

**MULTI-PHASE FLOW MODELING IN A
HELICO-AXIAL ELECTRIC SUBMERSIBLE
PUMP OPERATING AT HIGH SPEED AND
WITH HIGH GAS VOLUME FRACTION**

BY

ZHENGHUA SHI

A Thesis Presented to the
DEANSHIP OF GRADUATE STUDIES

KING FAHD UNIVERSITY OF PETROLEUM & MINERALS

DHAHRAN, SAUDI ARABIA

In Partial Fulfillment of the
Requirements for the Degree of

MASTER OF SCIENCE

In

MECHANICAL ENGINEERING

December, 2021

KING FAHD UNIVERSITY OF PETROLEUM & MINERALS

DHAHRAN- 31261, SAUDI ARABIA

DEANSHIP OF GRADUATE STUDIES

This thesis, written by **ZHENGHUA SHI** under the direction of his thesis advisor and approved by his thesis committee, has been presented and accepted by the Dean of Graduate Studies, in partial fulfillment of the requirements for the degree of **MASTER OF SCIENCE IN MECHANICAL ENGINEERING**.



Dr. Al-Athel, Khaled Saleh
Department Chairman



Dr. Suliman Saleh Al-Homidan
Dean of Graduate Studies

Date



Dr. Ben-Mansour, Rached
(Advisor)



Dr. Al-Sarkhi, Abdelsalam
(Member)



Dr. Badr, Hassan M
(Member)



© Zhenghua Shi

2021

This thesis work is dedicated to my beloved parents, Qing and Zhao, who have always loved me unconditionally and supported me to achieve my goal.

.

ACKNOWLEDGMENTS

I would like to express my sincere gratitude to my advisor Dr. Ben-Mansour, Rached at Mechanical Engineering Department for his patient guidance during the research. I also appreciate my thesis committee members Dr. Al-Sarkhi, Abdelsalam and Dr. Badr, Hassan M for their suggestion. In addition, I would like to thanks my parents and friends who inspire and help me during my master degree. Without their understanding and encouragement all through my studies, it would be impossible for me to achieve my goal.

TABLE OF CONTENTS

ACKNOWLEDGMENTS	V
TABLE OF CONTENTS.....	VI
LIST OF TABLES.....	VIII
LIST OF FIGURES.....	IX
ABSTRACT	XI
ملخص الرسالة	XII
CHAPTER 1 INTRODUCTION.....	1
1.1 Background	1
1.2 Multiphase Pump	2
CHAPTER 2 LITERATURE REVIEW	5
2.1 Literature Review	5
2.2 Objectives	13
CHAPTER 3 MATHEMATICAL MODELING AND NUMERICAL METHODS.....	15
3.1 Mathematical Modeling	15
3.2 Geometry Modeling	22
3.3 Numerical Methods.....	26
CHAPTER 4 RESULTS AND DISCUSSION	29

4.1	Validation Results.....	29
4.2	Original Pump.....	32
4.2.1	The Effect of the Bubble Diameter.....	32
4.2.2	Pump Simulations on Various Conditions	34
4.2.3	Tow-Phase Flow Behaviors	36
4.3	Modified Pump.....	39
4.3.1	Modified Impeller Models	39
4.3.2	Performance Comparison between Original and Modified Impellers	42
4.3.3	Tow-Phase Flow Behaviors	43
4.4	Oil-air Two-phase Flow Simulation	47
CHAPTER 5 CONCLUSIONS AND RECOMMENDATIONS		50
5.1	Conclusions	50
5.2	Recommendations	51
NOMENCLATURE		53
REFERENCES.....		56
VITAE.....		62

LIST OF TABLES

Table 1 Designed selected variables	24
Table 2 Solution Methods and Boundary Conditions	28
Table 3 Bubble sizes used for CFD simulation in different inlet GVFs and rotational speeds.....	36

LIST OF FIGURES

Figure 1 Meridional plane of the basic model	23
Figure 2 Blade angles of the impeller and diffuser	24
Figure 3 Grid of the impeller and diffuser	25
Figure 4 Whole computational domain of the multiphase CFD model	26
Figure 5 Pump performance Comparison of experimental data (Kim et al. 2015b) and CFD multiphase model numerical results (flow rate: 100m ³ /h, rotational speed: 3600rpm).....	31
Figure 6 The effect of the bubble size on pump pressure rise by CFD multiphase model (flow rate: 100m ³ /h and rotational speed: 3600rpm	34
Figure 7 Predicted performance by CFD multiphase model versus different inlet GVFs using different assumed bubble sizes at different rotational speed (flow rate: 100m ³ /h) with experimental data (flow rate: 60m ³ /h) by Shi et al. (2018).....	36
Figure 8 Gas distribution of the impeller and diffuser on the meridional plane (flow rate: 100m ³ /h, rotational speed: 3600rpm and inlet GVF is 50 %).....	38
Figure 9 Gas distribution of the impeller on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 3600rpm and inlet GVF is 50 %).....	39
Figure 10 Gas distribution of the impeller and diffuser on the meridional plane (flow rate: 100m ³ /h, rotational speed: 3600rpm and inlet GVF is 50 %).....	39
Figure 11 Gas distribution of the impeller on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 6000rpm and inlet GVF is 50 %).....	39

Figure 12 Split vane impeller model and mesh ($\delta = 2\text{mm}$)	41
Figure 13 Modified impeller model with 5 holes on each blade	41
Figure 14 Predicted pressure rise comparison between modified pumps and original pump by CFD multiphase model versus different inlet GVFs (flow rate: 100m ³ /h, rotational speed: 3600rpm).....	43
Figure 15 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 3600rpm and inlet GVF is 10 %)	45
Figure 16 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 3600rpm and inlet GVF is 50 %)	46
Figure 17 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 6000rpm and inlet GVF is 50 %)	47
Figure 18 Gas distribution of the impeller with rectangle holes on different span-wise planes (flow rate: 100m ³ /h, rotational speed: 6000rpm and inlet GVF is 50 %)	47
Figure 19 Predicted pressure rise comparison between modified pumps and original pump with oil and air versus different inlet GVFs and rotational speeds (flow rate: 100m ³ /h).....	49

ABSTRACT

Full Name : Zhenghua Shi
Thesis Title : Multi-phase Flow Modeling in a Helico-axial Electric Submersible Pump Operating at High Speed and with High Gas Volume Fraction
Major Field : Mechanical Engineering
Date of Degree : December 2021

Multiphase pumps are widely applied in offshore oil-gas exploitation. In this study, the numerical simulations are performed to investigate the multiphase flow through the single-stage helico-axial rotating multiphase pump. Commercial CFD software ANSYS-CFX is employed to solve the Navier–Stokes equations for the gas-liquid two-phase flow. The mixture flow of air and water is selected as the studied medium and the effect of gas bubble size on the differential pressure is investigated. The handling ability at different rotating speed with different inlet GVF, especially high region conditions, is very vital for evaluating the pump performance. Then the modifications for the impeller blade are designed for the development of the performance. Different types of liquid-gas fluids pairs are also used to compare, such as water-air, oil-air.

ملخص الرسالة

الاسم الكامل: Zhenghua Shi

عنوان الرسالة: نمذجة التدفق متعدد المراحل في مضخة غاطسة كهربائية بمحور حلزوني تعمل بسرعة عالية وبجزء كبير من حجم الغاز

التخصص: هندسة ميكانيكية

تاريخ الدرجة العلمية: ديسمبر 2021

يتم تطبيق المضخات متعددة الأطوار على نطاق واسع في استغلال النفط والغاز في الخارج. في هذه الدراسة ، تم إجراء عمليات المحاكاة العددية للتحقق من التدفق متعدد الأطوار من خلال مضخة دوارة متعددة الأطوار حلزونية Navier-Stokes لحل معادلات ANSYS-CFX التجاري CFD محورية أحادية المرحلة. يتم استخدام برنامج للتدفق ثنائي الطور للغاز السائل. يتم اختيار تدفق الخليط من الهواء والماء كوسيط مدروس ويتم فحص تأثير حجم مختلف ، خاصة GVF فقاعة الغاز على الضغط التفاضلي. تعد قدرة المناولة بسرعات دوران مختلفة مع مدخل ظروف المنطقة العالية ، أمرًا حيويًا للغاية لتقييم أداء المضخة. ثم تم تصميم التعديلات على شفرة المكره لتطوير الأداء. تُستخدم أيضًا أنواع مختلفة من أزواج سوائل الغاز السائل للمقارنة ، مثل الماء والهواء والزيت والهواء

CHAPTER 1

INTRODUCTION

1.1 Background

Nowadays energy is the primary issue of common concern among human society, and the empirical data from 69 countries reveals the marked increment in energy consumption, especially in the developing countries (Sharma 2011). Due to the intensive demand and growing consumption on traditional fossil fuel, on one hand, more measures should be taken to improve the overall energy utilization efficiency; on the other hand, the exploitation of nature sources deserves more attention. Therefore, the sea becomes a good alternative to supply abundant energy for economic development. Since the natural resources in the adjacent sea becomes gradually exhausted, more and more offshore plants are expanded to the deep sea.

If the oil fields are operated only by natural pressure, the crude extraction period have to be expanded, which leads to large cost on the plant facilities, labors, etc. It is no doubt that natural pressure alone is not efficient, and pressurization by pump systems is proposed to improve the oil production rate. In the offshore oil well, the pumps are key equipment

which are applied to develop oil in diverse systems, according to their demands and operation. The pumps selected for offshore oil fields are dissimilar to general pumps with reference to transporting flow and working conditions. As a result, various studies and experiments have been conducted for the design and development of these pumps. In order to transfer the petroleum production, which is the mixture of oil and gas, multiphase pump is developed for the offshore plants. Compared with the traditional methods, this system has a great number of advantages such as simplified structure, low investment cost, high multiphase flow rate, and great tolerance of solid particles. With a wide scope of operation conditions, multiphase pump can effectively boost the pressure of multiphase fluids without the requirements for production separation, single-phase pumps, and gas compressors. Due to its pressurizing and transporting the mixture of the liquid phase and the gas phase simultaneously, the multiphase pump is widely equipped for petroleum industrial applications, playing a vital role in offset plants (Saadawi 2007; Lastra and Johnson 2005; Hua et al. 2011). However, considering the complicated interphase movement between liquid-gas two-phase flow, a series of interaction problems have yet to be resolved and further investigation is still required for improving the efficiency of transporting multiphase flow.

1.2 Multiphase Pump

Multiphase flow transport technology is designed to transport the oil-gas flow through only one pipeline, which has the advantage over two pipelines for transporting gas and liquid

separately. It contributes to the cost savings of petroleum production and the lifetime extension of the oil field at once. However, the rotodynamic types still have some disadvantages in the process of transporting multiphase flow: the separation of gas and liquid resulted from different centrifugal forces, especially at high rotational speed and with high inlet GVF. When the mixture flows into a high speed rotodynamic multiphase pump, centrifugal force is in a dominant position. Due to the large difference of density between gas and liquid, it can cause the separation phenomenon of the multiphase flow. The fluids flow towards the shroud by large centrifugal force while the gas phase accumulates around the hub along the passage of the impeller, and finally even lead to gas pocket. Then the fluids flowing area in the channel decreases, causing the relative motion of the two-phase flow increase, the loss of the hydraulic energy as well as the deterioration of the pump performance.

For the multiphase pump, the key issue is the complicated movement interaction as the multiphase flow changes continuously resulted from the influence of the gas volume fraction (GVF). Common types of multiphase pump utilized in the oil field include the twin screw pump, the centrifugal pump, the electrical submersible pump (ESP) and the helico-axial pump. For the twin screw types, they are a positive displacement pump transferring the pumped fluid from the suction side to the pressure side. When pump works, two contra-rotational screws intermesh and form close chambers with the pump casing. Then these chambers filled with flow move in an axial direction. With such double chamber structure, they can be operated at very high inlet gas volume fraction (GVF) condition, and on certain

condition it is up to 100% GVF. However, due to the positive displacement method, they are easily affected by the foreign matters such as the sand, and the requirement for maintenance and repair which restrict their application and development (Müller-Link and Rohlfing 2009; Dorenbos, Müller-Link, and Jäeschke 2001). The others three pumps are rotodynamic types. The centrifugal types with radial impeller are designed for low inlet GVF, which is usually below 10% (Gulich 2014). ESPs have good pressure performance in petroleum industry, but they still have inferior inlet GVF ranges since they consist of the multistage centrifugal impellers. The use of semi-axial impeller on ESPs allows 40% GVF on the certain condition (J. Zhu and Zhang 2017; 2018). As to the helico-axial types, they have the advantages in large handling capacity on the high inlet GVF operational conditions while the maintenance is simple compared with the twin screw pumps (Jinya Zhang, Zhu, and Wei 2011). As a result, the market share of helico-axial pumps in subsea multiphase pump applications is up to 65% (Andrade, Sangesland, and Stanko 2016). In this study, one single-stage helico-axial rotodynamic multiphase pump is selected for CFD simulations. The analysis on the flow patterns of gas-liquid mixture is necessary for the pump development. Hence the separation of the mixture flow and gas concentration distribution can be further understood. As a results, effective guidance and modifications can be proposed for the improvement on the boosting capacity of the pump.

CHAPTER 2

LITERATURE REVIEW

2.1 Literature Review

Multiphase pump was initially proposed in 1990s and confirms its capacity and stability on pressure boost for crude production (Dal Porto and Larson 1997). Due to its great pressurization performance, multiphase pump has received a widespread acceptance of petroleum industry, and more investigations have been conducted for the pump improvement. Helico-axial type is a typical rotodynamic multiphase pump based on the axial compressor. The type generally has the multi-stage design, each stage containing a rotational impeller and a stationary diffuser. Multiphase flow into the pump at the axial direction can ease the separation of the liquid and gas due to centrifugal force in the rotational impeller; then the diffuser is able to eliminate the centrifugal force, adjust the velocity field and convert kinetic energy to static pressure of the flow. A lot of experimental studies are made on the helico-axial pump for developing performance. The multi-functional test-bed with helico-axial multiphase pumps was designed and manufactured to analyze the performances of the studied pump on the pure water and air-water two-phase flow condition (Q. Li and Xue 2000). The rotational speed, inlet GVF as well as suction pressure were analyzed with test results. Zhu et al. (2004) proposed a regressive equation by the complete experiment data of the long period. It helped investigate and summary the

effect of different compress cells at high rotational speed, which provided an example for well field production and design in further. Zhu et al. (2007) investigated a study experimentally on the influence of operating parameters during a long period. The studied parameters included rotational speed, volume flow rate, and GVF. Then the helico-axial model was developed based on the optimization of overall construction design and prepared for the following promotion on petroleum industry market. During the well exploitation period, the multiphase pumps play an important role in high production efficiency and stable operating condition. The complicated flow characteristics of mixture inside the pump plays a great role in the performance; therefore, the flow pattern is worth more investigation. One helico-axial multiphase pump with water-air two-phase mixture flow was self-developed to start an experimental study on the external characteristics (Z. Li, Miao, and Li 2008). The curves of pump external characteristic were plotted according to the data on the different inlet GVF conditions as well as pure water condition. The experimental results claimed that the designed pump operated perfectly on the working condition and hydraulic loss was low when inlet GVF was below 50%. Another liquid-gas two-phase rotodynamic pump was designed and built to test experimentally and analyze the transient pressure characteristics of the pump (Xu et al. 2019). It was concluded that for pure water flow and water-air mixture flow, the blade passing frequency was dominant in the multiphase pump, which was resulted from interaction of the rotor and the stator. In the downstream impeller region, the pressure fluctuation became significantly feeble due to the splitter-blade structure with better controlling capacity of flow characteristic. Besides, the pressure fluctuation with water-air mixture was much more obvious than that on pure

water condition, and because of the complicated movement of bubbles the impeller rotating frequency correspondingly increased. High-speed photograph technology was utilized to study the relationship between transient pressure and phase distribution inside the impeller. It shows that as the two-phase fluids flow into the impeller and pressure gradually rises along the passage, the relative movement and phase separation between air and water cause the extensive pressure fluctuations.

