFD) commercial software was used to solve three-dimensional turbulent flow field in the centrifugal pump. The finite volume method based on finite element method
is used in the software, and the discrete equation is solved by the full implicit coupled algebraic multi-grid method.

The Navier-Stokes equations were solved by using ANSYS CFX 16.0. In this paper, it is assumed that the gas-liquid two phase flow pattern in the impeller of the centrifugal pump is bubbly flow, and it is consider no mass source or interfacial mass transfer between gas and liquid phase under gas-liquid two-phase flow conditions. The continuity equation [3] of gas-liquid two phase flow can be written as:

$$\frac{\partial (\rho \alpha_i)}{\partial t} + \nabla \cdot (\rho \alpha_i \mathbf{u}_i) = 0$$  \hfill (1)

Where the subscript $i = l$ or $g$ denotes the liquid or gas phase, $\rho$, $\alpha$, and $\mathbf{u}$ are the density, volumetric fraction, and velocity of the $i$th phase, respectively. The Gas volume fraction $\alpha_g$ is defined as:

$$\alpha_g + \alpha_l = 1$$ \hfill (2)

$$\alpha_g = \frac{Q_g}{Q_g + Q_l}$$ \hfill (3)

The momentum equation of gas-liquid two phase flow can be written as eq. (4):

$$\frac{\partial (\rho \alpha_i \mathbf{u}_i)}{\partial t} + \nabla \cdot (\rho \alpha_i \mathbf{u}_i \mathbf{u}_i) = -\alpha_l \rho \mathbf{g} + \nabla \cdot (\tau_{ii}) + \alpha_i \rho \mathbf{g} \mathbf{g} + F_i + F_{l,i} + F_{vm,i}$$ \hfill (4)

Where $\mu$, $P$, $F_i$, $F_{Lift,i}$ and $F_{vm,i}$ represent dynamic viscosity, pressure, drag force, lift force and virtual mass force. The Eulerian approach was chosen for the purpose of obtaining phase distribution and its influence on the pressure and velocity fields. This method considers the liquid-phase and gas-phase as statistical continuity, and it is the most widely used method at present. The Eulerian-Eulerian non-homogeneous model was selected as multiphase model which allows separate velocity fields for each phase, and the pressure field is shared by all fluids [2]. Particle model was used to solve the transport equations of two-phase medium because it is suitable for a phase medium as a continuous phase, and the other phase is a dispersed phase. The SST model which combines the advantages of the k-$\omega$ model in the near wall simulation and the k-$\epsilon$ model in the outer boundary layer was chosen as turbulence model. The SST model also considers the transport of turbulent shear stress on the basis of BSL k-$\omega$ model and gives a more accurate prediction on the flow separation phenomenon under the inverse pressure gradient [4].

2.4. Boundary conditions

Boundary conditions were set to close the solution equation. A total pressure condition was set at the inlet of the domain. This condition is more accurate because the inflow energy is defined and the simulator is allowed to obtain gradients of velocity and pressure. At the outlet of the domain the mass flow was specified. The inlet gas volume fraction (IGVF) is set at the inlet of the domain, which expressed as eq. (3). Zhu [1] found that the numerical simulation results are in good agreement with the experimental results by using constant initial bubble diameter if the inlet gas volume fraction is less than 10%. The inlet gas volume fraction studied in this paper is less than 10%, so the initial bubble diameter $d_0$ at the inlet of the pump is set to a constant value ($d_0=0.1\text{mm}$). The frozen-rotor technique which also referred as moving reference frame was used to simulate the interaction between rotating and stationary parts in the centrifugal pump. This technique offers a better compromise between the computational efforts and numerical accuracy. The non-slip condition (relative velocity equal to zero) was specified in every wall of the domain.

The following basic assumptions were made in the process of CFD calculation:

- The velocity at inlet of computation domain is uniform distribution.
- The bubbles at inlet of computation domain are uniformly mixed.
- The gas phase and liquid phase are incompressible.
- The gas phase and liquid phase are dissolution with each other.
3. Results and Discussions

3.1. Single-Phase Simulation Results
The external characteristic curve of the model pump under single-phase flow conditions (IGVF=0%, n=3500 r/min, P_{tot,inlet}=5MPa) is shown in figure 5. The horizontal axis represents volume flow rate, the vertical axis on the left represents pressure increment and the vertical axis on the right represents efficiency.

