

Numerical calculation and analysis of gas-liquid two-phase flow in a centrifugal multiphase pump for deep-sea mining

C H Li, X Q Luo, L F Zhang S H Sun and S N Yan

Institute of Water Resources and Hydro-electric Engineering, Xi'an University of Technology, Xi'an, 710048, China

E-mail: drleech_xaut@126.com

Abstract. With the development of offshore oilfield, the gas-liquid centrifugal multiphase pump (CMP) has become the focus of research for its more effective way of working and economic benefits. In order to transport two-phase media, the internal flow of gas-liquid multiphase pump is complicated. In this study, the gas content effect on CMP performance was investigated by computational fluid dynamics (CFD). Gas and water are flowed through a DN127 CMP at varying gas content. This paper based on the Euler Model, the numerical simulation and analysis of three stages gas-liquid two-phase pump internal flow field are conducted, with the standard $k-\epsilon$, SST and SSG turbulence model. The influence of different turbulence models on the numerical results and the effects of different gas-liquid fraction on the characteristics of pump and internal flow law are analyzed. The results shows that the standard SST turbulence model is more suitable for the numerical calculation of gas-liquid two-phase flow; and at pump best efficiency point (BEP), the head of the multiphase pump decreases by 4.5%~26% and the efficiency of CMP decreases by 4.6%~20% when the gas content increases from 0% to 20%, the CMP becomes ineffective when the gas content is higher than 40%.

1. Introduction

With the rapid development of the industry, the land resources increasingly exhausted, abyssal sea, as the last unexploited area on earth, is rich in minerals, oil and gas resources. From the beginning of the 20th century, countries all around the world has conducted extensively research on the deep-sea oil and gas resources exploitation technology. Because the oil and gas is difficult to measure on the bottom sea, they must be transported to the surface of ocean or the coast for measurement, therefore, the undersea oil and gas blended transportation technology is proposed. Moreover, deep-sea oil and gas mixing pump is one of the key technologies.

Centrifugal pumps have been widely used in industries, agriculture and domestic applications for a long time, but their operational requirements are now becoming stricter and stricter. Its impeller design demands a detailed understanding of the internal flow at rated and part load operating conditions [1]. Centrifugal multiphase pump (CMP) is designed to sending medium including oil, gas and water at the same time, in order to reduce the transportation cost. In recent years, as the core equipment of multiphase transportation technology, multiphase flow pump is a set of conventional liquid pump and gas compressor performance in one of the multiphase transportation device [2]. In the numerical calculation of gas-liquid two-phase turbulence, Wu yu-lin et al, have made a lot of productive work. He calculated the gas-liquid two-phase flow situation in the impeller by using the dual fluid turbulence model, and discovered that the impeller outlet pressure will decrease in the case



of two-phase flow, by comparing the single-phase flow of oil and gas-liquid two-phase flow, and the flow distribution is uneven when flowing in single phase. In order to get pressure inside the impeller and more specific velocity distribution it still need to analyse the gas phase and liquid phase respectively [3]. In two-phase flow pump performance of prediction based on CFD, Minemura et al, analysed the two-phase flow in a centrifugal impeller by using three-dimensional finite element method and adopting bubbly flow in the dual fluid flow model, and through the analysis of the model, discovered that is only applicable to the condition of very low gas content [4,5]. Therefore, he proposed a fixed cavitation bubbly flow model, which showed that the bubbles gathered and attached to the surface of impeller in the high gas content and formed a new calculation boundary condition, but the calculation results are still not ideal, the model also needs further improvement [6].

In this study, for gas-liquid multiphase pump to deliver two phases medium, there will be a separation of two-phase medium within the multiphase pump in the process of conveying medium. The three-dimensional (3D), steady-state Reynolds-Averaged Navier–Stokes (RANS) equations with standard SST (shear stress transport) turbulence models are solved in ANSYS CFX by employing frozen-rotor technique [3]. With high-quality structured hexahedral mesh, the numerical simulation results are given. The results of the pump numerical simulation in small flow rate condition and the different gas content is analysed. At pump best efficiency point (BEP), the efficiency of CMP decreases 4.5%~40% when the gas volume fraction (GVF) increases from 0%~20%. The vortices exist near the suction side middle part of impeller vane and the pressure sides of diffuser vane. In gas-liquid two-phase flow condition, the lift of CMP decrease, which because the trapped air bubbles not only reduce the cross-sectional area of the flow channel, which increase the relative velocity of the liquid flow, but also increase the flow losses.

