Investigation on the gas pockets in a rotodynamic multiphase pump

J Y Zhang¹, Y J Li, S J Cai, H W Zhu, Y X Zhang

College of Mechanical and Transportation Engineering, China University of Petroleum, Beijing

E-mail: zhjinya@163.com

Abstract The appearance of gas pockets has an obvious impact on the performance of the rotodynamic multiphase pump. In order to study the formation of gas pockets in the pump and its effects on pump’s performance, the unsteady numerical simulation and the visualization experiments were done to investigate gas pockets in a three-stage rotodynamic multiphase pump developed by authors. Meanwhile, the mixture of water and air was selected as the medium. According to the distributions of pressure, gas volume fraction and velocity vector in three compression cells in unsteady flow process, the process of the formation of gas pockets in the pump were analysed generally. The visualization experiments were used to verify the validity of the numerical simulation. The results will be benefit for the hydraulic design of the compression cell of rotodynamic multiphase pump.

1. Introduction

Due to the further demand of energy, the active development of oilfield has become the inevitable trend of society. Because of the overexploitation of the oilfield, the phenomenon that the appearance of outgas from oilfield have restricted the production efficiency [1-3]. In order to prolong the life of the oilfield, the Multiphase Mixed Transportation Technology, which means replacing traditional flow pumps and gas compressors with a multiphase pump, has been developed. The technology has the following features: more simplified devices, low Wellhead Back Pressure, all these are benefit to the production of oil and gas.

But with the application of multiphase pumps, disadvantages emerged: First, due to the difference of density and speed between gas and liquid, the separation of gas-liquid two-phase has appeared; second, the gas pocket forms easily in high gas volume fraction, which causes big flow losses and the sharp drop of the performance of multiphase pumps. Therefore, it is important to study the gas pocket to reduce the separation of gas-liquid. For the further study, a three-stage rotodynamic multiphase pump developed by authors has been selected to do unsteady numerical simulation, visualization experiments were also used to verify the validity of the numerical simulation.

¹ Corresponding author: J Y Zhang
2. **the pump model and the numerical simulation**

A three-stage rotodynamic multiphase pump developed by authors was adopted in this paper. It contains three compression cells, a diversion cone at the inlet and a diffuser at the outlet of the pump, which is shown in Figure 1. The front and back extension were also set in the pump.

![Fig.1 Model of rotodynamic multiphase pump](image)

### 2.1 The geometry model and the mesh

The software CFX-BladeGen was used to establish the impeller and the diffuser models. The number of the impeller blade was three while the diffuser blade was ten. There was a gap whose distance is 2% times as the span near the rim of the impeller. Large wrap angle was selected to avoid the separation of the gas and the liquid. The software CFX-TurboGrid was used to generate mesh. The number of every impeller mesh was 410448 while every diffuser mesh was 142393, all these are shown in Figure 2. Finally, the hexahedron structured mesh was generated, as is shown in Figure 3.

![Fig. 2 Mesh of the impeller](image)  ![Fig. 3 Mesh of the diffuser](image)

### 2.2 calculation settings

The commercial software ANSYS-CFX was adopted in this paper. It contains the conservation of Finite Volume Method and the accuracy of Finite Element Method.

Due to the little change of the temperature in gas-liquid two-phase medium[5], the thermodynamic process was considered as an isothermal process, the reference pressure was set at 0.1 MPa, the boundary condition was set at mass flow in the inlet while average static pressure in the outlet, the turbulence model SST which is based on k-ω was applied. The residual convergence value was 1e-6 and with the value condition we can judge the stability of the numerical simulation.
interface of impellers and diffusers was defined as the mixing surface boundary.

The steady numerical simulation has been done when rotational speed was 2700 r/min, the volume flow was 33 m3/h and the IGVF was 30%, the results were the initial file of unsteady numerical simulation, where the time step was set at $\Delta t = 1.85 \times 10^{-4}$ s.

3. Results and discussion

3.1 Analysis of the flow field at a certain time

3.1.1 The distribution of velocity and pressure. When the calculation reached stable, the velocity vector (Figure 4) has been selected to analyse vortices in different spans of impellers and diffusers. Conclusions can be obtained:

(1) From the span 0.1 to 0.9, the movement of gas-liquid two-phase in impellers and diffusers was more regular. In the span 0.1, different vortices appeared in three stages within impellers and diffusers. In the span 0.9, smaller vortices were only existed in the outlet of every diffuser passage. The centrifugal force of water was so big that the water moved to the rim while the gas gathered in the hub.