Due to the great development of technology and science, computational fluid dynamics (CFD) technique has become a widely accepted method to study pump performance under different expected conditions. Owing to the complex internal structure of the multiphase pump, the flow velocity and pressure fields are hard to directly analyze through the experiment. Therefore, CFD simulations are utilized to evaluate the hydraulic performance and track the complicated flow momentum in a rotational multiphase pump. A CFD multiphase model was established for the prediction of pump performance with various geometry parameters on the changeable working conditions (Ban, Zhao, and Xue 2000). The gas distribution and static pressure along streamline of impeller passage were also investigated using this model. By comparing simulation results with experimental data, it was indicated the predicted results had a great agreement with experimental data when inlet GVF was below 30%. Zhang et al. (2014) studied numerical analysis on one three-stage helico-axial pump under the different inlet GVF conditions at steady and unsteady state. It was illustrated that GVF distribution decreased from the 1st stage to the 3rd stage. As the inlet GVF was extremely low or exceeded 80%, there was no much difference of the

pressure rise among these three stages which demonstrated it could be considered as incompressible flow. For inlet GVF from 20% to 90%, the compressibility of gas was not to be neglected. And it was found that the pressure decline occurred at the interface of the impeller and diffuser due to the interference between the rotor and stator. Boyarshinova et al. (2019) compared two CFD mathematical models by using the incompressible two-phase flow: volume of fluid (VOF) model and segregated flow model. It was indicated the segregated model had a better predicting ability of the pump performance and a handling capacity of higher inlet GVF range when compared with the VOF model which is a simplified model. A numerical method of dynamic mode decomposition was developed for multiphase flow pattern in a three-stage pump (Liu, Tan, and Cao 2020). It revealed that the dominant frequency was occasioned by the interaction of the impeller and diffuser. Flow parameters including volume rate, GVF and inlet angle were investigated in a multiphase CFD model on a five-stage pump (Y. Shi and Zhu 2020). These parameters all decreased along each stage caused by the compressibility and were largely influence by rotating speed and inlet GVF of the first stage. The difference of these variables almost changed little from the inlet to the outlet of the studied pump when inlet GVF is below 10% and above 90%. And it became much more obvious at 20-80% inlet GVF. The tip clearance also has a large influence on the pressurization performance and flow pattern. Tip clearance is a certain gap between the impeller shroud and blades to make sure the relative movement. The flow inside the rotational impeller can move from pressure side to suction side through the gap due to the pressure difference, which contributes to the generation of the tip leakage vortex and hydraulic losses. Different tip clearance sizes in the multiphase pump were

compared to analyze the effect on the pump performance and flow characteristics (Jinsong Zhang et al. 2019). It was indicated that pressure rise reduced when the tip clearance was enlarged. What's more, with the increment in the tip clearance, the results showed the interference of tip leakage vortex with fluid also increased, as well as the vortex area and flow rate of the tip leakage. Kan et al. (2021) developed a CFD multiphase model of a rotodynamic pump to simulate the tip leakage flow characteristics near the tip. It was stated that tip leakage vortex moved from the inlet to outlet of the impeller blade with the tip clearance increase. Meanwhile, the entrainment factor within the tip leakage vortex and flowing mixture in the main channel caused the pattern deterioration as well as hydraulic loss. Besides, tip clearance also enlarges the chance of cavitation phenomenon and greatly affects the helico-axial pump.

Meanwhile, the distribution of the gas bubbles in the gas-liquid flow are considered as the key points to analyze the pump performance. With the increment of the inlet GVF, the flow patterns are generally classified as four typical categories: bubble flow, agglomerated bubble flow, gas pocket flow and segregated flow. Deep researches are required to indicate the influence of bubble sizes and distribution on the pressurization performance of the multiphase pump by the experimental and numerical investigations. Some steady CFD numerical simulations were performed on one single-stage helical-axial pump at various high inlet GVFs to study the multiphase flow as well as the influence of inlet GVF and flow rate on the gas gathering situation (Faustini and Kenyery 2007). In two-phase mixture, the gas phase usually appears as dispersed spherical particle. Zhang et al. (2015) developed

and fixed a buffer tank to get better gas-liquid mixture flow without the aggregations of air bubble before the mixture flowed into the pump. Then the experiments were performed at various inlet GVF and rotational speed conditions. These collected data demonstrated the bubble sizes became larger with the increment of inlet GVF, and descended when the rotating speed increased. Then visualization study of the gas-liquid fluids through the multiphase pump was conducted for further investigation (Jinya Zhang et al. 2016). It was concluded that due to the inlet GVF increment, the average diameter of bubbles grew up and depleted the pump performance rapidly. Similar experiments were made to investigated the flow patterns and bubble diameters inside a multiphase pump with various inlet GVFs with the technology of the high-speed photography (J. Y. Zhang et al. 2015). It was also found the same relationship of bubble diameter and inlet GVF as well as rotational speed. The generation of gas pockets in the channel commonly occurred on the pressure side of the blade near the hub. They mainly gathered in the two-third region of the blade length. The pressurization performance dropped rapidly with the gas pocket flow and relative slowly with the segregated flow. Shi et al. (2018) made both experimental and numerical research on the three-stage pump. It was indicated that increasing the rotating speed to minimize the negative influence due to bubble sizes on the pump performance. It also found such influence mainly resulted from the interphase forces between multiphases. Interphase forces were investigated and compared in the bubbly flow through the rotodynamic pump (Yu, Zhu, and Cao 2015). The result indicted that the drag was superior among these forces in the impeller due to the rotational condition. Similar result was also found that the drag force was dominant and the magnitude ratio of non-drag forces to drag

force was commonly below 1 in the impeller region (W. Zhang et al. 2018).

Some previous studies paid attention to the structure and optimization of the impeller to raise the pressurization ability. Falcimaigne et al. (2002) applied an approach by the combination of an artificial neural network (ANN) system and the genetic algorithm. This technique obtained the optimization of impeller shape on both design and calculation side. Cao et al. (2005) developed a hydrodynamic structure of the impeller by inverse approach and CFD simulations. Final experimental results indicated the developed pump had a maximum efficiency of about 44% and could operate originally even if GVF was up to 50%. A multi-objective optimum model by Zhang et al. (2011) was studied to optimize the impeller conditions by ANN system and non-dominated sorting genetic algorithm-II. Compared to the original result, both differential pressure and static efficiency obtained the enhancement. Zhang et al. (2014) then proposed a three-dimensional blade hydraulic approach for the design of the impeller numerically and experimentally to prove its advantages on large flow rate conditions. In addition, to further improve the pressurization ability and efficiency, an orthogonal design approach was proposed based on experimental data and numerical results (Jinya Zhang et al. 2009). Then nine combination types constituted of four selected designed parameters were compared and analyzed to choose the optimum one. The results presented the improvement of the pressure increment and pump efficiency and the possibility of design optimization by the orthogonal method. Kim et al. (2015a) employed a design optimization on the multiphase pump performance. The key geometric parameters including impeller length, inlet and outlet angle of the blades

were selected to be the modified variables. For the optimized pump, it achieved the increase of 3.5% on static efficiency. Another optimization by numerical simulations was developed for the impeller and diffuser (Suh, Kim, et al. 2017). According to the predicted results on the CFD model, Suh employed a second-order polynomial function to search the optimum point for the hydraulic performance. Parameter optimization for the multi-stage rotodynamic pump is also necessary to enhance the hydraulic performance. A three-stage pump was selected to study the inlet condition at each stage by CFD simulations (Yi Shi et al. 2017). To reduce the influence of the changeable inlet conditions at each stage entrance, some development was applied on the geometric variables of the impeller for the second and third stage, including using smaller inlet angle of the blade, which affected the inlet angle of the flow near the shroud. One or more gaps cut off from the impeller blades was proposed to break large gas bubble form inside the impeller under high inlet GVF conditions (Serena and Bakken 2016). One new split vane impeller pump model based on the ESP with multi-vane was developed to transport mixture flow filled with smaller bubbles to suppress and delay the form of gas concentration area. Consequently, the ESP with split vane has a better operating ability than that of basic multiphase ESPs with inlet GVF, which can reach as high as 70 %. The flow patterns inside the diffuser and gas distribution on each stage inside a three-stage SVIP were also studied by the electrical resistance tomography system (Pirouzpanah, Gudigopuram, and Morrison 2017; Morrison, Pirouspanah, and Kirland 2014).

2.2 Objectives

Current studies are conducted with the designed rotating speed mainly at 1800 rpm (Serena and Bakken 2016), 2000 rpm (W. Zhang, Yu, and Zhu 2017) and 3600 rpm (Kim et al. 2015b; Suh et al. 2018). Since increasing the rotation speed is able to obviously enhance both the handling capacity and operability for wide flow conditions, much more attention should be paid on flow characteristic at higher rotational speed. The main object in the study is to study the hydraulic performance of a single-stage helico-axial rotodynamic multiphase pump in two-phase flow regime using a multi-phase CFD model. The performance is the key issue in industry applications, and it is largely influenced by the impeller structure. The differential pressure and efficiency are two main factors of the simulation results. In addition, the original range of the operational rotating speed of multiphase pump is between 3000rpm and 8000rpm (Falcimaigne et al. 2002). However, previous studies on the helico-axial multiphase focused on the relatively low rotational speed. Hence the specific objectives of the following work are:

1. To create a CFD model to simulate a single-stage helico-axial pump system by the parameters and properties in literature;
2. To study the influence of different bubble sizes on the pump pressure rise predicted by multiphase CFD models;
3. To compare the characteristics of the helico-axial multiphase pump under high inlet GVF as well as high rotating speed (3600-6000 rpm);
4. To modify the impeller blade for the performance improvement of helico-axial pump; including making grooves, slits or holes in the blade;

5. To further investigate the pump performance of both modified and original pump by utilizing different type of liquid-gas fluids pairs through the pump, such as oil-air.

CHAPTER 3

MATHEMATICAL MODELING AND NUMERICAL METHODS

3.1 Mathematical Modeling

ANSYS CFX provides two main multiphase models to conduct the simulation of the multiphase flow: the Eulerian-Eulerian model and the Lagrangian Particle Tracking model. In the study, an inhomogeneous Eulerian-Eulerian model is employed to analyze the characteristics of the multiphase fluids. The Eulerian-Eulerian model is available for a wide range of volume fractions. Within this model, each separate phase possesses one solution filed, so that the velocity and volume faction fields are calculated individually. The same pressure filed is shared by all the phases, and the transported quantities interact via interphase transfer terms like interphase drag. Using the approach, the governing balanced equations of the continuity and momentum can be individually solved for different phases. Assuming there is no mass source or interfacial mass transfer taking place, the continuity balanced equation can be defined as follows.

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

Where different flow phase is denoted by i, and α_i , ρ_i , $\vec{u}_i$ are volume fraction, density and

velocity of phase i, respectively.

Considering volume conservation, there is a constraint that the sum of volume fractions is unity.

$$\sum_{i=1}^n \alpha_i = 1 \quad (2)$$

And the momentum equation can be expressed as

$$\frac{\partial}{\partial t} (\alpha_i \rho_i \vec{u}_i) + \nabla \cdot (\alpha_i \rho_i \vec{u}_i \vec{u}_i) = -\alpha_i \nabla P + \alpha_i \nabla \cdot (\vec{\tau}_i) + \alpha_i \rho_i \vec{g} + \vec{F}_i + \vec{F}_{lift,i} + \vec{F}_{vm,i} \quad (3)$$

Where $\vec{\tau}_i$ is stress-strain tensor of phase i, $\vec{g}$ is gravity acceleration and $\vec{F}, \vec{F}_{lift,i}, \vec{F}_{vm,i}$ represent external body force (like buoyancy and centrifugal force), lift force, and virtual force, respectively.

$$\vec{\tau}_i = \mu_i (\nabla \vec{u}_i + (\nabla \vec{u}_i)^T) + (\lambda_i - \frac{2}{3} \mu_i) \nabla \cdot \vec{u}_i \vec{I} \quad (4)$$

In the study, water and air are used for gas-liquid two phase numerical analysis on the isothermal condition. Therefore, the energy equation is not considered. The water is considered as the continuous fluid while the air is assumed as the disperse fluid. For gas-liquid flow, the rate of each flow can be controlled via Gas Volume Fraction (GVF), which is written as:

$$GVF = \frac{Q_{air}}{Q_{water} + Q_{air}} \times 100(\%) \quad (5)$$

Phase dependent turbulence model is recommended for the Eulerian-Eulerian inhomogeneous model, allowing each phase with a different turbulence model. In ANSYS CFX-Pre, considering the case that a dilute phase is dispersed in a continuous phase, Fluid Dependent turbulence model is selected where the Shear Stress Transport (SST) model is set for the continuous phase while the Dispersed Phase Zero Equation model is set for the

dispersed phase. The transport equations for the SST k - ω model are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (6)$$

and

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (7)$$

Where G , Γ , Y and S represent the generation, effective diffusivity, dissipation and user-defined source terms of k and ω , respectively. And D_ω is the cross-diffusion term.

For the Dispersed Phase Zero Equation model, the equation of the turbulent Prandtl number σ is given as

$$v_{td} = \frac{v_{tc}}{\sigma} \Rightarrow \mu_{td} = \frac{\rho_d \mu_{tc}}{\rho_c \sigma} \quad (8)$$

Where v_t is kinematic eddy viscosity and subscript c or d means the continuous phase or dispersed phase.

The Particle model is supplied to calculate the interfacial area density A between two phases. This model for interfacial transfer is defined as:

$$A = \frac{6\alpha_d}{d_p} \quad (9)$$

Where d_p is particle diameter.

The drag force conducts mainly the interfacial momentum transfer between two phase flows due to the difference of velocity and the Grace Drag Force model is selected to gain the drag coefficient for air-water systems. For a single particle immersed in the continuous phase, the interphase drag for Particle Model can be written as

$$D = \frac{3}{4} \frac{c_D}{d} \alpha_l \rho_g |\vec{u}_g - \vec{u}_l| (\vec{u}_g - \vec{u}_l) \quad (10)$$

Where d represents bubble size, D is the interphase drag force, and C_D is the drag coefficient.

Subscript g or l means the gas or liquid phase.

The drag coefficient relationship is variable according to the bubble shapes. When bubbles rise or fall in the fluid, there are only small bubbles keeping the spherical shape with the terminal velocity, and the larger bubbles will turn into an ellipsoidal and spherical cap regime.