2. Calculation model and Numerical methodology

The main process of numerical calculation at flow field in the deep-sea multiphase centrifugal pump includes the following contents. With the help of UG three-dimensional (3D) modelling software, 3D hydraulic model is drawn according to the 2D wooden patterns. In order to realize the discretization of continuous flowing space, the whole computational domain of the pump is divided into structured grids. Then, the solution setting is performed in the ANSYS CFX-Pre, including initial conditions, boundary conditions, turbulence model, convergence conditions, time step and realizing the discretization of time; furthermore, the solver is used to solve the internal flow field of the whole pump, and the flow characteristics of centrifugal pump are analysed.

2.1. Geometry and meshing

In this paper, by using the UG 3D modeling software, the three-dimensional hydraulic model of CMP, including the impeller, diffuser, extension of import and export, are constructed respectively. The entire geometry includes three pump stages, In addition, each stage comprises of a channel wise-sliced diffuser and impeller, on which the structured hexahedral grids are generated with Turbogrid16. The meshes of inlet pipe, outlet pipe and cavity connection pipe are conducted by commercial code ANSYS ICEM 16, which are all structured quadrilateral elements. Thus, the spatial discretization of the entire three -dimensional flow field is realized.

There are 7 blades and 10 vanes in impeller and diffuser for each stage. The single stage multiphase pump geometry modal is shown in Figure1. The major geometrical specifications are listed in Table 1. At the best efficiency point (BEP), the operating parameters are: rotary speed $N=3500$ rpm, design mass flow rate $Q_r=7.22$ kg/s, single stage design hydraulic head $H=26$ m and rated efficiency $\eta=70\%$.

Table 1. Geometrical specifications of simulated the 3 stages multiphase pump.

component	Description	Values
-----------	-------------	--------

Impeller	Number of blade (Z_i)	7
	Inlet diameter (d_1 ,mm)	62.23
	Outlet diameter (d_2 ,mm)	127
	Blade thickness (b_i , mm)	3.789
	Inlet channel height (h_1 ,mm)	14.06
	Outlet channel height (h_2 ,mm)	7.42
Diffuser	Number of blade (Z_d)	10
	Blade thickness (b_d , mm)	5.532
	Inlet diameter (d_3 ,mm)	150
	Outlet diameter (d_4 ,mm)	65.49

Figure 1 displays the CMP single model, including impeller blades (figure 1(a)), diffuser blades (figure 1(b)), and the entire single-stage assembly (figure 1(c)). Since the flow fields inside centrifugal pump are axisymmetric [7], a single channel can be used to save computational cost and improve numerical efficiency [8]. Thus, the computational domains of impeller and diffuser can be stream wisely sliced into 1/7 and 1/10 as shown in figure 3(a) and (b). Similar configurations have been numerically implemented on a 3 stages CMP to study pump two-phase performance under gas-liquid flow conditions.

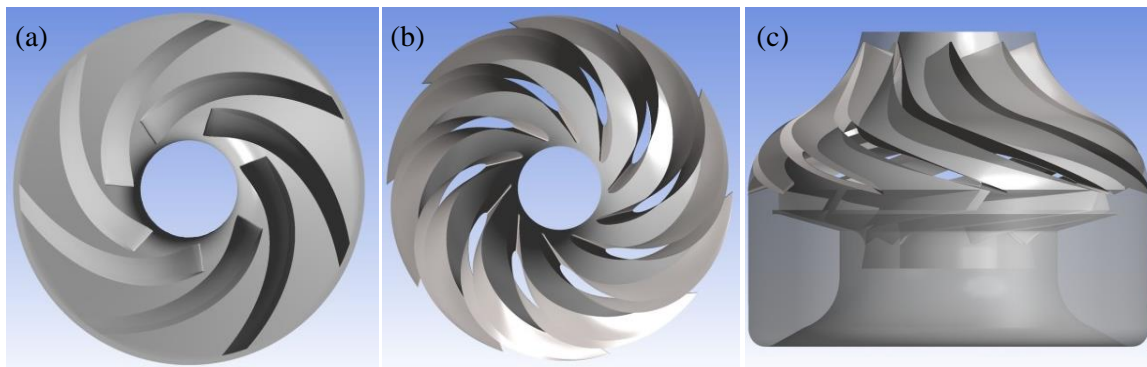


Figure 1. Geometries of single stage,(a)impeller blades,(b)diffuser vanes, (c)entire 3D assembly