(2) In the span 0.1, the number of the vortices in three compression cells decreased gradually from the first to the third stage in the passages. Vortices in impeller passages stretched from the pressure to suction surface of blades at the first and second stage, but only stayed in pressure surfaces at the third stage. This phenomenon indicated that there was a dynamic and static interference between impellers and diffusers. Vortices prevented the fluid from moving downstream, so that the vortex in impeller passages stretched from the pressure surface to suction surface of blades.

![Fig. 4 Air velocity on the cascade plane](image)

Figure 5 shows the pressure distribution in different spans with the unsteady numerical simulation. We can find that pressure distributions of the gas-liquid two-phase in impeller passages and diffuser passages were uneven. The fluid was not only affected by the rotational impeller that the fluid flowed through, but also affected by the post rotational impeller.
3.1.2 GVF distributions. Figure 6 shows the GVF distribution in different spans, conclusions can be obtained: In the span 0.1, the high GVF area in impellers of the first stage was so large that it stretched to the suction surface of blades. At the second stage, the high GVF area in impellers was smaller than the first one but still stretched to the suction surface. At the third stage, the high GVF area no longer stretched to the suction surface. The phenomenon showed that GVF was affected by the complex flow situation of the diffuser behind. Vortices formed and pretended the gas from diffusing downstream.

In the span 0.9, the high GVF area almost disappeared in the impeller passage. Unlike the span 0.1, the GVF areas were only existed in suction surface at the first and second stage.

3.2 Analysis of the flow field with time

3.2.1 Pressure distributions with time. From the Figure 7 we can get the analysis. Take four passages of diffusers, label A, B, C and D. At the time $t$, an impeller blade moved to passage A, the pressure of passage A was higher than passage B; at the time $t + 10\Delta t$, the blade head of the impeller passed
passage A, the pressure in passage A decreased obviously while the pressure in passage B increased; at the time \( t + 20\Delta t \), the impeller blade moved to passage B, the pressure in passage A further decreased and the pressure in passage B increased continuously. The phenomenon can also be observed in passage C and D, at the time \( t + 30\Delta t \), the impeller blade has not reached passage C, the pressure in passage C and D were uniform; at the time \( t + 40\Delta t \), the impeller blade moved to passage C, we can find the pressure in passage C was higher than passage D. All these can conclude that the pressure distribution was affected significantly by the rotational impeller downstream, the pressure in diffusers had a periodic change.

**Fig. 7** GVF distributions on the cascade plane with the time

3.2.2 *GVF distributions with time.* Figure 8 shows the GVF distribution with time in diffusers at the first and second stage and the impellers at the second stage. Gas pockets formed when the blade head
of impellers continuously passed the front diffuser blade tail. A series of discrete gas pockets moved downstream along the impeller passage, and then entered the high GVF area.

From the unsteady calculation results, we find that the transport of gas from the inlet to outlet of impellers was not continuous, but discrete with the formation of gas pocket one after another. At the time $t$, when a blade head in impellers was closed to a blade tail in diffusers at the first stage, the gas pocket was about to appear in the blade head of impellers. At the time $t + 10\Delta t$, gas pockets formed when the blade head of impellers passed the blade tail of diffusers, with the further movement of impeller blades, bubbles grew bigger and moved downstream. “Discrete movement of gas pocket”, the phenomenon happened when the blade of impellers passed the blade of diffusers, all these showed that the appearance of bubbles were affected by the rotational impellers.

4. Experimental comparison
Compared with the experiment, we find that the experimental phenomena were in good agreement with the results of unsteady calculation. As is shown in Figure 9 and Figure 10:
(1) Gas pockets first appeared in impeller passages at the first stage. There were no small bubbles closed to the gas pocket, and the phenomenon meant that bubbles around gathered the gas pocket. Fluid was forced to move along the gap due to the existence of gas pockets.

(2) Compared to the first stage, the position of the gas pocket at the second stage was behind the first one and the scale of the pocket was smaller. Similar to the first stage, the gas pocket was closed to the impeller hub and stretched from the pressure surface to suction surface of blades. The difference was caused by the compressibility of gas.