For small particle Reynolds number ranges, where fluids are in the viscous regime, the particles flow with the similar characteristics to solid spherical particles. Therefore, Schiller-Naumann Drag model is applied to define drag coefficient as

$$C_D(\text{sphere}) = \frac{24}{Re} (1 + 0.15Re^{0.687}) \quad (11)$$

For larger particle Reynolds number ranges, which are in the inertial or distorted particle regime, the importance of surface tension develops. The shape of fluid particles is considered as ellipsoid at first and then develops as spherical cap.

The drag coefficient of the sparsely fluid particles in the spherical cap regime is described as:

$$C_D(\text{cap}) = \frac{8}{3} \quad (12)$$

And for densely distributed fluid particles, which is in the dense spherical cap regime, the drag coefficient should be expressed with a correction:

$$C_D(\text{cap}) = \frac{8}{3} (1 - \alpha_d)^2 \quad (13)$$

Several correlations are available for the distorted particle regime. Grace (Acrivos 1979) correlation is selected in this study. For the Grace model which was developed based on the air-water data, the drag coefficient on the bubbly flow yields

$$C_D(\text{ellipse}) = \frac{4}{3} \frac{gd}{U_T^2} \frac{\Delta\rho}{\rho_c} \quad (14)$$

$$U_T = \frac{\mu_c}{\rho_c d_p} M^{-0.149} (J - 0.857) \quad (15)$$

$$M = \frac{\mu_c^4 g \Delta\rho}{\rho^2 \sigma^3} \quad (16)$$

$$J = \begin{cases} 0.94H^{0.757} & 2 < H \leq 59.3 \\ 3.42H^{0.441} & H > 59.3 \end{cases} \quad (17)$$

$$H = \frac{4}{3} Eo M^{-0.149} \left(\frac{\mu_c}{\mu_{ref}} \right)^{-0.14} \quad (18)$$

Where U_T is the terminal velocity, M is Morton number and $\mu_{ref} = 0.0009 \text{ kg m}^{-1} \text{ s}^{-1}$ for water at 1 bar and 25°C.

$$C_D(\text{dist}) = \min(C_D(\text{ellipse}), C_D(\text{cap})) \quad (19)$$

$$C_D = \max(C_D(\text{sphere}), C_D(\text{dist})) \quad (20)$$

Since Grace drag model considers the fluid flowing a single bubble, another correction is applied for multiphase flow with dense bubbles by a power law:

$$C_D = \alpha_c^p C_{D\infty} \quad (21)$$

Where $C_{D\infty}$ means Grace Drag Coefficient for single bubble, and c is continuous phase.

Equation 14 is based on the force balance where a single bubble rises in a liquid. In this case, the change rate of the bubble momentum should be equal to the difference of the net buoyancy force and drag force. The unsteady state terms are neglected for simplification because the state of bubble in the fluid becomes steady very fast. Therefore, the balance equation can be simplified as:

$$F_d = \frac{4}{3} \pi r_e^3 (\rho_l - \rho_g) g \quad (22)$$

$$C_D = \frac{F_d}{\frac{1}{2}\rho_l U_T^2 (\pi r_e^2)} = \frac{4}{3} \frac{gd}{U_T^2} \frac{\Delta\rho}{\rho_l} \quad (23)$$

Where r_e means equivalent radius, which is the radius of the sphere with the same volume as bubble.

In the above drag coefficient models, terminal velocity means as a bubble rises in the fluid, it will finally reach the steady state at a maximum velocity with the drag force equal to the driving forces. Since fluid is quiescent, the terminal velocity of bubble is equal to the relative velocity between the bubble and surrounding fluid.

However, since the commonly used drag coefficient models including Grace drag model are almost obtained based on bubbles rising in a static fluid, further researches on the bubble moving through the turbulent flow inside the rotating machinery are necessary. To study the gas–liquid two-phase flow inside the rotodynamic pump, a correction for the drag coefficient model in quiescent flow was developed for turbulent flow (Y. Li, Yu, and Sun 2021). The drag coefficient in two fluids was correlated through the Stokes number. By comparing the pump performance and the flow pattern between original and modified drag models at three rotating speeds (2350, 2650 and 2950 rpm), it showed that the simulation results with the correction were closer to the experiment data at relative low speed (2350 rpm) and the discrepancy of the drag coefficient was largest. But when rotating speed increased to 2650 rpm or 2950 rpm, the effect of the eddies on the bubble motion weakened since the frequency was very high and the time acting on the bubbles reduced. As a result, the difference of C_D before and after the correction became smaller and it was stated the ΔC_D would continue to reduce with the further increment of the rotating speed.

In addition, non-drag forces exist in the multiphase flow, such as lift force and virtual mass force. Lift force works on the dispersed phase in the rotating continuous gradients, which remains perpendicular to the relative motion direction between the phases. Virtual mass force is caused by the relative phasic acceleration between gas and liquid. The negligible effect by virtual mass force on the steady-state simulations was pointed out due to little acceleration in a steady state (Deen, Solberg, and Hjertager 2001). Numerical analysis was also conducted with different coefficient value which confirmed little effect on the performance at different inlet GVF's (Suh, Choi, et al. 2017). One characteristics analysis on the phase interaction between gas and liquid flowing through the multiphase helico-axial pump was carried out and the magnitude ratios of different non-drag force to drag force were compared (W. Zhang et al. 2018). The result showed that on the 21% inlet GVF condition, the magnitude ratios of virtual mass force to drag force and lift force to drag force were typically below 1 with the value of turbulent dispersion force to drag force generally below 0.2 along the impeller and diffuser passages. Due to the centrifugal force in the high speed rotodynamic pump, the liquid phase moves towards the shroud while the gas phase gathers near the hub. The gas-liquid separation occurs when mixture flows into the rotating impeller, especially being obvious in the downstream region of impeller. Such inhomogeneous distribution in the impeller passage causes the relative velocity between two phases being much larger. According to Equation (10), it is clearly that the drag force become much larger and thus dominant among the interphase forces. Considering the work situation that the fluids flow very fast with large centrifugal force in this study, it is proposed that the influence of the non-drag forces is limited during the analytical modeling

on the pump performance.

Besides, the effect of particle induced turbulence cannot be neglected. In the multiphase flow, large particles lead to turbulence enhancement in the continuous phase because of the appearance of wakes behind the particles. CFX provides the Sato Enhanced Eddy Viscosity model to solve such effect via a corrected continuous phase eddy viscosity:

$$\mu_{tc} = \mu_{ts} + \mu_{tp} \quad (24)$$

$$\mu_{tp} = C_{\mu p} \rho_c r_d d_p |U_d - U_c| \quad (25)$$

Where μ_{tc} refers to Shear Induced Eddy Viscosity, μ_{tp} refers to Particle Induced Eddy Viscosity, and $C_{\mu p}$ is a constant variable at 0.6.

3.2 Geometry Modeling

ANSYS Bladegen 19 is used for the impeller and diffuser model generation. This model provides a full passage domain for the simulation to analyze the multiphase pump. As to the single-stage rotating pump, the computational domain includes an impeller and a diffuser. In addition, the extended pipes are supplied as the inlet and outlet pipes to ensure the inflow and outflow of the pump under the steady conditions. The model is designed mainly based on the following parts: the meridional plane and blade angles. The meridional plane in Fig. 1 illustrates the main design geometry parameters, such as the inlet area length, the outlet area length, and the pump blade length. Such parameters contribute to the basic

size design of the model, which were optimized in the previous studies (Kim et al. 2015b; Joonhyung, Youngseok, and Joonyong 2014).

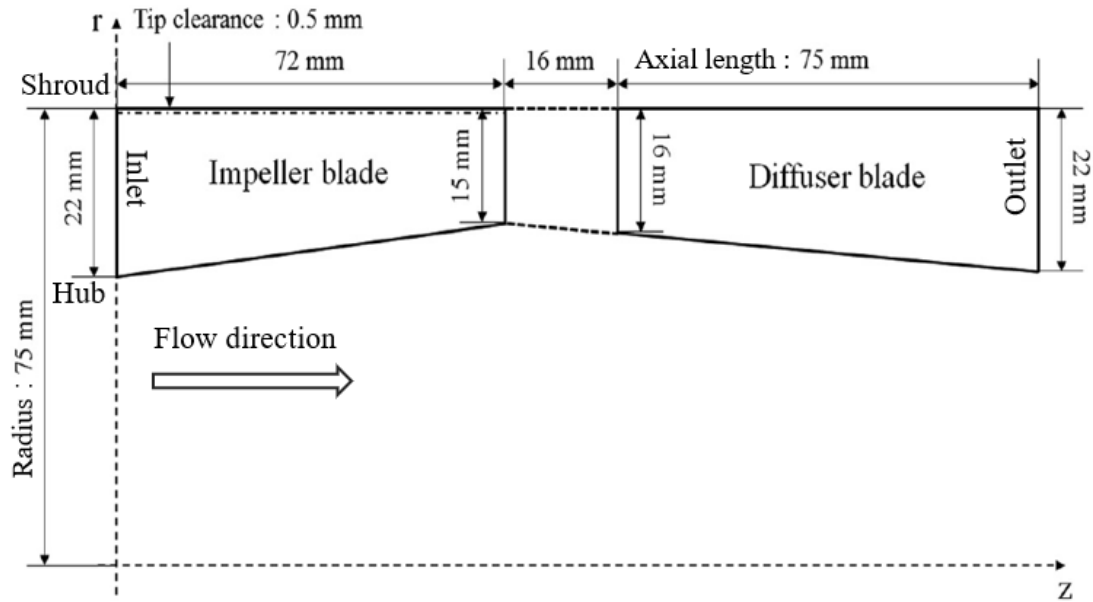


Figure 1 Meridional plane of the basic model

Besides the parameters displayed in the meridional plane, the pump blade angles also play vital roles in the model design, including the hub inlet angle (β_{1h}), the shroud inlet angle (β_{1s}), the shroud outlet angle (β_{2s}), and the hub outlet angle (β_{2h}). For the impeller, the shroud shares the same sweep angle with hub to avoid the blade torsion phenomenon. And in order to ensure the outlet fluids flowing out of the pump along the axial direction, all outlet angles at the hub and shroud of the diffuser are set at 90 deg, which means the outlet blade is parallel to the axis. Fig. 2 shows the linear curve of the

blade angles of the impeller and diffuser. And for the structural stability, the thickness of the impeller blade and the diffuser blade are fixed at the constant, which are 4mm and 3mm, respectively. All these selected variables are shown in Table 1.

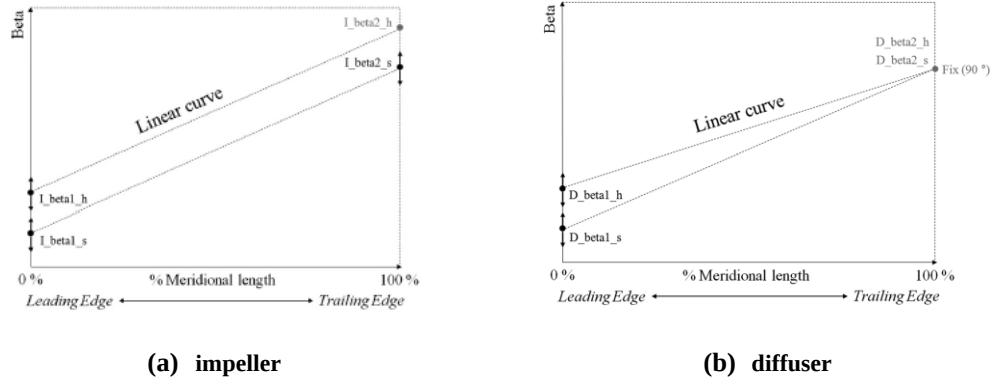


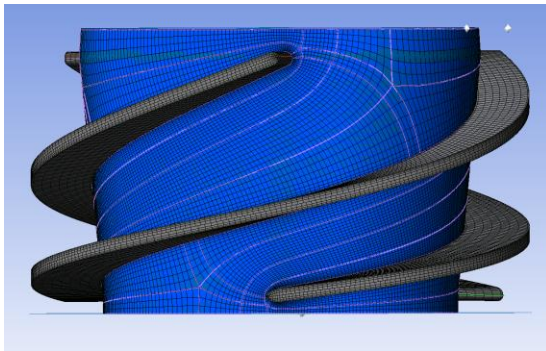
Figure 2 Blade angles of the impeller and diffuser

Table 1 Designed selected variables

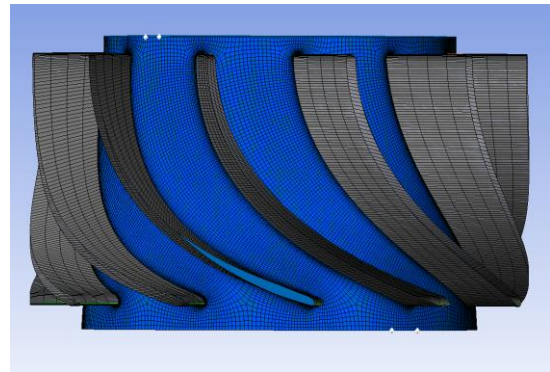
Variables	Impeller	Diffuser
beta(1_h)	10 deg	17 deg
beta(1_s)	6 deg	15 deg
beta(2_h)	25.5 deg	90 deg
beta(2_s)	21 deg	90 deg
blade number	3	11
thickness	4mm	3mm

In this study, the structured hexahedron mesh is selected in the numerical analysis. ANSYS TurboGrid 19 is employed to obtain the mesh generation in the impeller and diffuser region.

The physical model creation and mesh generation for inlet and outlet pipes are both conducted in ANSYS ICEM 19. The pipe is extended to obtain fully developed flow. To obtain the mesh with high quality, the O-block technique is employed in the procedure of the passage topology structure. The method allows for the near wall grids locally refined, which contributes to the analysis of the characteristics of the flow inside the boundary layers. Fig. 3 and Fig. 4 display the grid on the blades and the hub of the single-stage pump components as well as the whole computational domain of the multiphase CFD model. The studied fluids flow into the inlet pipe in the opposite direction to the gravity and finally flow out.