2.2. Governing equations

In computational fluid dynamics (CFD), a set of conservation equations are solved based on the continuous medium presumption, a fundamental hypothesis that treats fluid medium and motion infinitely differentiable both in time and space domains. In this study, the isothermal condition is applied to the fluid flow domain. Therefore, the conservation equation of energy can be omitted. The mass conservation equation is given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (1)$$

Where ρ , \vec{u} are liquid density and velocity vector. The source in mass conservation equation is not taken into account. The momentum conservation equation is written as:

$$\frac{\partial (\rho \vec{u})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla P + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + S \quad (2)$$

Where $\vec{\tau}$ is the stress-strain tensor given in equation (3). \vec{g} is the gravity acceleration vector, S is external forces. For fluid flow in centrifugal pump, $S = S_{Cor} + S_{cfg}$, S_{Cor} and S_{cfg} represent the Coriolis force and centrifugal force effects. For stationary frame of reference, $S_{Cor} = S_{cfg} = 0$. For a rotating frame with constant angular velocity (Ω), $S_{Cor} = -2\rho \vec{\Omega} \times \vec{V}$ and $S_{cfg} = -\rho \vec{\Omega} \times (\vec{\Omega} \times \vec{r})$, where $\vec{\Omega}$ and \vec{r} are angular velocity vector and position vector, respectively.

$$\vec{\tau} = \mu(\nabla \vec{u} + (\nabla \vec{u})^T) + \left(\lambda - \frac{2}{3}\mu\right) \nabla \cdot \vec{u} \vec{I} \quad (3)$$

The RANS equations are used in CFD solver, which statistically average the turbulence fluctuations by decoupling instantaneous velocity vectors. The additional Reynolds stress terms in RANS equations are modeled by two-equation turbulence model. Several turbulence models are available in literature, including standard $k-\varepsilon$ (Launder and Spalding, 1974), RNG (renormalization group) $k-\varepsilon$ (Yakhot et al., 1992), standard $k-\omega$ (Wilcox, 1998), BSL (baseline) $k-\omega$ (Menter, 1994) and SST $k-\omega$ (Menter, 1994), among others. As recommended by ANSYS (2016), the SST $k-\omega$ two-equation turbulence model is applied here due to its ability of handling separation flow and resolving flow very close to walls.

2.3. Numerical schemes and boundary conditions

The water and air are used as high-speed centrifugal multiphase pump transmission medium, in this study. Therefore, the computational domain is selected for fluid domain (Fluid); In order to control the flow rate, the inlet boundary of inlet pipe is set to the quality of import (Mass Flow Rate), the outlet pipe outlet is arranged for pressure outlet boundary condition (Static Pressure), which has the better convergence results when the reflux occurs inside the impeller [9].

The frozen-rotor algorithm is used to simulate interactions across the interfaces of impellers and diffusers. This model treats each component of computational domain with an individual frame of reference, while it keeps the relative orientation of these components across the interface fixed. It requires the least amount of computational effort compared to other interface models. However, the frozen-rotor model is unable to capture transient effects at the frame change interface due to its steady state nature. In our simulation, the axisymmetric property of CMP geometries is used by assuming periodic flow characteristics if pump working condition is stable. The stream wise-designed blades and vanes inside CMP provide additional compensation that further weakens interactions across impeller-diffuser interfaces. Thus, the frozen-rotor algorithm is used as it offers an acceptable compromise between computational effort and numerical efficiency.

Due to the simplified geometries of impeller and diffuser, the grids at interface are non-conformal and mismatching with different pitch angles. In consideration of this, the GGI (general grid interface) mesh connections are employed, which permit nonmatching of grids on either side of the two connected surfaces (ANSYS, 2016).

Boundary conditions are specified according to the corresponding experimental configurations from CMP inlet to outlet, for all wetted walls, the no-slip velocity condition is imposed. For steady-state simulation, a false time step as a means of under relaxing governing equations is applied, which requires a relatively large time scales due to robust and fully implicit CFX-solver. A fixed physical timescale of $1/(\Omega)$ is used with maximum 1000 outer loop iterations to achieve convergence. The convergence criterion is satisfied if RMS (root mean square) residual drops below 10^{-4} .

The hexahedral mesh has 6404550 elements and 6892470 nodes totally. The structured hexahedral grids are shown in figure 3, including single flow channel of impeller (a) and diffuser (b).