(3) The position of the gas pocket at the third stage was behind the second one and the scale of the pocket was smallest. Similar to the first and second stage, the gas pocket was closed to the impeller hub and stretched from the pressure surface to suction surface.

Fig. 9 Gas pockets obtained from transient simulation

Fig. 10 Gas pockets obtained from visualization experiment

5 Calculations
The mixture of water and air was selected as the medium. A three-stage rotodynamic multiphase pump developed by authors has been selected to do unsteady numerical simulation and visualization experiments. With the unsteady calculation, we analysed the velocity distribution, the pressure distribution and the GVF distribution in the second compression cell, as well as the formation of discrete gas pockets.

(1) Gas pockets appeared both in pressure surface and suction surface. It first gathered in pressure surface, then stretched to the suction surface for the rotational impellers, finally be closed to the outlet of impellers.

(2) The gas mainly gathered in pressure surface of the impeller hub. There were a lot of vortices in the hub and the flow was in disorder. The supercharging increased gradually from the first to third compression cell but decreased with the increase of the GVF.

(3) The alternative variation of pressure and GVF at the interface was caused by the dynamic and static interference between impellers and diffusers, which had an effect on flow upstream and
downstream; the transport of gas from the inlet to outlet of impellers was not continuous, but discrete with the formation of gas pocket one after another.
References

[1] Cheng L, Chen T 1999 The Performance and the Present Research Situation of Multiphase Pumps. *Journal of Oil Field Equipment*, 28(5):22-25.

[2] Li Q, Xue D 2000 The Research on Multiphase Pumps of Oil and Gas, *Journal of China Offshore Oil and Gas*, 12(1): 47-51.

[3] Yves C, Elise A, Sylvain S 2003 Two-Phase Helical Axial Turbines. *SPE Annual Technical Conference and Exhibition*, Colorado, 2003: SPE 84062.

[4] Abelsson C, Busland H, Henden R, et al. 2011 Development and testing of a hybrid boosting pump[C]/ *Proceedings of the Offshore Technology Conference*. Houston: Offshore Technology Conference, 2011: OTC 21516

[5] Gao T, Yang D, Cao F, et al. 2011 Temperature and Thermodynamic Deformation Analysis of the Rotors on a Twin Screw Multiphase Pump with High Gas Volume Fractions. *Journal of Zhejiang University Science A*, 12(9): 720-730.

[6] Caridad J, Asuaje M, Kenyery F, et al. 2008 Characterization of a Centrifugal Pump Impeller under Two-Phase Flow Conditions. *Journal of Petroleum Science and Engineering*, 63(1): 18-22.

[7] Zhang J, Cai S, Zhu H, et al. 2014 Experimental Investigation of The Flow at the Entrance of a Rotodynamic Multiphase Pump by Visualization. *Journal of Petroleum Science and Engineering*, 2(26): 254-261.

[8] Yu Z, Zhang Q, Huang R, et al. 2013 Numerical Simulation of Unsteady Flow in Multiphase Rotodynamic Pumps. *Journal of Drainage and Irrigation Machinery Engineering*, 04: 284-288.

[9] Sooran N, Seyed H, Hashemabadi. 2009 CFD simulation of Inlet Design Effect on Deoiling Hydrocyclones Separation Efficiency. *Chemical Engineering and Technology*, 32(12): 1885-1893.

[10] Dogru A H, Hamoud A A, Barlow S G 2004 Multiphase Pump Recovers More Oil in a Mature Carbonate Reservoir. *Journal of Petroleum Technology*, 56(02): 64-67.

[11] Humoud A A, Boudi A A, Al-Qahtani S D, et al. 2008 Deployment of a Rotodynamic Multiphase Pump in a Remote Hilly-terrain Oil Field in Saudi Arabia. *Abu Dhabi International Petroleum Exhibition and Conference*, Abu Dhabi, 2008: SPE 117462.

[12] Scott S L 2003 Multiphase Pumping Addresses a Wide Range of Operating Problems. *Oil & Gas Journal*, 101(37): 59-71.

[13] Li Q, 2001 The Performance Research on Multiphase Pump of Oil and Gas. *Journal of China Offshore Oil and Gas*, 13(2): 52-56.