(a) impeller



(b) diffuser

Figure 3 Grid of the impeller and diffuser

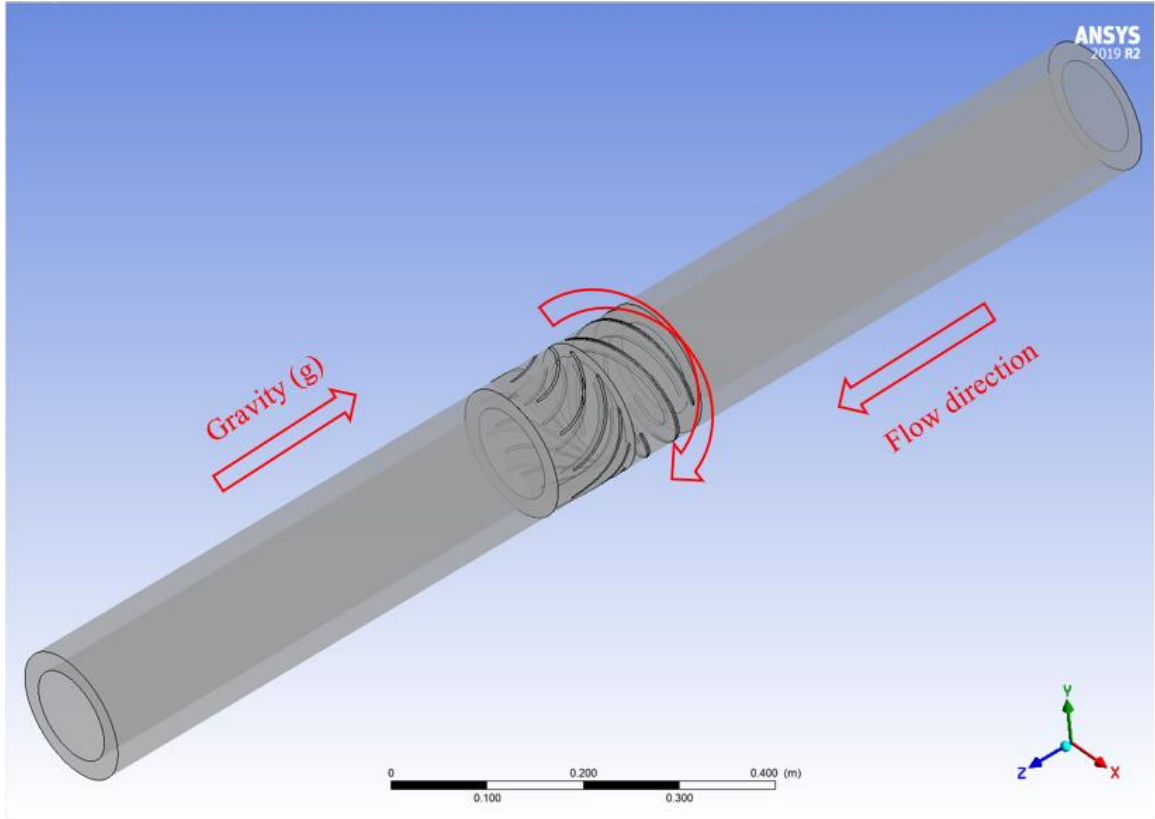


Figure 4 Whole computational domain of the multiphase CFD model

3.3 Numerical Methods

The numerical simulations are conducted with ANSYS CFX 2019 R2, which is a commercial CFD analysis program widely used for fluid machinery modeling. Boundary conditions are important in the computational domain, which affect the prediction results greatly. No-slip wall condition is selected on all the wall surfaces. A rotational speed is given to the impeller domain. Considering the tip clearance between the shroud and blades, the counter rotational wall condition is additionally set on the wall of the pump shroud, which actually models the stationary inner wall. For the whole computational domain, the

total pressure condition is given at the inlet region and the bulk mass flow rate is set at the outlet region. The total pressure refers to the pressure conversion by the dynamic energy without any losses if the flowing fluid is suddenly brought to rest, which equals the sum of the static pressure and dynamic pressure. The frozen rotor technique is used on the interfaces between the rotational region and adjacent stationary region for the steady simulation. The frozen rotor model in CFX is the steady- state method which is similar to the rotating reference frame model in Fluent. When the fluid flows through two adjacent components, usually the impeller and diffuser, this model solves the interaction across the interfaces by changing the reference frame with the relative position maintained constant. The mesh does not move and the rotation effect is automatically included by employing the Coriolis and centrifugal force terms in the NS equations. This algorithm requires less computer resources compared with other reference frame models. Since the grids of the impeller and diffuser on the interface side are unable to match, General Grid Interface technique is employed to connect these grids. The basic setting for the CFD modeling is shown in the Table 2. Besides, the following situations are assumed during the numerical simulation:

- 1) The multiphase flow is homogeneous at the inlet of the computational domain;
- 2) No mass transfer occurs between different phases;
- 3) No cavitation phenomenon appears inside the multiphase pump.
- 4) The main phase is continuous fluid and air ideal gas is dispersed.

Table 2 Solution Methods and Boundary Conditions

Component	Details
Calculation method	Steady-state
Medium	Air ideal gas and water
Turbulence models	Air ideal gas: Dispersed-phase zero equation Water: k- ω SST
Turbulence transfer	Sato Enhanced Eddy viscosity
Boundary conditions	Inlet: Total pressure and inlet GVF Outlet: Mass flow rate (kg/s) Domain interface: Frozen rotor Wall: No slip condition and isothermal (25C)
Drag force	Grace model

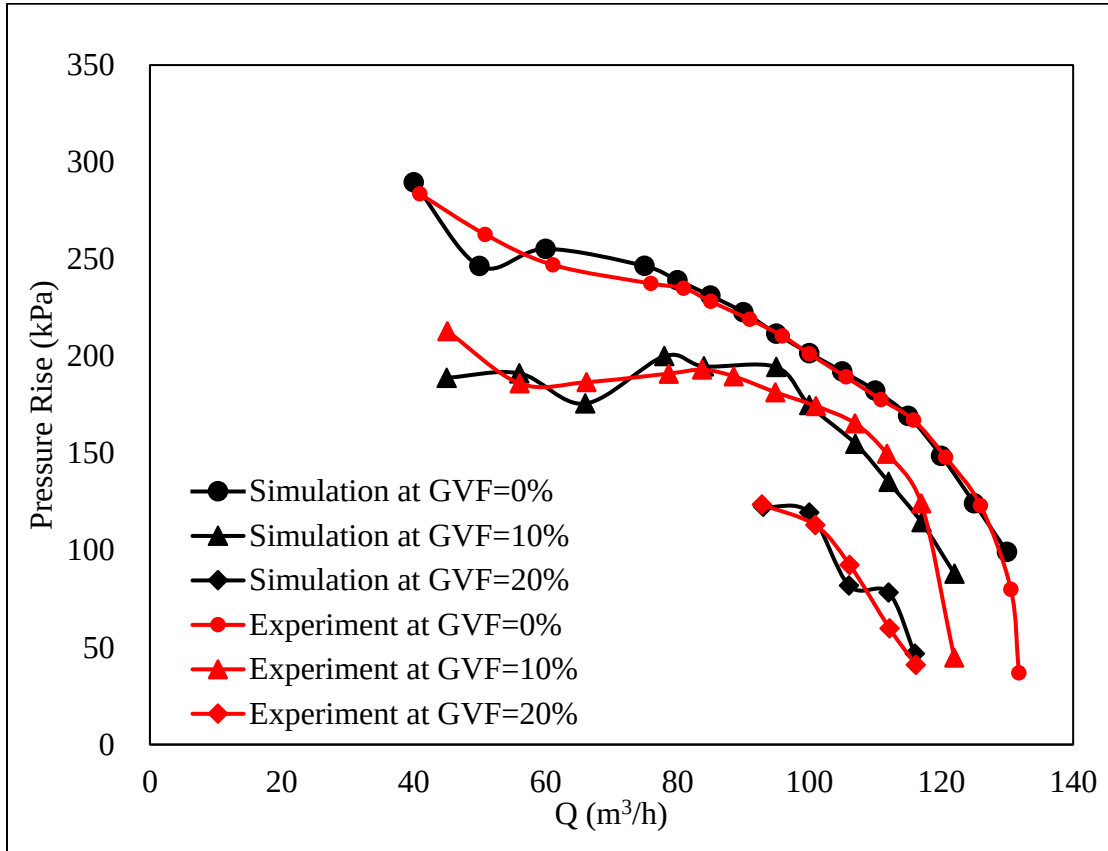
CHAPTER 4

RESULTS AND DISCUSSION

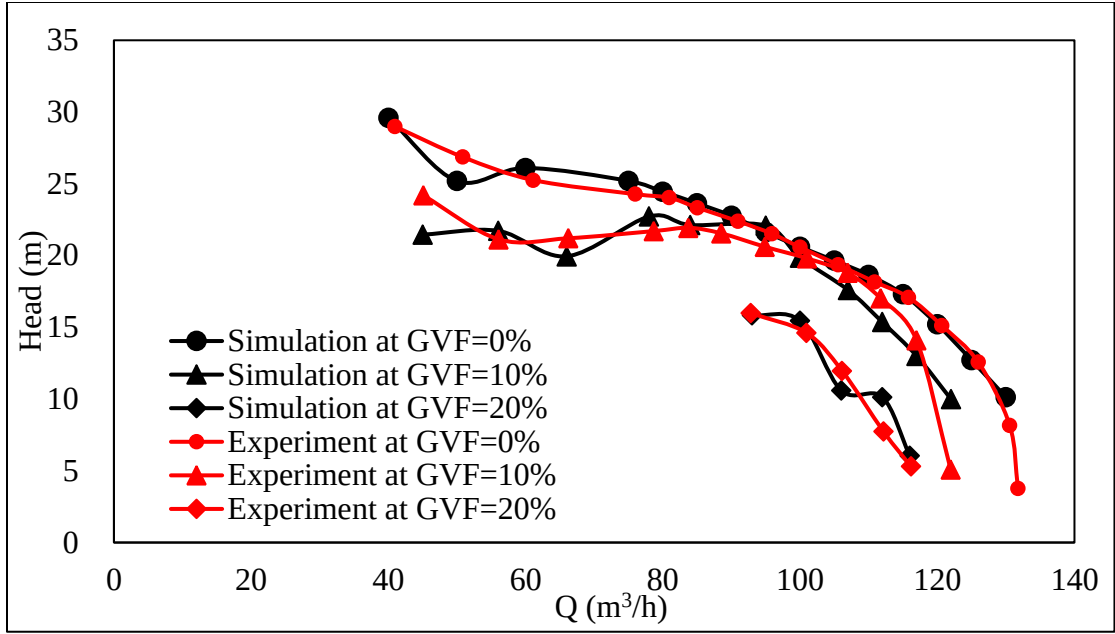
4.1 Validation Results

In order to confirm the prediction accuracy of the numerical simulation of the studied model, experimental data from the previous study is supplied to compare the pressure increment and efficiency at the different flow rates. Fig. 5 displays the comparison results of the pressure rise and efficiency with the related experiment data (Kim et al. 2015a). For single phase flow, i.e., $GVF=0$, experiment and simulation results of pressure rise reach a good agreement, where the pressure rise declines as the flow rate increases. And the efficiency results match well at the designed point of $100\text{m}^3/\text{h}$ and higher flow rate. But the error ranges of efficiency rise at the low flow rate region. In this region, the statics efficiency increases when the flow rate augments. However, the efficiency reduces rapidly at the high flow rate region. The best efficiency points exist with the corresponding flow rates. For the multiphase flow simulations, the trend of the pump performance curves is similar with that of the single flow. With the flow rate enhancement, the pressure rise keeps decrement and the efficiency rises at first and the turns to fall. And it is clear that both the differential pressure and efficiency decrease as inlet GVF rises from 0 to 20% at the same

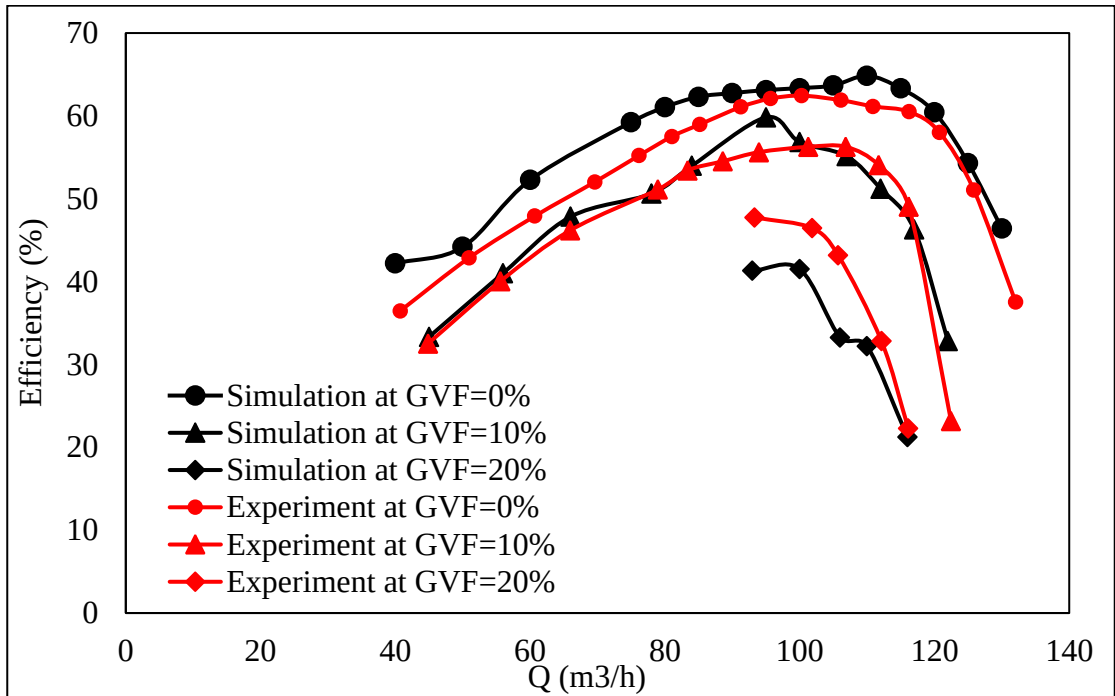
flow rate, which illustrates that inlet GVFs have a great effect on the overall performance. It can be concluded that the pressure rise and statics efficiency will continue to decline when inlet GVF increases.



(a) Pressure rise



(b) Pump head



(c) Efficiency

Figure 5 Pump performance Comparison of experimental data (Kim et al. 2015b) and CFD multiphase model numerical results (flow rate: $100\text{m}^3/\text{h}$, rotational speed: 3600rpm)

4.2 Original Pump

4.2.1 The Effect of the Bubble Diameter

For the multiphase pump performance, it is largely affected by the diameters of gas bubbles in the flow of gas-liquid two-phase. The exact value of bubble diameter is variable and difficult to determine because of the complicated flow pattern inside the pump. Several previous works have been conducted to evaluate the gas bubble diameters by visual experiments. The visualization of dispersed bubbly flows was conducted to evaluate the bubble diameters (Falcimaigne et al. 2002). A high-speed photography camera method was applied in the experiment for the evaluation of bubble size distribution by successive photographs. It showed the bubble diameters ranged mainly from about 0.4 mm to 0.8 mm when experiments were made at the best efficiency point under the 10% inlet GVF condition. One high-speed photography system was employed to study the characteristics of mixture flow in the three-stage multiphase pump (Jinya Zhang et al. 2016). The bubbles were assumed axisymmetric and the mean diameters were calculated according to a series of pictures under various inlet GVF and rotating speed conditions. The conclusion stated the mean gas bubble diameter augmented with inlet GVF increment but reduced with the rotating speed up.

Based on the conclusions from the previous works, it is important to analyze the effect of the bubble sizes on the prediction of pump performance. In this work, an analytical investigation is conducted by estimating the multiphase pump performance with various bubble diameters and inlet GVFs. The selected flow rate is $100 \text{ m}^3/\text{h}$, the rotating speed is 3600 rpm and the inlet GVFs are set as 5, 10, and 15%. The diameter of the gas bubble ranges from 0.1mm to 1.5mm. The prediction of pressure rise with different bubble diameters is shown in the Fig. 6. The pressure rise decreases gradually when the mean bubble size increases, which shows bubble sizes is relative to the pressurization performance inside the helicon-axial pump. On the flow condition where inlet GVF is 10%, the differential pressure matches the experiment data well when the average diameter is set as 0.7 mm. When inlet GVF increases to 15% with other conditions still the same, the pressure rise decreases and the corresponding bubble diameter expands to nearly 1.0 mm. When it declines to 5%, the related bubble diameter is around 0.4 mm and the curve of pressure rise versus bubble diameter becomes relatively stable. All the bubble diameters mentioned under above flow and pump conditions are also consistent with the data recommended for the selection of the bubble size according to GVF in the previous study (Suh, Choi, et al. 2017).