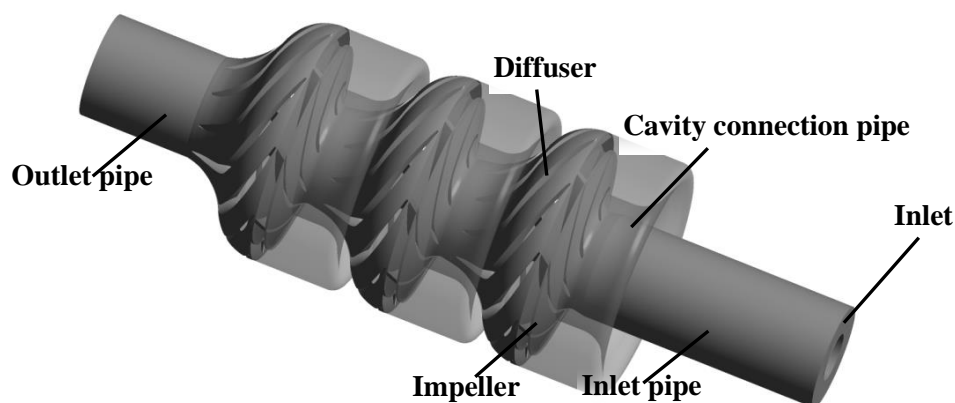


Figure 2. CFD computational domain of the 3 stages multiphase pump

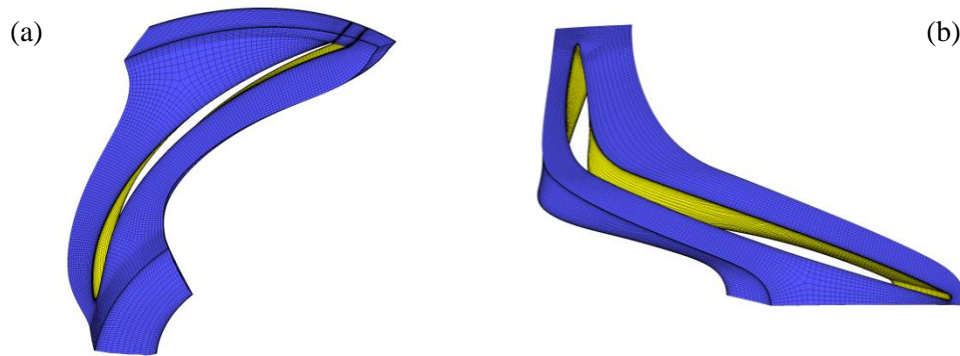


Figure 3. Grid generation, (a) impeller channel, (b) diffuser channel

3. Results and discussions

In this section, the overall performance of CMP numerical simulations at different flow condition and gas contents is presented. At first, the overall performance of CMP is calculated and analyzed at the best efficiency point, the performance of the three-stage multiphase pump and every stage pump is analyzed under different flow conditions of the same medium, respectively. Then, at the best efficiency point, the mixed model of gas phase uniform distribution is used for numerical calculation to analyze the overall performance of centrifugal multiphase pump when the gas content is different at gas-liquid two-phase flow. The outputs from CFX-post include the performance of pump, pressure and velocity fields, streamlines. Seven flow rate conditions and five gas contents are used to conduct numerical simulations, namely, 1.0Q_r, 0.85Q_r, 0.7Q_r, 0.6Q_r, 0.5Q_r, 0.34Q_r, 0.13Q_r and 3%, 5%, 10%, 15%, 20%.

3.1. Mesh and turbulence model validation

The mesh quality depends on the dimensionless distance (y^+) at the first grid point near the wall. According to boundary layer theory, the viscous sub-layer exists in the near-wall region. Within viscous sub-layer, the dimensional velocity (u^+) is a logarithmic function of y^+ away from the wall. This is also referred to as standard wall-function, which holds for $y^+ < 200$. Thus, the first grid layer should be sufficiently fine to meet wall function requirement.

The mesh number is counted on single-stage simplified fluid domains of impeller and diffuser. As shown in figure 4, the simulated stage head becomes constant when the grid number reaches about 0.3 million, where the average value of y^+ on blade surface is below 70. Therefore, the grids used for simplified impeller and diffuser sections consist of 271,374 and 290,336 elements, respectively. Total grids for the complete three-stage computational domain contain 6404550 elements, which are sufficient to guarantee grid independence.

Figure 5 shows the effect of turbulence models on simulated stage head variation and comparison with corresponding design results under water flow. The selection of turbulence model is a delicate task for CFD simulation, which is also a compromise of computational effort and numerical accuracy. As it can be seen in figure 5, there is no prominent trend variance among different turbulence model predictions. However, the aforementioned SST turbulence model is used due to its advantages in capturing shear flow spreading and separation at low Reynolds number flow, and because the SST model is closer to the trend of design results, especially near the design flow area.