![Figure 5. External characteristics of the model on single-phase simulation.](image)

The pressure increment and efficiency as functions of flow rate within the pump operation points under single-phase flow conditions is summarized in figure 5. Figure 5 shows that the pressure increment decreases with the increase of the flow rate, and the efficiency reaches the maximum value under design condition. This trend is consistent with the actual operation laws of the pump.

3.2. Two-Phase Simulation Results
CFD simulation was carried out on the pump to investigate the influence of the inlet gas volume fraction on the gas-liquid centrifugal pump based on Eulerian-Eulerian non-homogeneous model. The pump rotational speed is 3500 r/min, and the inlet pressure of the pump is 5 MPa. Different operation points (liquid flow rate ranging from 0.6 Q_d to 1.6 Q_d) with the inlet gas volume fraction from 1% to 10% have been calculated.

![Figure 6. Pressure increment versus inlet gas volume fraction](image)

![Figure 7. Pressure increment at small flow rate conditions](image)

Figure 6 represent the curves of pressure increment changing with the inlet gas volume fraction at the design and large flow operation points. It can be observed that the pressure increment of the gas-
liquid two-phase flow centrifugal pump gradually decreases with the increase of the inlet gas volume fraction at design point and large flow rate conditions. The reason is that the separation between gas and liquid is becoming more and more serious with the increase of gas volume fraction, and the instability phenomenon in flow field increases, thus the performance of the pump is reduced.

The pressure increment of the inlet gas volume fraction under small flow conditions are presented in figure 7. The pressure increment of the centrifugal pump trends to decrease with the increase of the inlet gas volume fraction. It is obvious that when the inlet gas volume fraction is increased from 3% to 4%, the pressure increment curve suddenly drops. When the gas volume fraction is 4%, the pressure increment at 0.6 \( Q_d \) condition is reduced by 14% compared with the inlet gas volume fraction of 3%, and it is reduced by 17% under the 0.8 \( Q_d \) condition. It is known as the pump surge when the centrifugal pump operating under the operating condition of the gas-liquid two phase flow. Through the above research, it is found that the effect of surge on the performance of ESP is very large, so it is necessary to study the mechanism of pump surge.

![Figure 8. Distribution of gas void fraction at Span=0.5, \( Q_l=0.6Q_d \).](image)

In order to find the reason of the pump surge, the internal flow field of the model pump was analyzed. Under the condition of gas-liquid two-phase flow, the decrease of ESP performance is mainly caused by the increase of the inlet gas volume fraction. In order to compare the difference between the surge before and after the surge, the internal flow field of the centrifugal pump was analyzed at the liquid flow rate \( Q_l=0.6Q_d \) operation point, and the inlet gas volume fraction is 3% and 4% respectively. Figure 8 shows the gas volume fraction distribution at 50% relative leaf height (Span=0.5) flow surface expansion diagram. Span is defined as the dimensionless distance from the shroud to hub [5]. Streamline distribution at 50% relative leaf height under 0.6 \( Q_d \) is presented in figure 9. The pressure load distribution of the middle streamline on the blade surface with different inlet gas volume fractions under 0.6\( Q_d \) are plotted in figure 10. The water velocity vector in meridian surface of impeller under 0.6 \( Q_d \) are plotted in figure 11.

As shown in figure 8, the gas phase is prone to accumulate in impeller and diffuser channel. There are seven channels in the impeller, which was marked on the picture. Figure 8 (a) shows that the accumulation of gas phase in Channel 3 is serious, channel 1, Channel 2, Channel 4 and Channel 5 is slighter, Channel 6 and Channel 7 with very little gas phase accumulation. We can see from figure 8 (b), the gas phase accumulation in Channel 1 and Channel 2 is very serious, almost blocking the flow channels, Channel 3, Channel 4 and Channel 5 is comparatively serious. By comparing the figure 8 (a) and (b), it is found that when the inlet gas volume fraction is 4%, the phenomenon of gas accumulation is more serious, especially in figure 8 (b), and the gas phase in Channel 1 and Channel 2 almost blocked the whole flow path. From the above analysis, it is implied that the main reason for the pump...
surge is that a large number of gases accumulate in the flow Channel, which will lead to the blocking of the flow channel, and cause the unsteady flow of the flow field, thus the performance of the pump reduced suddenly.