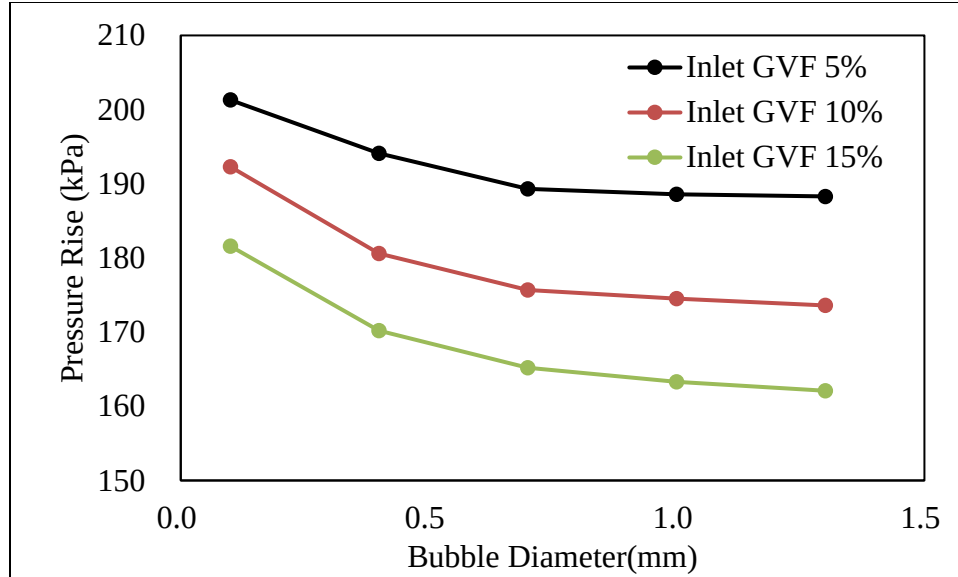


Figure 6 The effect of the bubble size on pump pressure rise by CFD multiphase model (flow rate: $100\text{m}^3/\text{h}$ and rotational speed: 3600rpm)

4.2.2 Pump Simulations on Various Conditions

The results of the pressure rise at various inlet GVFs and rotating speeds at the same flow rate ($100\text{m}^3/\text{h}$) are displayed in Fig. 7. There are three rotating speeds (3600 rpm , 4800 rpm and 6000 rpm) applied for the rotational impeller in the CFD simulations. The flow pattern in the single-stage pump with inlet GVF range from 10% to 90% are all studied by CFD simulations. The bubble diameters are assumed with the linear growth relationship at low inlet GVFs, and constant at high GVFs, which are shown in the Table.1. It is illustrated that under the same inlet GVF and volume flow rate condition, higher differential pressure can be obtained by increasing the rotating speed. For the same rotational speed and flow rate, when inlet GVF increases from 10% to 90% , the pressure rise declines rapidly and then becomes gradually stable. In order to make a qualitative comparison, a set of

experiment results of pressure rise measurements are also utilized, which are measured on a three-stage helicon-axial pump working at 4500 rpm rotational speed and with 100 m³/h volume flow rate by Shi et al. (2018). It also shows that on the same operating conditions, the pressure rise decreases as the inlet GVF grows, especially when inlet GVF exceeds 15%, the pressure rise decline becomes more rapid. The comparison between the pressure rise obtained from the present numerical simulation and that obtained experimentally shows reasonable qualitative agreement at low values of inlet GVF.

According to the results from section 4.2.1 in this study as well as the previous works, it is concluded that when inlet GVF increases, the exact value of the bubble diameter should increase in response. It is important to mention that the bubble size diameters in the GVF range from 60% to 90% are considered exactly the same as their values at GVF 50% as indicated in Table 3. This may have affected the accuracy of our results in that range.

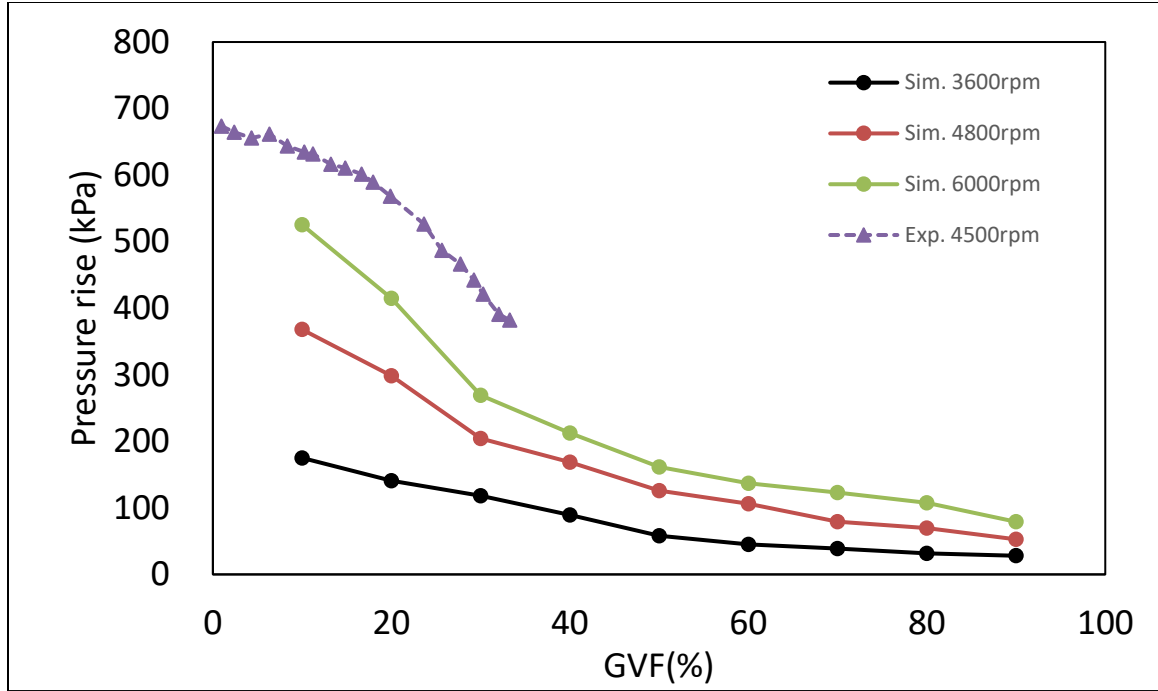


Figure 7 Predicted performance by CFD multiphase model versus different inlet GVFs using different assumed bubble sizes at different rotational speed (flow rate: 100m³/h) with experimental data (flow rate: 60m³/h) by Shi et al. (2018)

Table 3 Bubble sizes used for CFD simulation in different inlet GVFs and rotational speeds

Inlet GVF (%)		10	20	30	40	50	60	70	80	90
Average	3600rpm	0.7	1.3	1.9	2.5	2.8	2.8	2.8	2.8	2.8
Bubble	4800rpm	0.5	1.1	1.7	2.3	2.5	2.5	2.5	2.5	2.5
Diameter	6000rpm	0.3	0.9	1.5	2.1	2.2	2.2	2.2	2.2	2.2
(mm)										

4.2.3 Tow-Phase Flow Behaviors

To start a better investigation the flow characteristics of mixture inside the multiphase pump, CFD simulation with 50% inlet GVF is selected as an example to study the multiphase flow distribution. The rotating speed is 3600 rpm, and the volume flow rate is 100 m³/h. Fig. 9 displays the average GVF distribution in the whole single pump on the meridional plane. Higher GVF area occurs at the hub, and lower GVF appears near the shroud. This flow pattern happens because the fluids flowing through the high-speed rotational impellers is mainly subjected to the centrifugal force. With a large density difference of air and water, the force makes water moving toward the shroud and air gathering near the hub. As a result, high GVF area appears at the hub near the outlet position, where the maximum value of GVF can even reach as high as 100 %. When the fluids flow into the diffuser, it is the stationary part which can eliminate the centrifugal force and adjust the fluid field. As a result, high GVF flow near the hub and low GVF flow at the shroud remix and disappear gradually in the diffuser. The following parts show the gas gathering situation at different blade-to-blade span views of the impeller in Fig. 10. The 10 % span-wise means closing to the hub and the 90 % span-wise is near the shroud. It is illustrated that majority space of flow passage is full of gas on the 10 % span-wise plane. For the 90 % span-wise plane, it shows that gas gets gather on suction side of the blade around the center region of the passage, while water mainly gathers on the pressure side at the outlet region. As to high inlet GVFs, high GVF region occurs and bubbles occupy the most area of flow passage, which finally causes larger hydraulic losses. As a result, the pump performance deteriorates significantly. When the rotating speed is 6000rpm, the separation of mixed phase becomes more obvious due to the centrifugal force

being larger. And more gas concentrates and occupies the space of the flow passage while water gathers near the shroud, which causes more hydraulic losses. Thus, reducing or diminishing the gas concentration area inside the impeller should be considered when improving the multiphase pump performance.

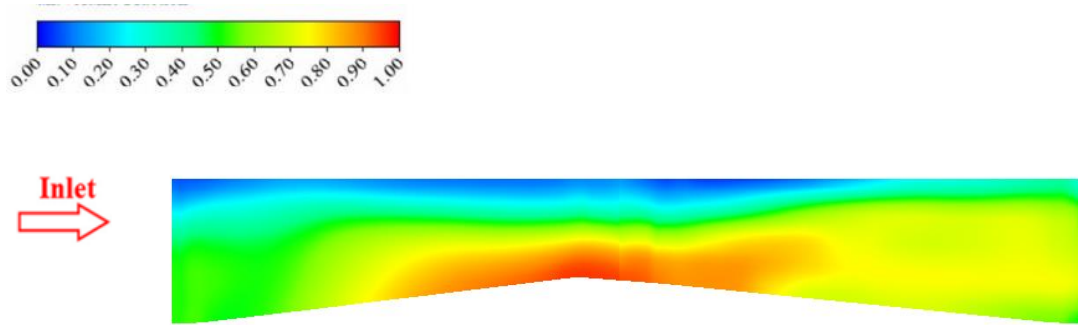
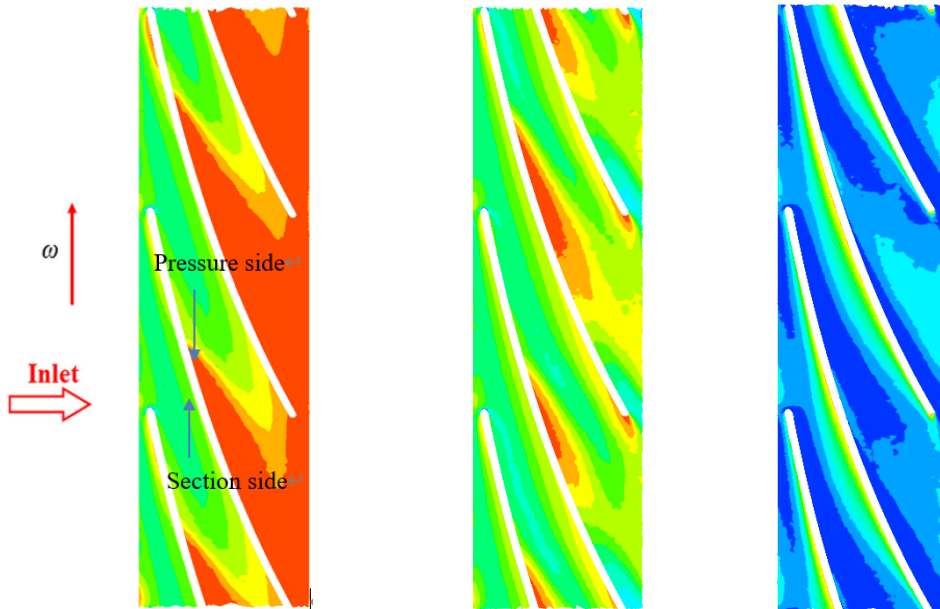


Figure 8 Gas distribution of the impeller and diffuser on the meridional plane (flow rate: $100\text{m}^3/\text{h}$, rotational speed: 3600rpm and inlet GVF is 50 %)



(a) 10 % span-wise (near hub)

(b) 50 % span-wise

(c) 90 % span-wise (near shroud)

Figure 9 Gas distribution of the impeller on different span-wise planes (flow rate: $100\text{m}^3/\text{h}$, rotational speed: 3600rpm and inlet GVF is 50 %)

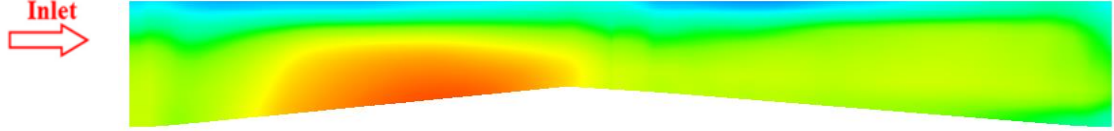
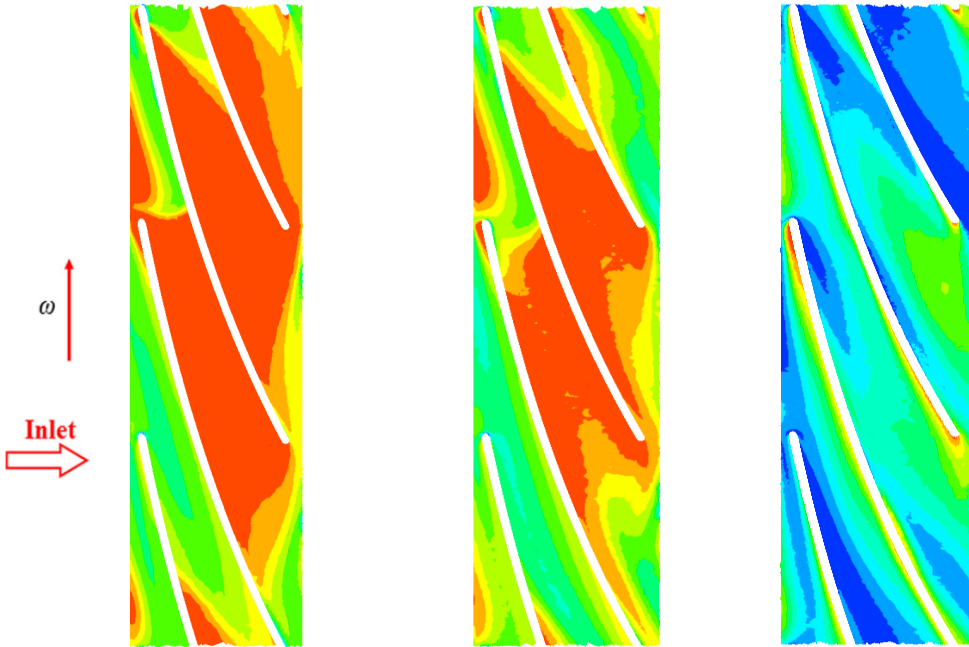


Figure 10 Gas distribution of the impeller and diffuser on the meridional plane (flow rate: $100\text{m}^3/\text{h}$, rotational speed: 3600rpm and inlet GVF is 50 %)



(a) 10 % span-wise (near hub)

(b) 50 % span-wise

(c) 90 % span-wise (near shroud)

Figure 11 Gas distribution of the impeller on different span-wise planes (flow rate: $100\text{m}^3/\text{h}$, rotational speed: 6000rpm and inlet GVF is 50 %)

4.3 Modified Pump

4.3.1 Modified Impeller Models

Since high gas gathering area has a large influence on the pump performance, one effective way to boost the pressure increment at high inlet GVF is to suppress the form of the gas concentration. Then modified impeller with split blades are proposed to remix the separated liquid and gas in the flow passage. In this study, the design of new impeller is based on the original impeller, which is achieved by creating a small flow passage on the blade. Therefore, the position, shape as well as axial width of the slits should be taken into consideration to obtain the better improvement on the pump performance. According to the above investigation of the flow pattern on the original impeller in this work, gas concentration area usually take place around the second half part of the blades on the suction side. Thus, a small part is cut off on the middle position from the impeller blade along the axial direction. Then the liquid flow on the pressure side can move to the suction side from the passage to remix with the gas and decrease the high GVF area. Fig. 13(a) is the 3D geometry of the impeller with split vane, where H is the axial length of the impeller and δ is the axial width of the split. According to the previous works, the value of the split was recommended within the range of 0-7.5 % of the impeller axial length (Yi Shi et al. 2019). In this study, the axial width δ is regulated to 2 mm to estimate the split vane pump performance under high inlet GVF conditions. The mesh of the split vane impeller is created by ANSYS Workbench Mesh, which is shown in Fig. 13(b).