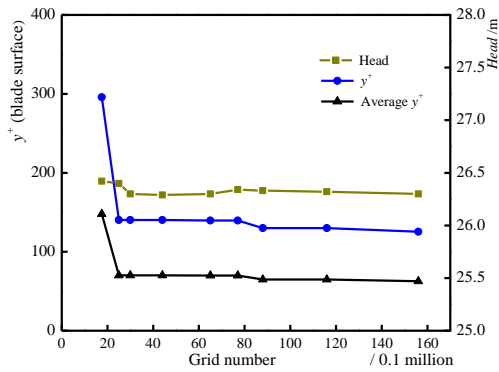


Figure 4. Mesh validation

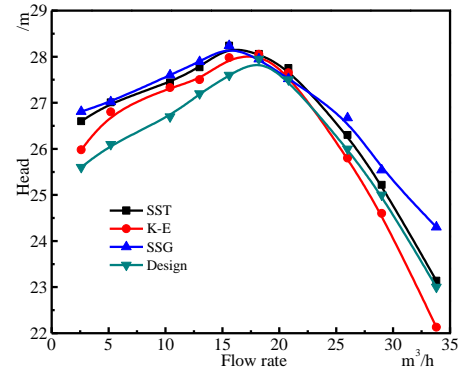


Figure 5. Influence of turbulence models on single-phase water simulation

3.2. Analysis of pump performance under low flow rate condition

In this section, based on steady numerical simulation results of CMP in the case that the shaft power, rotational speed and medium temperature are constant, we can obtain the calculation condition prediction of centrifugal pump lift and hydraulic efficiency. Then, the lift and hydraulic efficiency variation trend of CMP were analysed under different flow rate conditions, and analyze the interrelationship between them.

For comparison of simulated three-stage CMP overall performance with every stage performance, the dimensionless variables: flow coefficient and head coefficient are defined by equations (4) ~ (5):

$$\text{Flow coefficient: } \varphi = \frac{Q}{\Omega d_2^3} \quad (4)$$

$$\text{Head coefficient: } \psi = \frac{gH}{\Omega^2 d_2^5} \quad (5)$$

Where d_2 is impeller diameter (m), and the Ω is CMP rotary speed (rad/s)

As figure 6(a) shows that the total lift of three-stage deep-sea centrifugal multiphase pump can be approximately equal to the sum of each stage pump head, but the numerical calculation results is slightly less than the sum. When the flow rate increase, the total lift decrease, this may be due to the neglect of leakage flow through the radial clearance between impeller and diffuser, which causes additional boosting pressure loss in reality; and there is a cavity connection pipe between the various stages. Meanwhile, the smooth wall assumption also contributes to the deviation by underestimating wall shear stresses. Among them, the head variation trend of the 2nd stage pump is consistent with the total lift, which can predict the variation of pump lift in the middle stage of three-stage deep-sea pump tends to be a stable trend. The head and efficiency curve of the all levels pump changes rapidly when the flow rate is too small. The reason is that the pump is sucked in too much gas, which affects the flow field in pump and can lead to the danger of cavitation, thus reducing the performance of pump. All of that indicates that a small flow rate will break the first stage flow field of the three stages deep-sea pump and an unstable flow will occurs.

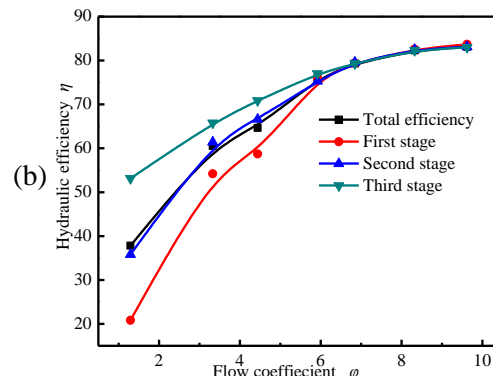
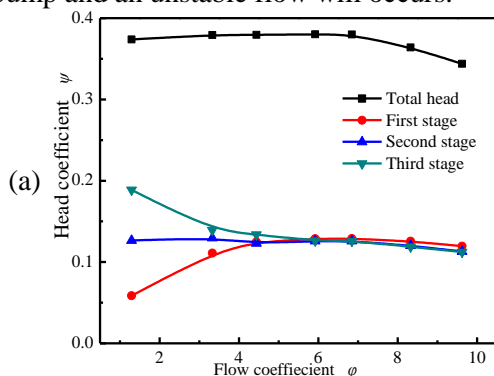