Figure 9 shows the streamline distribution at 50% relative leaf height, and the working conditions and the section positions are the same as in figure 8. Channel 1 to 7 in figure 9 corresponds to Channel 1 to 7 in figure 8. It can be seen from figure 8 and figure 9 that the channel with more gas accumulation will seriously affect the direction of water, and vortices will occur in severe cases. In addition, the water velocity distribution under IGVF=4% operation point is worse than IGVF=3% operation points, and more vortices are generated in the same channel under IGVF=4% operation points.

![Figure 9. Water velocity distribution at Span=0.5, 0.6 Q_d](image)

Figure 10 shows the pressure load distribution curve on the blade surface between Channel 1 and Channel 2 at 50% relative leaf height. At lower flow rate $Q_l=0.6Q_d$ conditions, there is a considerable pressure difference between IGVF=3% and IGVF=4% operation points. The pressure difference at IGVF=3% operation points on the pressure and suction sides are more uniform. Near the leading edge, there is a saddle region on the pressure side. On the suction side of the leading edge, the pressure increases slowly to the trailing edge without any pressure drop. However, the pressure difference at IGVF=4% operation points on the pressure and suction sides are very uneven. The pressure difference on the pressure and suction sides is about 0 to 30 kPa when stream wise less than 0.55, and about 80 to 100 kPa when stream wise from 0.65 to 0.85. We can see from figure 10, increasing the inlet gas volume fraction would result in the decrease of the impeller blade surface pressure and uneven pressure distribution on the surface of the blade.

![Figure 10. Pressure distribution on the blade surface at Span=0.5, 0.6Q_d](image)
Figure 11 shows the velocity vector in meridian surface of impeller at 0.6 $Q_d$. The streamline is more smoothly under IGVF=3% operation points. However, there is a large amount of vertex under IGVF=4% operation points, as shown in figure 11(b) reded with circles. It is mainly because a large number of gases gathered in channel, blocking the flow path, and water could not flow smoothly, resulting in a large number of vortices.

![Figure 11. Water velocity vector in meridian surface of impeller at 0.6 $Q_d$](image)

4. Conclusions

In this paper, the influence of inlet gas volume fraction on the performance of the pump is studied by numerical simulations. The following conclusions can be drawn:

- The numerical simulation results of single phase flow coincide with the actual operation rule of the pump, so the results of the numerical simulation are credible.
- Under gas-liquid two phase flow, the operation of the pump is stable at large flow and design conditions. However, with the increase of inlet gas volume fraction, the pump will surge and the performance of pump suddenly reduced;
- Under small flow conditions, it is indicate that with the increase of inlet gas volume fraction, there will be a large number of gas blockages in the impeller channel, resulting in uneven pressure load on the blade surface, and some large vortices generated in the flow channel.
- In this paper, the influence of inlet gas volume fraction on the performance of the pump was studied by numerical simulation, which has important reference value for the future gas-liquid centrifugal pump design.

5. References

[1] Zhu J J, ZHANG H Q 2014 CFD simulation of ESP Performance and Bubble Size Estimation under Gassy Conditions. SPE Technical Conference and Exhibition, Amsterdam, the Netherlands.

[2] Caridad J, ASUAJE M, KENYERY F, et al. 2008 Characterization of a centrifugal pump impeller under two-phase flow conditions. Journal of Petroleum Science and Engineering. Vol 63. P18-22.

[3] Zhang J Y, Cai S J, et al. 2014 Numerical investigation of compressible flow in a three-stage helicon-axial multiphase pump. Transactions of the Chinese Society for Agricultural Machinery. Vol. 45. P89-95. (in Chinese)

[4] Yu Z Y, Liu Y. 2013 Character analysis of unsteady gas-liquid two-phase flow in a multiphase rotodynamic pump. Transactions of the Chinese Society for Agricultural Machinery. Vol. 44. P66-69. (in Chinese)

[5] Liu H L, Liu D X. 2012 Applicative evaluation of three cavitating models on cavitating flow calculation in centrifugal pump. Transactions of the CSAE. Vol.28 P54-59. (in Chinese)

Acknowledgments

The authors would like to thank the National Natural Science Foundation of China (Grants No. : 51527808, 51339005, 51679195) and the Natural Science Fundamental Research Program Funding Project of Shaanxi Province (Grant No. 2018JM5102) for their financial support.