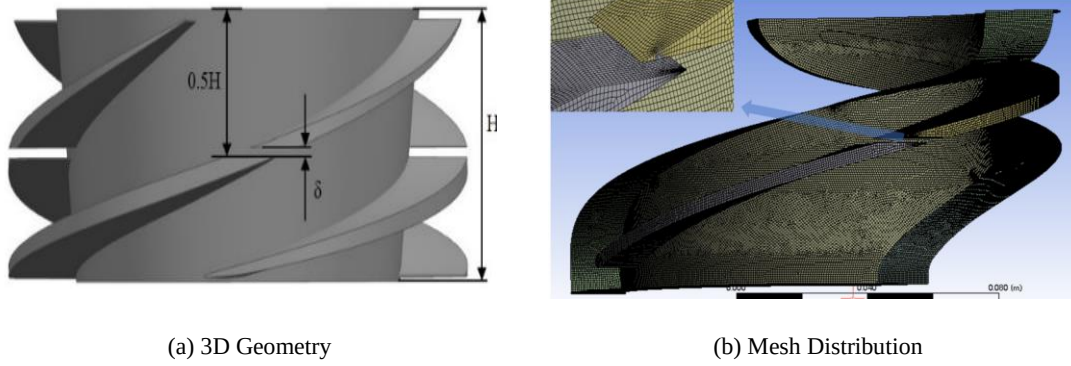


Figure 12 Split vane impeller model and mesh ($\delta = 2\text{mm}$)

Additionally, to compare the influence of the split vane impeller, another type of modified blade with holes is designed by creating 5 holes along the blade from the inlet to the outlet. Each hole shares the same cross section area, which equals to one-fifth of the area of the split. Two different shapes of hole are selected: rectangle hole and square hole. For the rectangle type, one side length is set as 2mm while another is 3.7mm. For the square type, the side length is designed as 2.7mm. Fig. 14 shows the mesh of the impeller blade with 5 holes.

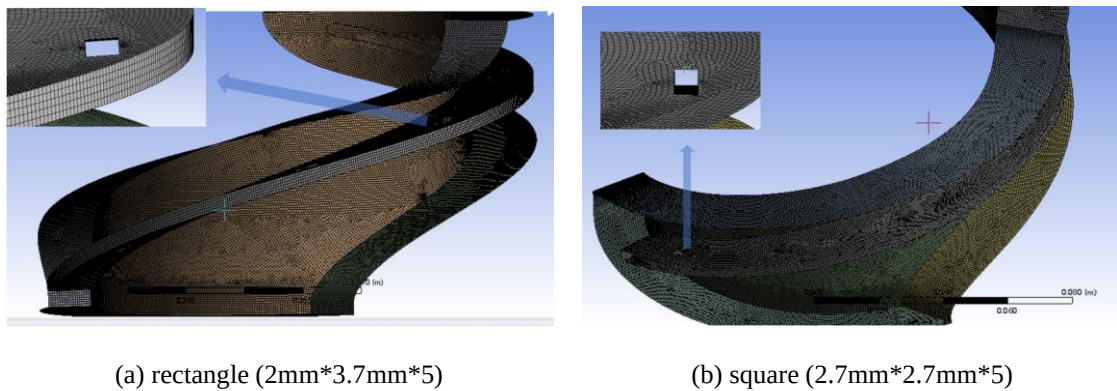


Figure 13 Modified impeller model with 5 holes on each blade

4.3.2 Performance Comparison between Original and Modified Impellers

As is shown in Fig. 15, the performance of rotodynamic pump with the modified impellers is plotted to compare and analyze the difference with the results of the original impeller with inlet GVF range from 10 % to 90 %. The volume flow rate is regulated to 100 m³/h and the rotating speed is set as 3600 rpm. It states there is little increment of the differential pressure for modified impellers at 10% and 90% GVF compared with the original one. The improvement of pressure rise in the modified impeller is not obvious when handling low inlet GVFs multiphase flow. When inlet GVF reaches 90 %, the mixture flow is similar to pure gas and air concentrating area occurs in the most of the flow passage. The effect of small flow channel and holes. The modified pumps have little advantage over the original pump when operating under the low and extremely high inlet GVF conditions. Besides such extreme conditions, the performance on the pressure increment of modified impellers is better than that of the original impeller. Best improvement effect appears around 50% - 60% GVF. Then as the inlet GVF increases, the difference of boosting capacity between the modified impeller and original one declines gradually. It is also shown that the pump with holes perform a little better than the pump with split vane when comparing the pressure rise under the same operating conditions. In addition, the curve of rectangle holes are almost coincided with the square one, which indicates the shape of holes has little influence on the pump performance development.

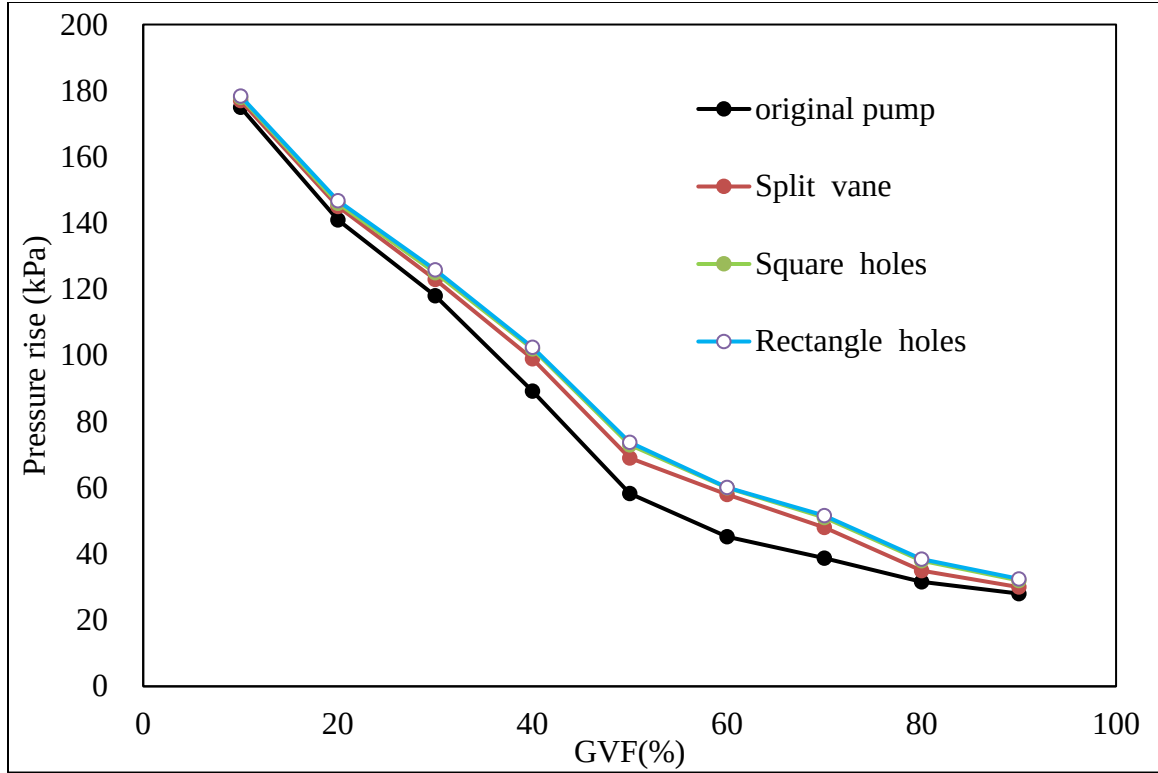


Figure 14 Predicted pressure rise comparison between modified pumps and original pump by CFD multiphase model versus different inlet GVFs (flow rate: 100m³/h, rotational speed: 3600rpm)

4.3.3 Tow-Phase Flow Behaviors

Flow pattern inside the impeller with split vane are also investigated within different inlet GVFs. The gas distribution on different planes at 10% inlet GVF is shown in Fig. 16. It is obvious that there is little high GVF area occurring in the flow passage and only a little difference of the gas distribution at three span-wises. Gas distribution of high inlet GVF in the flow passage is also investigated in Fig. 17, in which average GVF is presented on different span-wise planes when inlet GVF is 50 %. It can be seen that the GVF concentration inside the impeller with split vane is a bit better than that in the original

impeller. On the 10 % span-wise plane, small gas bubbles gather on suction side of the blade near the outlet. Gas concentration area becomes less from hub to shroud and almost disappears on 90 % span-wise plane. For the flow channel on the center of the blade, the value of GVF is nearly zero, which illustrates liquid flow on the pressure side can move to the suction side through the split. The maximum GVF area also appears on the hub near the impeller outlet region. But the value and the area are much smaller than that of the original impeller. Such situation occurs due to liquid flowing through the added channel to the gas concentration on the suction side. In this way, liquid phase and gas phase remix again inside the impeller. As a result, the high gas concentration area largely reduces and the pressurization performance of the split vane pump can be enhanced. And for the pump with rectangle holes on the same conditions, the desperation of gas and liquid is a little more obvious at 10% and 90% span-wises due to the lack of the flow channel. At the 50% span-wise, no much difference appears around the first two holes. And then the gas concentration area is greatly reduced due to the rest of holes. The pressure rise of the pump with rectangle holes is 76.4kPa, which is nearly 10% greater than the former pump. When the rotational speed increases to 6000rpm, more gas concentration occurs. The holes and slits in the impeller blades play a great role in breaking large bubbles. For the pump with split vane, the slit can still eliminate the gas concentration and remix two separated phases, but the separation of gas phase with liquid occurs again and gas continues to gather at the second half part of the flow passage gradually, especially near the hub, which reaches almost 100% GVF. As to the pump with rectangle holes, due to the larger centrifugal force difference of gas and liquid phase, more high gas volume area appears on the center

position of the passage in the radial direction. The function of the holes for remixing the air and water is limited because less water at pressure side around the holes can move through the small channel. As a result, the development of the pump with rectangle holes is less than that of the pump of split vane.

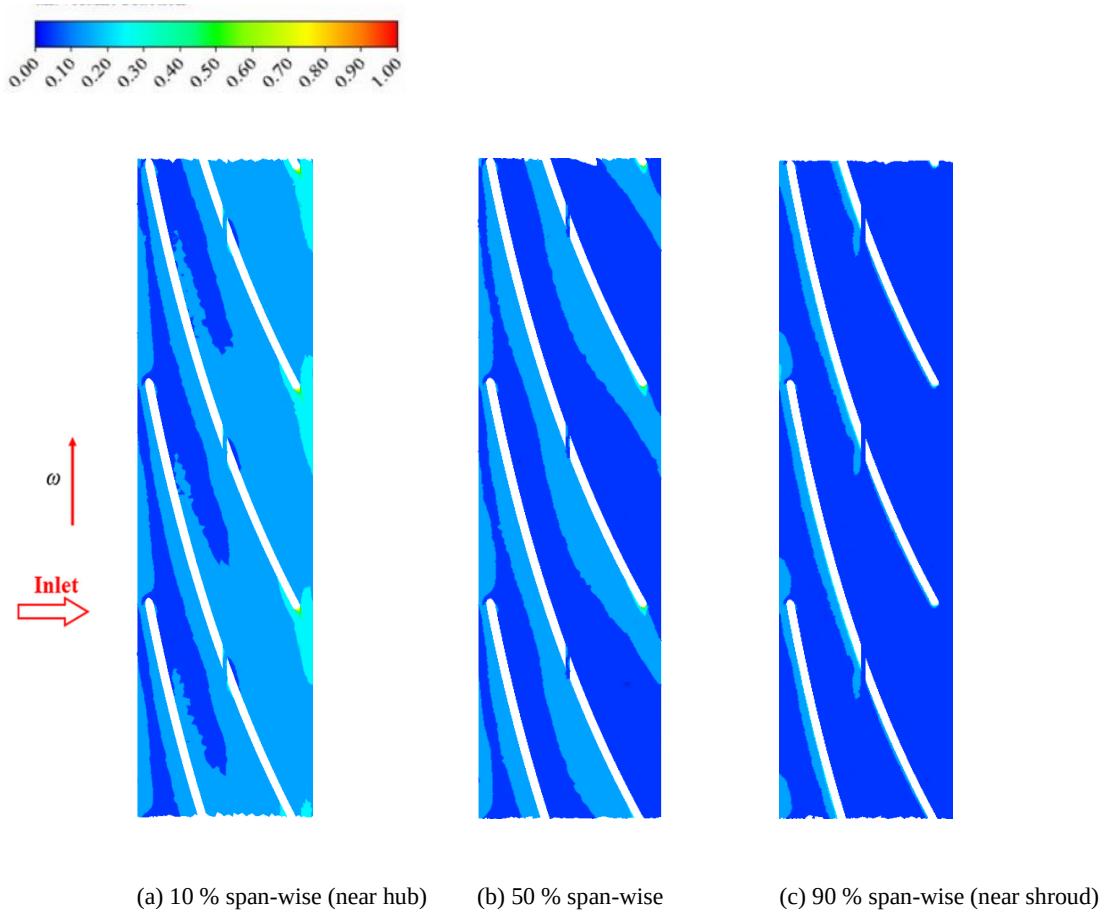


Figure 15 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m³/h, rotational speed: 3600rpm and inlet GVF is 10 %)