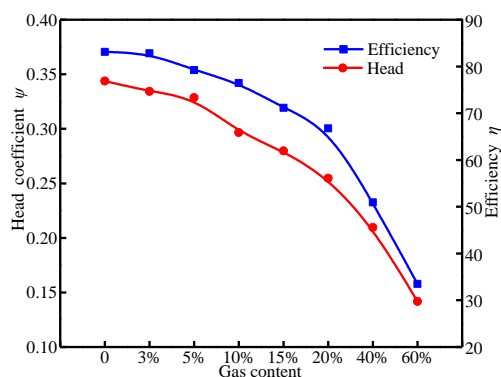
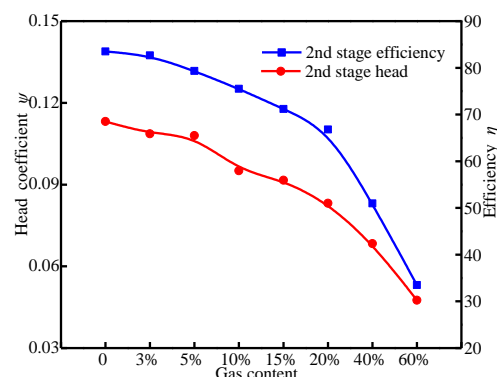
Figure 6. Head curves(a) and Efficiency curves(b) of CFD simulated CMP performance

Figure 6(b) summarizes the change tendency of hydraulic efficiency with the flow rate variation. As we can see, the change trends of all efficiency curves are consistent, and all curves are declines with the decrease of flow rate. A good agreement can be found in the comparison for total efficiency curve and the efficiency curve in the second stage pump. Furthermore, this phenomenon of the change trend is very similar to the lift curve that is the same trend as the intermediate level change trend, which confirms that the external characteristic of the three-stage deep-sea CMP and the intermediate ones are the same change tendency. When the flow rate is too small, the first stage pump performance curve decreases rapidly with flow rate decrease, but the characteristic curve of the final pump can still be maintained at a better level, the efficiency value is 53.12%.

3.3. Influence of gas content

In this paper, the three-stage centrifugal multiphase pump is used as the research object, the performance parameters of the centrifugal multiphase pump are obtained by changing the gas content, which is used to studying the influence of gas content on pump performance, in the condition that the rotary speed and the flow rate are constant. This study provides a strong evidence for the analysis of the flow field inside the centrifugal multiphase pump in the future, which can be used to help design of multiphase pumps according to the influence, later. In this study, the different gas content conditions are researched by numerical simulation under the same flow rate. The 25°C air was selected as gas phase and the standard density of 1000 kg/m³ water was selected as liquid phase in the calculation, and the mixture model of gas phase homogeneous continuous distribution is used in the calculation.

From figure 7 shows that in the gas-liquid mixed transportation, the lift of pump and the efficiency curve showed a gradually decreasing characteristic with the gas content rate rise. When the rotary speed is not changed, this is because the higher gas content, the smaller density of the mixed medium and the lower energy of the multiphase medium is obtained during the rotation, thus the lift and efficiency of pump decrease. When the gas content more than 20%, the head curve drops rapidly and the gradient of velocity varies greatly. As shown in figure 8, the performance curves of the second stage multiphase pump are in good agreement with the overall performance curve of the pump. At pump best efficiency point (BEP), the head of the multiphase pump decreases by 4.5%~26% and the efficiency of CMP decreases by 4.6%~20% when the gas content increases from 0% to 20%, the CMP becomes ineffective when the gas content is higher than 40%.

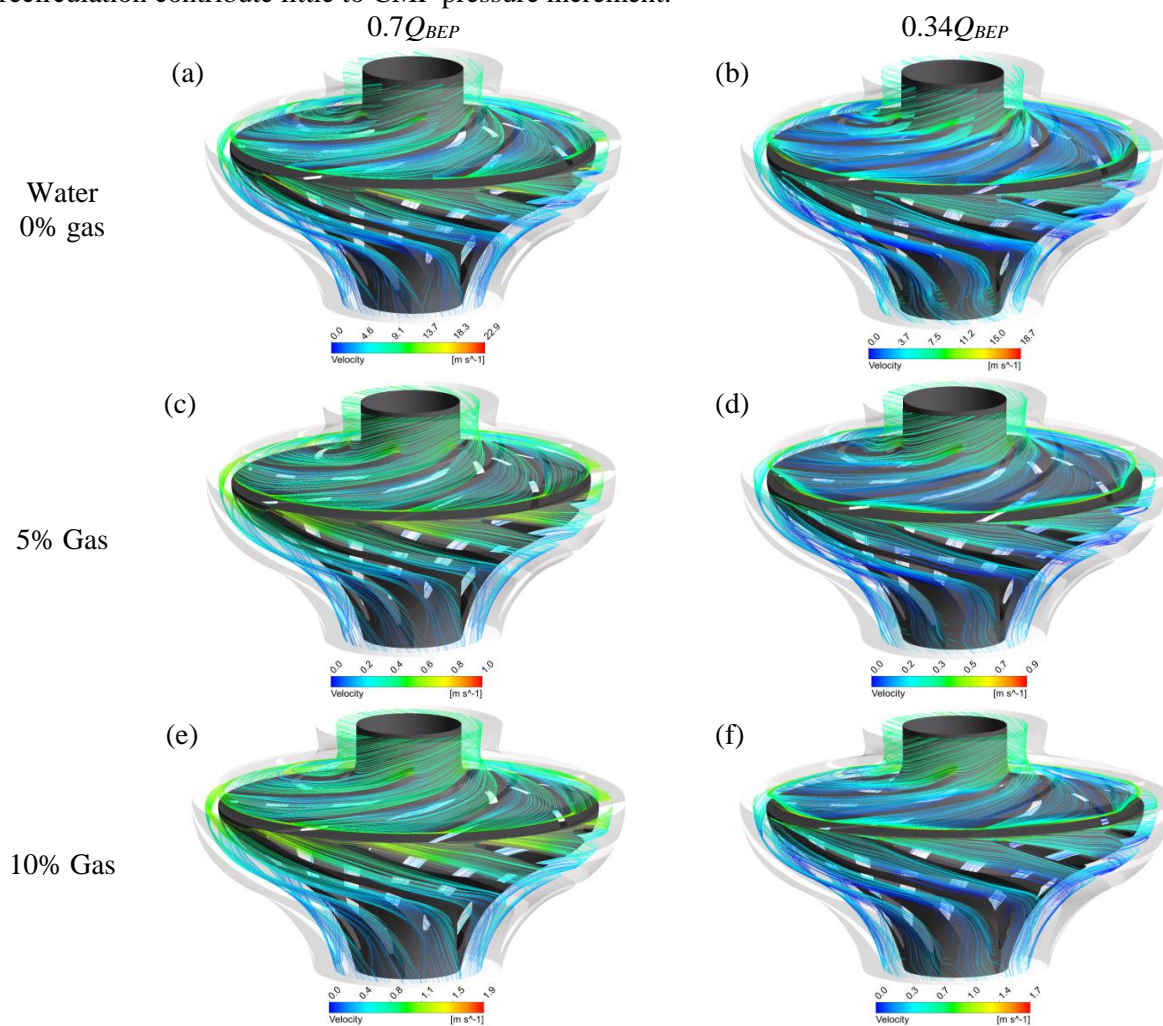
**Figure 7.** three stages overall performance curve**Figure 8.** the second stage performance curve

3.4. Flow analysis under different gas content

As shown in Figure 9, under the gas-liquid two-phase mixture transportation, the liquid flow streamlines plots under different flow conditions inside the 2nd stage. When the pump is used to convey the gas-liquid two-phase medium, the decline of pump lift is closely related to the retention of

air bubbles in the impeller. By analyzing the liquid streamlines distribution diagram of centrifugal multiphase pump in gas-liquid two-phase flow, we can know that vortices obviously exist near the suction side middle part of impeller vane, in small flow rate conditions, when the gas content increase gradually inside the pump impeller. This is caused by the increase of air content, and the flow channel in impeller is blocked by the accumulation of gas. The retention of air bubbles in the impeller leads to the decrease of the pump lift that is because the trapped air bubbles not only reduce the cross-sectional area of the flow channel, which increase the relative velocity of the liquid flow, but also increase the flow losses.

For the simulated cases shown in figure 8, the vortices still exist near the pressure sides of diffuser vane due to highly twisted vane geometry. The vortex shape inside diffuser channel is affected by several factors, including rotary speeds, gas content and liquid flow rates. However, this vortex and recirculation contribute little to CMP pressure increment.



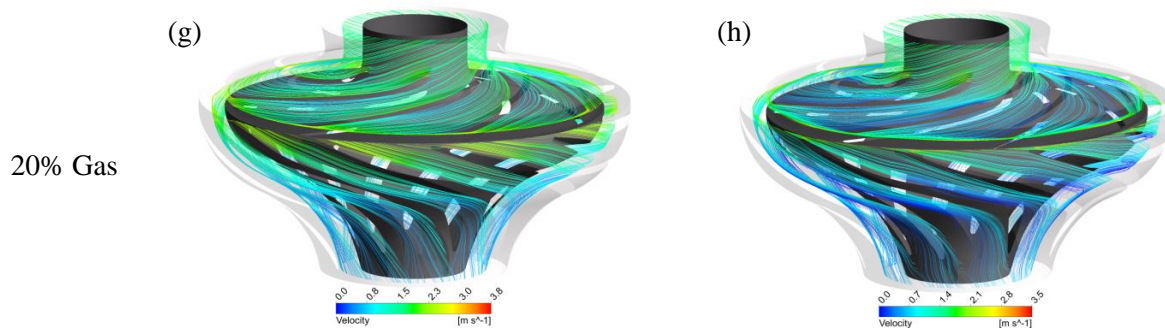


Figure 9. The liquid flow streamlines comparison under different flow conditions at half span of stage 2

4. Conclusions

In this paper, the gas content effect on a CMP overall performance is investigated through CFD simulations. Based on the detailed analyses, the following conclusions can be drawn:

- For 3D numerical simulations, the hydraulic performance of the centrifugal multiphase pump hydraulic model can be simulated accurately by using the standard SST turbulence model.
- When the CMP works under the two phases flow conditions of gas and liquid, the flow channel in impeller is blocked by gas accumulation, the flow regime deteriorated, the lift and efficiency of the pump decrease obviously with the increase of the gas content. The vortices exist near the middle part of impeller and diffuser vane, when working under the small flow rate conditions.
- Under the small flow rate conditions, the performance of the first and third stage pump in the three-stage CMP is relatively poor by comparing to the intermediate pump. However the change tendency of second stage pump performance curve and the overall performance of the CMP are consistent. Due to the neglect of leakage flow through the radial clearance between impeller and diffuser, which causes additional boosting pressure loss in reality, the total lift of the CMP is slightly less than the sum of each stage pump head.

The following work will be done in the future:

- The transportation of more types of medium will be studied. Especially, the performance of this multiphase pump will be researched in the transportation of oil and natural gas.
- The 7 stages and 25 stages multiphase pump will be applied to the experimental study.
- Flow visualizations should be done in model test to validate the conclusions in this study.

Acknowledgements

The authors would like to thank the National Natural Science Foundation of China (NSFC, No.51527808) for their financial support.

References

- [1] Usha P, and Syamsundar C, 2010, Computational analysis on performance of a centrifugal pump impeller, *Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power*, India, TM-07.
- [2] Kaixin L, 2012, Optimization design and numerical simulation of gas-liquid multiphase pump
- [3] Yulin W et al, 1998, Three-Dimension Calculation Of Oil-Bubble Flows Through a Centrifugal Pump Impeller, *Proceedings of The Third Internal Conference On Pump and Fan*, Tsinghua University Press, 526-532
- [4] Wugao Z, and Dunsong X, 1995, The research progress of multiphase flow pump abroad, *Pump Technology*, (6):7-13
- [5] Kiyoshi M, and Tomomi U, 1993, Three-Dimension Calculation of Air-Water Two-Phase Flow in Centrifugal Pump Impeller Based on a Bubbly Flow Model, *Journal of Fluids Engineering (Transaction of The ASME)*, 115(4):781-783

- [6] Kiyoshi M, and Tomomi U, 1994, Three-Dimension Calculation of Air-Water Two- Phase Flow in Centrifugal Pump Impeller Based on a Bubbly Flow Model with Fixed Cavity, *JSME Internal Journal, Series B*, 37(4):726-735
- [7] Zhu, J., Zhang, and H.-Q., 2014, CFD simulation of ESP performance and bubble size estimation under gassy conditions. *SPE Annual Technical Conference and Exhibition, SPE 170727, Society of Petroleum Engineers.*
- [8] Caridad, J, Asuaje, M., Kenyery, F, Tremante, A and Aguillon, O, 2008, Characterization of a centrifugal pump impeller under two-phase flow conditions, *J. Pet. Sci. Engineer*, (63):18-22
- [9] Houlin L, Shouqi Y and Weidong S et al, 2003, Study on three-dimensional solid modeling of hydraulic element of centrifugal pump, *Pump Technology*, (3):22-24 Stein P, Sick M and Dörfler P K 2006 Numerical simulation of the cavitating draft tube vortex in a Francis turbine Proceedings of 23rd IAHR Symposium on Hydraulic Machinery and Systems Yokohama Japan.