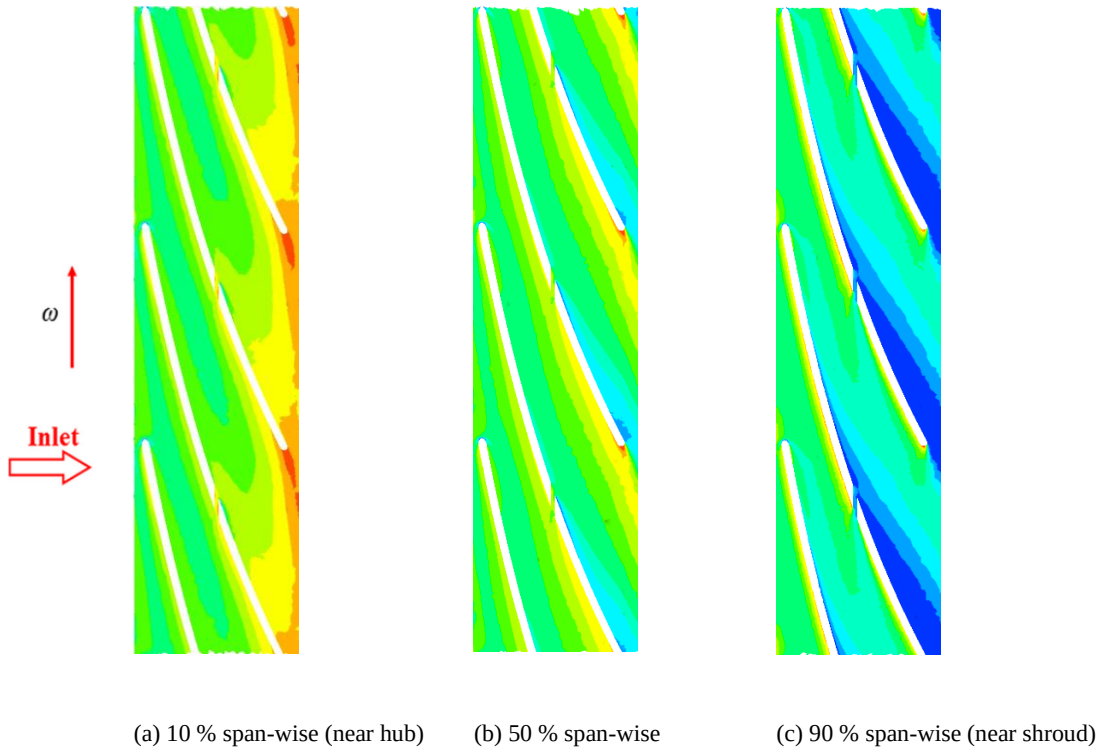
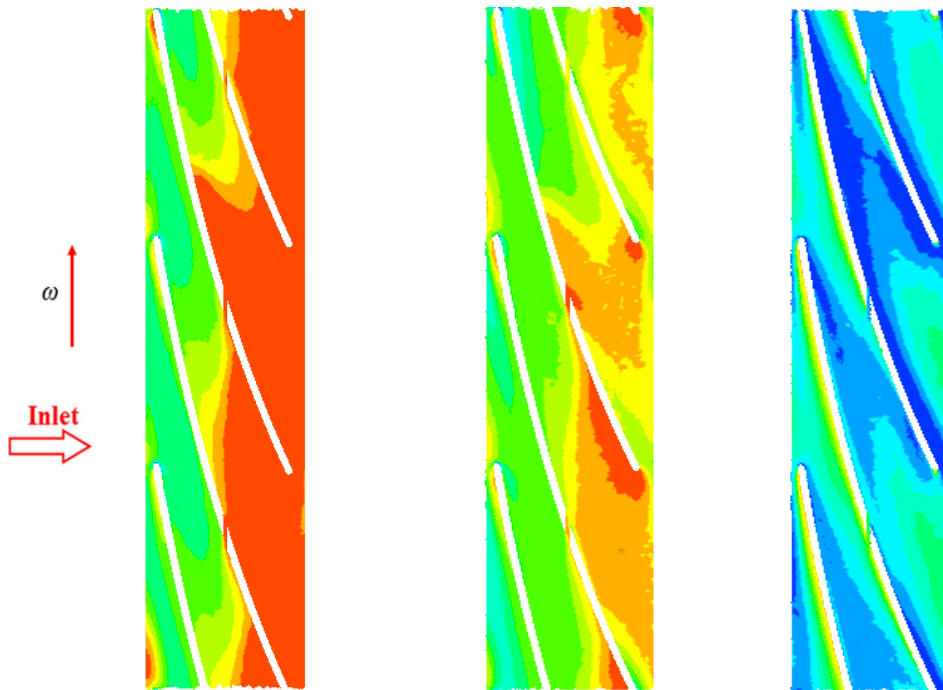
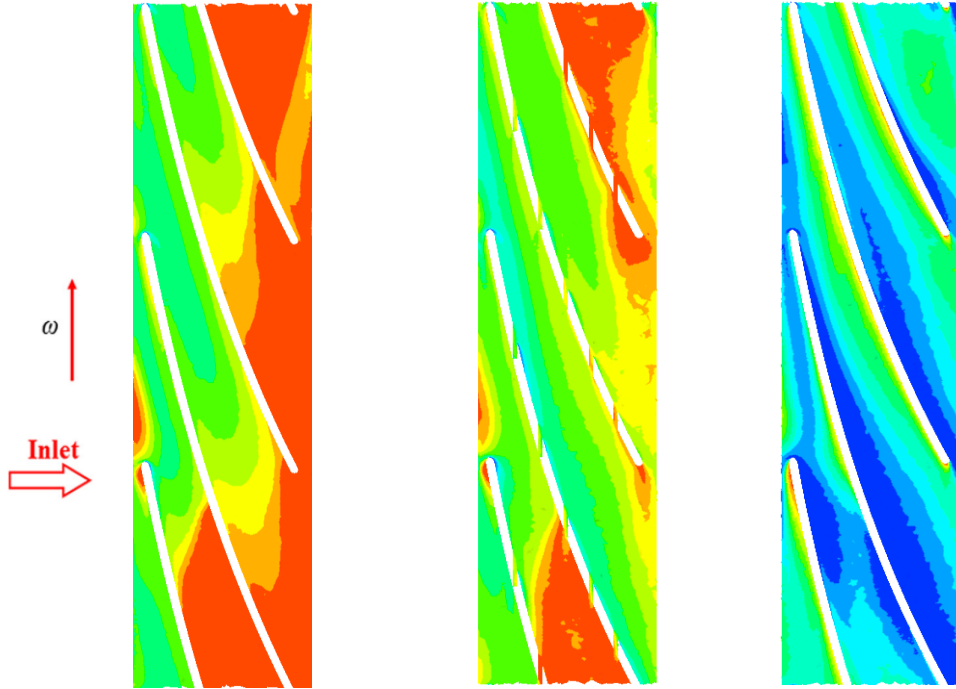


Figure 16 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m³/h, rotational speed: 3600rpm and inlet GVF is 50 %)



(a) 10 % span-wise (near hub) (b) 50 % span-wise (c) 90 % span-wise (near shroud)

Figure 17 Gas distribution of the impeller with split vane on different span-wise planes (flow rate: 100m³/h, rotational speed: 6000rpm and inlet GVF is 50 %)



(a) 10 % span-wise (near hub) (b) 50 % span-wise (c) 90 % span-wise (near shroud)

Figure 18 Gas distribution of the impeller with rectangle holes on different span-wise planes (flow rate: 100m³/h, rotational speed: 6000rpm and inlet GVF is 50 %)

4.4 Oil-air Two-phase Flow Simulation

To conduct further research on the multiphase pump performance, another type of liquid-gas fluids pairs inside the pump is analyzed. In this study, oil-air two-phase flow is used as the medium for the simulation instead of water-air mixture. Arabian Heavy Crude Oil is selected as continuous phase, whose degree of API gravity is 27.4° API. The pump performance curves at different rotational speeds and inlet GVFs are shown in Fig. 20. In

the CFD simulation mode, the flowrate is regulated to $100\text{m}^3/\text{h}$. And two rotational speeds are selected, which are 3600 rpm and 6000 rpm, respectively. It is clearly that the pressure rise curve of oil- air mixture has a similar trend with the curve of water-air flow. The pressure rise still becomes larger as pump rotating speed increases, and reduces as inlet GVF grows. New modified pump with rectangle holes is selected to compare with the original pump. It is obvious that modified pump has a better boosting capacity of pressure increment than original pump, especially at relative high inlet GVFs. For the rotodynamic pump, the multiphase flow is mainly subjected to centrifugal force. Due to large difference of density between heavy oil and air, it consequently causes the separation of gas phase and liquid phase when rotational speed and inlet GVF increase. Then the existence of the small holes on the impeller blades breaks the large gas bubbles, contributing to the remixture of the separated gas bubble as well as liquid fluid in the second half part of the flow passage. However, when inlet GVF is up to 90%, the oil-air two-phase flow is almost considered as pure gas flow and the performance development of the modified pump declines greatly. The pressure rise results of modified pump and original pump at 10% and 90% inlet GVFs are almost same. Similar discussion is also justified for the simulation with air-water flow.

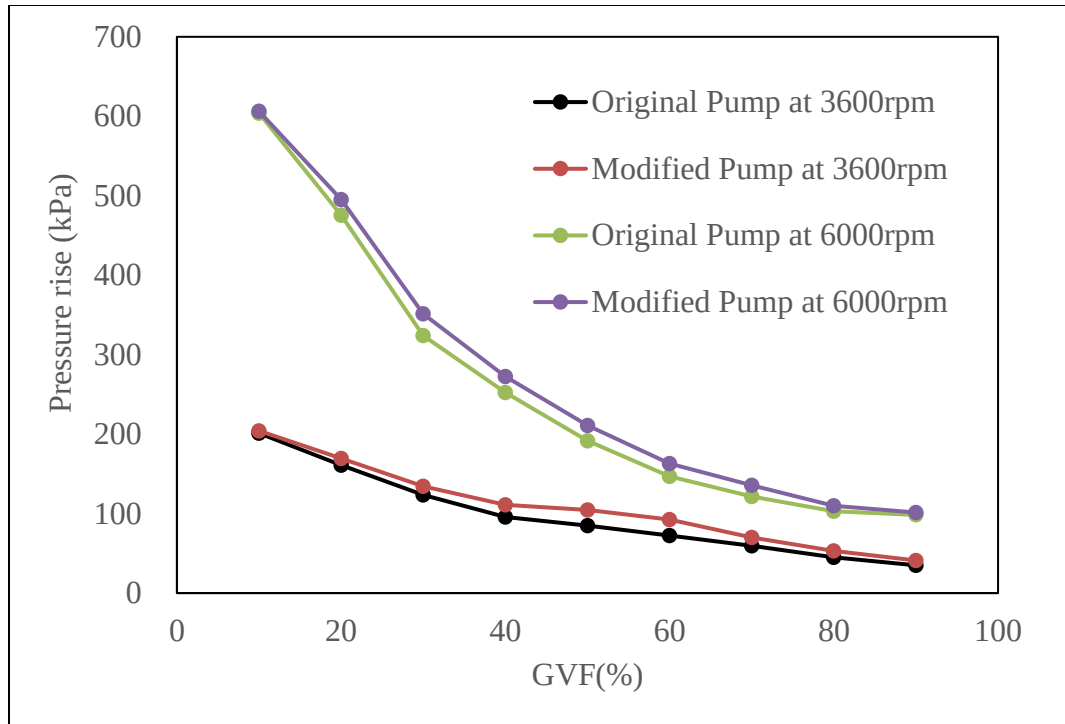


Figure 19 Predicted pressure rise comparison between modified pumps and original pump with oil and air versus different inlet GVFs and rotational speeds (flow rate: 100m³/h)

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

In this study, numerical simulations are utilized to perform a comprehensive study on the hydraulic performance of a single-stage rotodynamic helico-axial multiphase pump on the gas-liquid flow conditions. Then some development by modifying the impeller blade, including making slits or holes in the blades, is studied to enhance the performance of the pump system especially for high inlet GVFs and high rotational speeds. Gas gathering situation in the flow field of both the original impellers and the modified impellers at different conditions are analyzed. The pump performance results of above impellers are compared to illustrate the boosting capacity of the modification.

5.1 Conclusions

1. The CFD simulations on the single-stage helico-axial pump performance with single-phase flow ($\text{GVF} = 0\%$) and gas-liquid two-phase flow ($\text{GVF} = 10\%$ and 20%) are successfully performed and validated with the measurements from the experiment.
2. For two-phase flow simulations such as water-air and oil-air, a steady-state 3D CFD scheme with SST turbulence model is implemented on the rotodynamic pump to evaluate the pressure rise under various flow conditions by employing the frozen-rotor technique in CFX. The pressure increment increases as the pump rotational speed

increases and decreases as inlet GVF increases. At low inlet GVFs, the numerical simulation results of pump pressure rise with assumed bubble size show a reasonable qualitative agreement with experimental measurements.

3. Due to the centrifugal force, water moves toward the shroud and air gathers near the hub. As a result, high gas volume fraction appears mainly at the hub, especially near the outlet region inside the impeller. When the pump rotates faster, the centrifugal force increases and the separation phenomenon becomes more obvious. More high gas volume area occurs in the flow passages of the multiphase pump.
4. For the influence of bubble diameter, the pump performance declines as average bubble size increases. The inlet GVF and rotational speed should be mainly considered since they have a large influence on the bubble diameters. Raising the inlet GVF leads to the increment of the bubble size, while boosting the rotational speed results in the decline of bubble size.
5. The modified impellers break the large gas bubbles, and remix the separated gas bubble and liquid fluid in the flow passage, which effectively develop the pump performance at specific high GVF ranges. It is recommended to select the modified pump at inlet GVF from 30% to 70%. The difference of the modified and original pumps is not obvious when inlet GVF is extremely low or high.

5.2 Recommendations

The following parts are suggested in future studies:

1. The numerical simulations can be improved by studying the transient simulation of

helicon-axial pump dynamic phenomena, so that the two-phase flow characteristics inside the rotodynamic pump can be better simulated. In addition, incorporating more sophisticated closure relationships is considered, e.g. poly-dispersed phases (oil, water, air and CH₄).

2. Although the influence of the no-drag force is relatively limited when compared with centrifugal force on the high rotating speed condition, it is still useful to add multi-momentum-transfer mechanisms (virtual mass force, turbulence dispersion force, wall lubrication force etc.) to improve the accuracy of the prediction results, especially at high inlet GVF ranges.
3. For the mathematic modeling, the drag coefficient correction can be introduced to improve the simulation results since typical drag models are based on the experiment in the static or laminar flow rather than turbulence flow in a rotating pump.
4. The further relationship of bubble diameter and GVF in a turbulent flow field inside a rotodynamic pump can be improved by comparing different available models.
5. Since gas mainly gathers from middle position to the outlet region inside the impeller, it is recommended to conduct more modifications on the second half part of the blade in the axial direction, such as adding more holes or slits, which may have a better improvement on the pump performance.

NOMENCLATURE

A	interfacial area density, m^{-1}
C_D	drag coefficient
CFD	computational fluid dynamics
d_p	particle diameter, m
D	interphase drag force
D_ω	cross-diffusion term
$\vec{F}$	external body force
$\overrightarrow{F_{lift}}$	lift force
$\overrightarrow{F_{vm}}$	virtual mass force
M	Morton number
$\vec{u}$	phase velocity, m/s
U_T	terminal velocity, m/s
G	generation term
GVF	gas volume fraction

r_e	equivalent radius
Re	Reynolds number
S	user-defined source term
Y	dissipation term

Greek Symbols

α	volume fraction
Γ	effective diffusivity term
μ_{tc}	shear induced eddy viscosity
μ_{tp}	particle induced eddy viscosity
ρ	density, kg/m ³
σ	Prandtl number
$\bar{\bar{\tau}}$	stress-strain tensor
ν_t	kinematic eddy viscosity

Subscripts

c	continuous phase
d	dispersed phase

g	gas phase
i	phase i
k	turbulence kinetic energy
l	liquid phase
ω	the specific dissipation

References

- Acrivos, A. 1979. "Bubbles, Drops and Particles. By R. C LIFT , J. R. G RACE and M. E. W EBER . Academic Press, 1978. 380 Pp. £20.80 or \$32.00." *Journal of Fluid Mechanics* 94 (4): 795–96. <https://doi.org/10.1017/s0022112079221290>.
- Andrade, Jesus, Sigbjørn Sangesland, and Milan Stanko. 2016. "AVAILABLE TECHNOLOGIES AND PERFORMANCE PREDICTION MODELS FOR MULTIPHASE BOOSTING." <https://www.researchgate.net/publication/303823449>.
- Ban, Yaotao, Hong Zhao, and Dunsong Xue. 2000. "Performance Prediction Model of Helico-Axial Multiphase Pump." *Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics* 21 (2).
- Boyarshinova, A., V. Lomakin, and A. Petrov. 2019. "Comparison of Various Simulation Methods of a Two-Phase Flow in a Multiphase Pump." In *IOP Conference Series: Materials Science and Engineering*. Vol. 589. <https://doi.org/10.1088/1757-899X/589/1/012014>.
- Cao, Shuliang, Guoyi Peng, and Zhiyi Yu. 2005. "Hydrodynamic Design of Rotodynamic Pump Impeller for Multiphase Pumping by Combined Approach of Inverse Design and CFD Analysis." *Journal of Fluids Engineering, Transactions of the ASME* 127 (2). <https://doi.org/10.1115/1.1881697>.
- Dal Porto, D. F., and L. A. Larson. 1997. "Multiphase-Pump Field Trials Demonstrate Practical Applications for the Technology." *SPE Production and Facilities*. <https://doi.org/10.2118/36590-PA>.
- Deen, N G, T Solberg, and B H Hjertager. 2001. "Large Eddy Simulation of the Gas-Liquid Dow in a Square Cross-Sectioned Bubble Column." *Chemical Engineering Science*. Vol. 56. www.elsevier.com/locate/ces.
- Dorenbos, Cornelis Kees, Dietrich Müller-Link, and Axel Jäeschke. 2001. "Sand Handling During Multiphase Operations With Twin-Screw Pumps." In . <https://doi.org/10.2118/69846-ms>.

- Falcimaigne, J., J. Brac, Y. Charron, P. Pagnier, and R. Vilagines. 2002. "Multiphase Pumping: Achievements and Perspectives." *Oil & Gas Science and Technology* 57 (1). <https://doi.org/10.2516/ogst:2002007>.
- Faustini, Roberto, and Frank Kenyery. 2007. "3D Flow Modeling In One Helical-Axial Multiphase Pump Stage through CFD."
- Gulich, Johann Friedrich. 2014. *Centrifugal Pumps*. *Centrifugal Pumps*. Vol. 9783642401145. Springer-Verlag Berlin Heidelberg. <https://doi.org/10.1007/978-3-642-40114-5>.
- Hua, Gong, Gioia Falcone, Catalin Teodoriu, and Gerald L. Morrison. 2011. "Comparison of Multiphase Pumping Technologies for Subsea and Downhole Applications." In *Proceedings - SPE Annual Technical Conference and Exhibition*. Vol. 4. <https://doi.org/10.2118/146784-ms>.
- Joonhyung, Kim, Choi Youngseok, and Yoon Joonyong. 2014. "Development of Multiphase Pump for Offshore Plant." *Transactions of the Korean Society of Mechanical Engineers, B* 38 (2). <https://doi.org/10.3795/KSME-B.2014.38.2.183>.
- Kan, Nengqi, Zongku Liu, Guangtai Shi, and Xiaobing Liu. 2021. "Effect of Tip Clearance on Helico-Axial Flow Pump Performance at off-Design Case." *Processes* 9 (9). <https://doi.org/10.3390/pr9091653>.
- Kim, Joon Hyung, Him Chan Lee, Jin Hyuk Kim, Young Seok Choi, Joon Yong Yoon, Il Soo Yoo, and Won Chul Choi. 2015a. "Improvement of Hydrodynamic Performance of a Multiphase Pump Using Design of Experiment Techniques." *Journal of Fluids Engineering, Transactions of the ASME* 137 (8). <https://doi.org/10.1115/1.4029890>.
- . 2015b. "Improvement of Hydrodynamic Performance of a Multiphase Pump Using Design of Experiment Techniques." *Journal of Fluids Engineering, Transactions of the ASME* 137 (8). <https://doi.org/10.1115/1.4029890>.
- Lastra, Rafael, and Ikouwem Johnson. 2005. "Feasibility Study on Application of Multiphase Pumping towards Zero Gas Flaring in Nigeria." In *Society of Petroleum Engineers - Nigeria Annual International Conference and Exhibition 2005, NAICE 2005*. <https://doi.org/10.2118/98800-ms>.
- Li, Qingping, and Dunsong Xue. 2000. "Experimental Studies on Performances of the Helico-Axial Multiphase Pump." *Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics* 21 (4).

- Li, Yongjiang, Zhiyi Yu, and Weihua Sun. 2021. "Drag Coefficient Modification for Turbulent Gas-Liquid Two-Phase Flow in a Rotodynamic Pump." *Chemical Engineering Journal* 417 (August). <https://doi.org/10.1016/j.cej.2021.128570>.
- Li, Zengliang, Changshan Miao, and Jizhi Li. 2008. "Experimental Evaluation on the Prototype of Helico-Axial Multiphase Pump." *Shiyou Xuebao/Acta Petrolei Sinica* 29 (4): 605–7.
- Liu, Ming, Lei Tan, and Shuliang Cao. 2020. "Method of Dynamic Mode Decomposition and Reconstruction with Application to a Three-Stage Multiphase Pump." *Energy* 208 (October). <https://doi.org/10.1016/j.energy.2020.118343>.
- Morrison, G, S Pirouspanah, and K Kirland. 2014. "Performance Evaluation of a Multiphase Electric Submersible Pump."
- Müller-Link, D., and G. Rohlfing. 2009. "Grown-up Twin-Screw Multiphase Pumps - from Marginal to Full Size Applications." In *14th International Conference on Multiphase Production Technology*.
- Pirouspanah, Sahand, Sujan R. Gudigopuram, and Gerald L. Morrison. 2017. "Two-Phase Flow Characterization in a Split Vane Impeller Electrical Submersible Pump." *Journal of Petroleum Science and Engineering* 148. <https://doi.org/10.1016/j.petrol.2016.09.051>.
- Saadawi, Hisham. 2007. "Operating Multiphase Helicoaxial Pumps in Series to Develop a Satellite Oil Field in a Remote Desert Location." In *Proceedings - SPE Annual Technical Conference and Exhibition*. Vol. 2. <https://doi.org/10.2118/109785-ms>.
- Serena, Alberto, and Lars E. Bakken. 2016. "Experimental Study of the Influence of the Operating Parameters on the Performance and Capability of a Mixed-Flow Multiphase Pump." In *Proceedings of the ASME Turbo Expo*. Vol. 9. <https://doi.org/10.1115/GT2016-56576>.
- Sharma, Susan Sunila. 2011. "Determinants of Carbon Dioxide Emissions: Empirical Evidence from 69 Countries." *Applied Energy* 88 (1). <https://doi.org/10.1016/j.apenergy.2010.07.022>.
- Shi, Y., and H. W. Zhu. 2020. "Research on the Flow Characteristic in a Multi-Stage Multiphase Pump by Numerical Simulations." *Journal of Applied Fluid Mechanics* 14 (2): 555–56. <https://doi.org/10.47176/jafm.14.02.31797>.

- Shi, Yi, Hongwu Zhu, Binbin Yin, Ruiting Xu, and Jiate Zhang. 2019. "Numerical Investigation of Two-Phase Flow Characteristics in Multiphase Pump with Split Vane Impellers." *Journal of Mechanical Science and Technology* 33 (4). <https://doi.org/10.1007/s12206-019-0317-y>.
- Shi, Yi, Hongwu Zhu, Jinya Zhang, Binbin Yin, Ruiting Xu, and Junlin Zhao. 2017. "Investigation of Condition Parameters in Each Stage of a Three-Stage Helico-Axial Multiphase Pump via Numerical Simulation." In *Proceedings of the International Offshore and Polar Engineering Conference*, 156–62. Society of Petroleum Engineers.
- Shi, Yi, Hongwu Zhu, Jinya Zhang, Jiate Zhang, and Junlin Zhao. 2018. "Experiment and Numerical Study of a New Generation Three-Stage Multiphase Pump." *Journal of Petroleum Science and Engineering* 169. <https://doi.org/10.1016/j.petrol.2018.06.011>.
- Suh, Jun Won, Young Seok Choi, Jin Hyuk Kim, Kyoung Yong Lee, and Won Gu Joo. 2017. "Multiphase Flow Analysis for Air-Water Bubbly Flow in a Multiphase Pump." In *American Society of Mechanical Engineers, Fluids Engineering Division (Publication) FEDSM*. Vol. 1A-2017. <https://doi.org/10.1115/FEDSM2017-69239>.
- Suh, Jun Won, Jin Hyuk Kim, Young Seok Choi, Won Gu Joo, and Kyoung Yong Lee. 2017. "A Study on Numerical Optimization and Performance Verification of Multiphase Pump for Offshore Plant." *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* 231 (5): 382–97. <https://doi.org/10.1177/0957650917702263>.
- Suh, Jun Won, Jin Woo Kim, Young Seok Choi, Jin Hyuk Kim, Won Gu Joo, and Kyoung Yong Lee. 2018. "Development of Numerical Eulerian-Eulerian Models for Simulating Multiphase Pumps." *Journal of Petroleum Science and Engineering*. <https://doi.org/10.1016/j.petrol.2017.10.073>.
- Xu, Yun, Shuliang Cao, Takeshi Sano, Tokiya Wakai, and Martino Reclari. 2019. "Experimental Investigation on Transient Pressure Characteristics in a Helico-Axial Multiphase Pump." *Energies* 12 (3). <https://doi.org/10.3390/en12030461>.
- Yu, Zhiyi, Baoshan Zhu, and Shuliang Cao. 2015. "Interphase Force Analysis for Air-Water Bubbly Flow in a Multiphase Rotodynamic Pump." *Engineering Computations (Swansea, Wales)* 32 (7). <https://doi.org/10.1108/EC-10-2014-0210>.
- Zhang, Jin Ya, Shu Jie Cai, Hong Wu Zhu, and Rui Qiang. 2015. "Experimental Study of Gas-Liquid Two-Phase Flow Pattern in a Helico-Axial Multiphase Pump by

- Visualization.” *Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics* 36 (9): 1937–41.
- Zhang, Jinsong, Honggang Fan, Wei Zhang, and Zhifeng Xie. 2019. “Energy Performance and Flow Characteristics of a Multiphase Pump with Different Tip Clearance Sizes.” *Advances in Mechanical Engineering* 11 (1). <https://doi.org/10.1177/1687814018823356>.
- Zhang, Jinya, Shujie Cai, Yongjiang Li, Hongwu Zhu, and Yongxue Zhang. 2016. “Visualization Study of Gas-Liquid Two-Phase Flow Patterns inside a Three-Stage Rotodynamic Multiphase Pump.” *Experimental Thermal and Fluid Science* 70. <https://doi.org/10.1016/j.expthermflusci.2015.08.013>.
- Zhang, Jinya, Shujie Cai, Hongwu Zhu, Ke Yang, and Rui Qiang. 2014. “Numerical Investigation of Compressible Flow in a Three-Stage Helico-Axial Multiphase Pump.” *Nongye Jixie Xuebao/Transactions of the Chinese Society for Agricultural Machinery* 45 (9): 89–95. <https://doi.org/10.6041/j.issn.1000-1298.2014.09.015>.
- Zhang, Jinya, Shujie Cai, Hongwu Zhu, and Yongxue Zhang. 2015. “Experimental Investigation of the Flow at the Entrance of a Rotodynamic Multiphase Pump by Visualization.” *Journal of Petroleum Science and Engineering* 126. <https://doi.org/10.1016/j.petrol.2014.12.017>.
- Zhang, Jinya, Hongwu Zhu, Yan Li, and Chun Yang. 2009. “Optimization of Helico-Axial Multiphase Pump Impeller Based on Orthogonal Experimental Design.” In *Fourth International Conference on Experimental Mechanics*, 7522:75225W. SPIE. <https://doi.org/10.1117/12.850693>.
- Zhang, Jinya, Hongwu Zhu, and Huan Wei. 2011. “Three-Dimensional Blade Design of Helico-Axial Multiphase Pump Impeller Based on Numerical Solution of Meridian Flow Net and Blade Mean Camber Lines.” In *ASME-JSME-KSME 2011 Joint Fluids Engineering Conference, AJK 2011*. Vol. 1. <https://doi.org/10.1115/AJK2011-06030>.
- Zhang, Jinya, Hongwu Zhu, Chun Yang, Yan Li, and Huan Wei. 2011. “Multi-Objective Shape Optimization of Helico-Axial Multiphase Pump Impeller Based on NSGA-II and ANN.” In *Energy Conversion and Management*. Vol. 52. <https://doi.org/10.1016/j.enconman.2010.07.029>.
- Zhang, Wenwu, Zhiyi Yu, Muhammad Noaman Zahid, and Yongjiang Li. 2018. “Study of the Gas Distribution in a Multiphase Rotodynamic Pump Based on Interphase Force Analysis.” *Energies* 11 (5). <https://doi.org/10.3390/en11051069>.

- Zhang, Wenwu, Zhiyi Yu, and Baoshan Zhu. 2017. "Numerical Study of Pressure Fluctuation in a Gas-Liquid Two-Phase Mixed-Flow Pump." *Energies* 10 (5). <https://doi.org/10.3390/en10050634>.
- Zhang, Yongxue, Jinya Zhang, Hongwu Zhu, and Shujie Cai. 2014. "3D Blade Hydraulic Design Method of the Rotodynamic Multiphase Pump Impeller and Performance Research." *Advances in Mechanical Engineering* 2014. <https://doi.org/10.1155/2014/803972>.
- Zhu, Hong Wu, Qing Ping Li, Luo Chen, and Jin Ya Zhang. 2007. "Experimental Studies of the Helico-Axial Multiphase Pump." *Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics* 28 (4): 601–3.
- Zhu, Hong Wu, and Zhong Fang Li. 2004. "Compare with the Experimental Studies of the Helico-Axial Multiphase Pump." *Kung Cheng Je Wu Li Hsueh Pao/Journal of Engineering Thermophysics* 25 (4): 606–8.
- Zhu, Jianjun, and Hong Quan Zhang. 2017. "Numerical Study on Electrical-Submersible-Pump Two-Phase Performance and Bubble-Size Modeling." *SPE Production and Operations* 32 (3): 267–78. <https://doi.org/10.2118/170727-pa>.
- . 2018. "A Review of Experiments and Modeling of Gas-Liquid Flow in Electrical Submersible Pumps." *Energies*. <https://doi.org/10.3390/en11010180>.

Vitae

Name	:Zhenghua Shi
Nationality	:Chinese
Date of Birth	:12/23/2021
Email	:s951981056@outlook.com
Address	:Baolicheng, Wuhan, China
Academic Background	: (1) Bachelor of Science in Thermal and Power Engineering in 2016 from Huazhong University of Science and Technology, Wuhan, China. (2) Master of Science in Mechanical Engineering in 2021 from King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia