Numerical simulation of two-phase flow in an energy recovery micro-hydraulic turbine based on Francis hydraulic model

S H Sun, Y Pang, P C Guo*, X B Zheng, J G Yan, D H He and X Q Luo

State Key Laboratory of Eco-hydraulics in Northwest Arid Region of China (Xi’an University of Technology), Xi’an 710048, Shaanxi Province, China

*E-mail address: guoyicheng@126.com

Abstract. In industrial processing, a centrifugal pump is mostly used reversely as the micro-hydraulic turbine for its lower cost. In some application situation, the working fluid will contain a part of gas which will reduce the efficiency, stability, and the power output of the machine. Due to the lower efficiency and stability of the pump as turbine (PAT) working with liquid flow, the working characteristic will deteriorate furtherly when the working fluid contains gas. Therefore, a micro-hydraulic turbine based on Francis turbine model is designed and proposed to except to obtain good performance under two-phase flow working condition. In this study, the three-dimensional transient two-phase flow model which adapts the k-ω model as turbulence model and Eulerian-Eulerian model as two-phase flow model with the commercial code ANSYS-CFX was established and solved to study the two-phase flow characteristics of the micro-hydraulic turbine with fixed guide vane openings under different gas volume fraction (GVF). The numerical simulation results show that the power, efficiency and torque of the turbine decrease with GVF. Due to the expansion of the gas in working fluid, the velocity of fluid increases with the gas, which leads to the bigger lower pressure area near the exit of the impeller. The vortex at the back of the blade is also intensified when the gas fraction increase. When the GVF is larger than 10%, the gas distribution is inhomogeneous, and most of them gather at the back side of the blade near the exit of the impeller where the GVF can reach 90%. The efficiency of the micro-hydraulic turbine is about 60% when the GVF is 20%, which needs to be improved in the future.

1. Introduction

In recent years, in addition to being a power generation device for small hydropower and micro-hydro power projects, PAT are widely used in various applications such as oil smelting hydrocracking, liquefied natural gas and natural gas purification, ammonia synthesis of fertilizers, desalination plants, and pipeline water supply [1-3]. In the process of purification of natural gas, liquefaction of natural gas, synthetic ammonia in fertilizers and urban water supply, the working fluid will be gas-liquid two-phase flow sometimes. Two-phase fluid will have a great impact on the efficiency and stability of the PAT and require careful research by researchers [4].

Many researchers have performed theoretical, experimental, and numerical studies on the performance parameters of the PAT when the pump is reversed. Singh [5] had found experimentally that the impeller rounding in centrifugal pumps as turbine can increase turbine efficiency by about 3% to 70%. Pugliese [6] tested two kinds of centrifugal pumps in reverse mode, whose efficiency reached about 68%. Derakhshan [7] had conducted theoretical, experimental, and numerical studies on the PAT. The results showed that the theoretical efficiency and the experimental efficiency were 72% and
70% respectively. Shi[8,9], Qian[10] and Yang[11-13] systematically studied the effects of the PAT’s geometry parameters on turbine’s performance. Their research results showed that under clean water conditions, the modification of geometric parameters can improve the efficiency of turbines by 5%-10%. In total, the efficiency of the PAT in clean water conditions is still low, no more than 70%, and the area of high efficiency is quite narrow.

Yang[14,15] separately performed numerical simulations of a single-stage single-suction centrifugal pump and a five-stage centrifugal pump which were used as turbine under gas-liquid two-phase flow conditions. The single-stage PAT’s efficiency decreased with the increase of GVF. When the GVF was 0.2, the efficiency of the turbine’s optimal operating point dropped by 4% to 55%. As the GVF increased, the pressure distribution inside the PAT became more uneven and the pressure gradient became larger. At the same time, the fluid velocity increased. And the high GVF area became larger in the flow channel. The efficiency of the multi-stage PAT was 73% in clean water condition, while 66% in two-phase condition. The head and power of the multi-stage PAT at the optimal operating point under the clean water condition were higher than that under the two-phase condition. Shi[16,17] studied the performance of a single-stage PAT under different GVF conditions. The results showed that with the increase of GVF, the gas accumulation phenomenon was more obvious. When the GVF was 0.2, the head loss of the PAT was about 40% and the efficiency was 65%.

The overall efficiency of the PAT was low under the two-phase flow conditions. Therefore, it is necessary to design a new turbine and study its performance under two-phase flow conditions. This paper proposes a micro-turbine which use the fixed guide vane based on the Francis hydraulic turbine model. The two-phase numerical simulation of the turbine was performed under the given guide vane opening. The characteristics of the two-phase flow of the turbine will be analysed and summarized.

2. Numerical Methods

2.1 Physical model and mesh generation
This model was based on the hydraulic model of the pump turbine. To reduce the axial force of the turbine and improve the mass flow rate, the blade was installed back to back. The model structure is shown in figure 1. The diameter of the runner is 0.65m; the outlet diameter of the runner is 0.32m; the inlet diameter of the volute is 0.32m; the double outlet diameter is 0.45m; the number of blades is 18; and 20 guide vanes are used.

![Figure 1. The model structure of pumps as turbines.](image)

The model grid and geometry diagram are shown in figure 2, 3 and 4. The mesh of the model was drawn with ICEM. Unstructured tetrahedral elements were applied for the fluid domain of the volute, the guide vane and double pipe structure outlet. In order to capture the flow characteristics of the near wall better, 8-layer boundary mesh structure was established on the blade and the guide vanes. The boundary layer mesh was drawn using unstructured hexahedrons. The total grids of turbine which used to calculate were approximately 7.9 million.
2.2 Turbulence model and boundary conditions
Considering that there are many guide vanes and the flow near wall region is complex, the SST k-w turbulence model was used to capture the flow characteristics near wall and in the main flow areas [18]. The volute inlet was given a uniform flow. The outlet pressure of the double pipe structure was static pressure. The inlet and outlet of the runner, the guide vanes and the outlet pipe were used interface transition calculations. The near-wall mesh was refined for the use of wall functions. The frozen-rotor technique, also referred to as moving reference frame, was used to simulate the interaction of rotating and stationary parts in turbine, which offers a better balance between the computational resource requirements and numerical accuracy [19].

3. Analysis of external characteristics of the turbine
The fixed guide vane opening of the turbine model is $11^\circ$. The operation points are shown in table 1. The inlet GVF of 0, 0.05, 0.1, 0.15, and 0.2 under every operation point were numerically simulated. The external characteristics of the turbine under different inlet GVF were compared and analysed.

| Operation point | a   | b   | c   | d   | e   |
|-----------------|-----|-----|-----|-----|-----|
| $Q_{11}(l/s)$   | 413 | 367 | 349 | 322 | 272 |
| $n_{11}(rpm)$   | 33.9| 39.9| 42.2| 44.0| 46.0|
Figure 5. Efficiency under different inlet GVF.

Figure 5 shows the efficiency of the turbine under different unit flow rate conditions when the inlet GVF increases from 0 to 0.2. The efficiency of the turbine increases with the uniform flow rate. And it decreases with the inlet GVF. From the Figure 5, it can be seen that the efficiency of the turbine in low flow rate condition is 62%. With the increase of the uniform flow rate, the efficiency of the turbine quickly climbs up to 80%, and then slowly reaches the maximum efficiency which is 90%. When the GVF is 0.2, the efficiency of the model in large flow condition is 21% lower than it in clean water condition.

Figure 6. Power under different inlet GVF.

Figure 7. Torque at different inlet GVF.

It can be seen from the figure 6. and 7, the power and torque of the turbine increase with the unit flow rate. The efficiency and torque of the model under two-phase condition are lower than that under the clean water condition. And they decrease with the increase of GVF. In the low flow rate condition, the GVF has little effect on the turbine’s power and torque. On the contrary, in the large flow rate condition, the inlet GVF has a great influence on the efficiency and torque of the turbine. The higher the GVF, the more power and torque are reduced. As the GVF increases, the flow rate of the liquid in the inlet working fluid becomes smaller, which results in a drop of output power and torque. In addition, when the GVF increases, the loss in the flow channel increases, and the turbine’s efficiency and torque may reduce.

4. Analysis of the internal flow field

This study selects five working conditions under the fixed guide vane opening to carry out the numerical simulation under different inlet GVF. The changes of the external characteristics of the
turbine are summarized. In order to understand the reason of the changes more deeply, the flow field inside the impeller at the operation point \((Q_{11}=0.413\text{m}^3/\text{s}, \ n_{11}=33.7\text{rpm})\) was analysed.

4.1 Static pressure distribution

![Image of static pressure distribution](image)

Figure 8. Static pressure distribution under different inlet GVF.

Figure 8. is the static pressure distribution inside the impeller when the inlet GVF changes from 0 to 0.2 through the outside of the impeller which rotates clockwise. The pressure inside the impeller decreases from the inlet side to the outlet side of the impeller, and the negative pressure area appears at the outlet side. The pressure at the working surface of the blade is greater than that at the suction surface. The area of the low pressure which appears at the outlet side of the impeller gradually increases along the impeller rotation direction. It also increases with the inlet GVF. Because of the guide vane structure, the pressure inside the impeller is evenly distributed. As the GVF increases, the area of the negative pressure area increases and the pressure gradient becomes bigger. More gas enters the flow passage with the increase of the GVF. As the gas expands, its volume becomes very big and occupies more space of the flow passage. Therefore, the flow area for liquid decreases largely and its velocity increases, which results in the low pressure of the outlet side.

4.2 Velocity vector distribution

![Image of velocity vector distribution](image)

(a) \(\phi=0\)  
(b) \(\phi=0.10\)  

(b) \(\phi=0.20\)
Figure 9. Velocity vector distribution.

Figure 9 is a plot of liquid phase velocity vector distribution inside the impeller under different inlet GVF. Due to the presence of the guide vane structure, the distribution of liquid-phase velocity vectors in each flow channel is approximately symmetrical. There is a similar size vortex at the back of the blade in each flow channel. When the inlet GVF increases, the vortex becomes bigger as shown in enlarged drawing. Near the front side of the impeller, the velocity vector in the flow channel becomes redder and bigger, and the high-speed area becomes larger when the volume fraction of gas increases. This is because the gas-containing fluid expands when it flows from the high pressure to the lower pressure. The larger the inlet volume fraction, the bigger volume the gas expands to. Due to the gas occupies the space, the liquid velocity increases. The pressure drops rapidly and forms a negative pressure area in the outlet side of the impeller. The closer to the exit, the negative pressure area is lager. The increase of the flow velocity and the vortex area on the back of the blade result in an increase of the flow loss in the flow channel. It causes the further decrease of its working ability.

4.3 Variable gas content distribution map

Figure 10. GVF distribution map.
Figure 10. is a GVF inside the impeller. The volume fraction of gas in each flow channel gradually increases along the direction of rotation of the impeller. It increases from the inlet side to the outlet side of the impeller. When the GVF is greater than 10%, the gas is full of the entire impeller flow passage. But the gas distribution is not even, and even 90% of the GVF distribution is occurred. Under the two-phase flow condition, the gas and the liquid separate in the flow channel and collect at the back of the impeller near the outlet. According to the definition of internal mechanical Coriolis force in the impeller, it can be seen that the direction of the Coriolis force in the impeller is from the back of the impeller to the working surface. Because of the small mass of the gas molecules and the large molecular mass of the liquid, most of the gas molecules are squeezed toward the back of the impeller under the influence of the Coriolis inertial force.

5. Conclusion
In this study, a micro-hydraulic turbine based on Francis turbine model is designed. The external characteristics and flow field distribution of the turbine under different uniform flow rates and inlet GVF were studied. The following conclusions were obtained.

(1) The efficiency, power and torque of the turbine decrease with the increase of the volume fraction of the inlet gas.

(2) As the volume fraction of gas increases, the pressure gradient inside the impeller increases, and the area of the low-pressure area at the exit of the impeller increases.

(3) With the increase of the GVF, the flow velocity in the impeller increases because the gas occupies a larger flow area after decompression and expansion. It results in an increase in liquid phase velocity. There is a vortex on the back of the blade, which increases with increasing GVF.

(4) The volume fraction of gas in the impeller gradually increases from the inlet side to the outlet side of the impeller. When the gas phase volume fraction is greater than or equal to 10%, the gas distribution in the flow channel is not uniform. Gathering of gas at the back of the impeller near the outlet of the gas, the volumetric gas content reaches over 90%. Coriolis inertial force is the main reason for gas accumulation.

Acknowledgments
This work was supported by the National Natural Science Foundation of China (Grant No. 51479166), the Key Research and Development Program of Shaanxi Province (Grant No. 2017ZDXM-GY-081), the China postdoctoral science foundation (Grant numbers: 2015M580866), the Scientific Research Program of Shaanxi Provincial Education Department (Grant No. 17JF019) and the Natural Science Foundation of Shaanxi Province (Grant No. 2018JM5147).

References
[1] Fecarotta O, Caravetta A and Ramos H-M 2011 CFD and comparisons for a pump as turbine: mesh reliability and performance concerns Int J Energy Environ 2 39-48
[2] Kramer M, Terheiden K and Wieprecht S 2018 Pumps as turbines for efficient energy recovery in water supply networks Renewable Energy 122 17-25
[3] Binama M, Su W-T, Li X-B, Li F-C, Wei X-Z and An S 2017 Investigation on pump as turbine (PAT) technical aspects for micro hydropower schemes: A state-of-the-art review Renewable and Sustainable Energy Reviews 79 148-79
[4] Steller J, Adamkowski A, Stankiewicz Z, Łojek A, Rduch J and Zarzycki M 2008 Pumps as turbines for hydraulic energy recovery and small hydropower purposes in Poland. In: Proceedings of the International Conference HIDROENERGIA, (Bled,Slovenia: p.10)
[5] Singh P and Nestmann F 2011 Internal hydraulic analysis of impeller rounding in centrifugal pumps as turbines Experimental Thermal and Fluid Science 35 121-34
[6] Pugliese F, De Paola F, Fontana N, Giungi M and Marini G 2016 Experimental characterization of two Pumps As Turbines for hydropower generation Renewable Energy 99 180-7
[7] Derakhshan S and Nourbakhsh A 2008 Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation *Experimental Thermal and Fluid Science* **32** 1620-7

[8] Shi G-T, Yang J-H, Miao S-C and Li T-L 2015 Calculation of pressure fluctuation within volute of hydraulic turbine under different guide vane numbers *Journal of Aerospace Power* **30** 1228-35 (In Chinese)

[9] Yang J-H, Gong C-H and Xia S-Q 2014 Numerical analysis on influence of guide vanes on performance of centrifugal pump acting as hydraulic turbine *J Drain Irrig Mach Eng* **32** 113-8 (In Chinese)

[10] Qian Z, Wang F, Guo Z and Lu J 2016 Performance evaluation of an axial-flow pump with adjustable guide vanes in turbine mode *Renewable Energy* **99** 1146-52

[11] Wang T, Kong F-Y and Yang S-S 2016 Determination of special impeller diameter for pump as turbine and its effects on turbine performance *J T Chin Soc Agric Mach* **32** 60-6 (In Chinese)

[12] Yang S-S, Kong F-Y and Chen H 2012 Effects of Blade Wrap Angle Influencing a Pump as Turbine *J Journal of Fluids Engineering-Transactions of the ASME* **134**

[13] Yang S-S, Kong F-Y and Qu X-Y 2012 Influence of Blade Number on the Performance and Pressure Pulsations in a Pump Used as a Turbine *J Journal of Fluids Engineering-Transactions of the ASME* **134**

[14] Yang J-H, Xu T and Liu H-R 2014 Numerical Simulation of Whole Flow Passage for a Multi-Stage Hydraulic Turbine with Gas-liquid Two-Phase Media *J Lanzhou Univ Technol* **33** 103-8 (In Chinese)

[15] Yang J-H and Xia S-q 2013 Numerical analysis of hydraulic turbine operating with gas-contained medium *J Lanzhou Univ Technol* **39** 49-54 (In Chinese)

[16] Shi F-X, Yang J-H and Wang X-H 2017 Unsteady analysis of effect on hydraulic turbine under variable content *Journal of Aerospace Power* **32** 2265-72 (In Chinese)

[17] Shi F-X, Yang J-H and Wang X-H 2018 Analysis on Characteristics of Gas-liquid Two-phase Hydraulic Turbine under Variable Working Conditions *Fluid Machinery* **46** 40-5 (In Chinese)

[18] Chen T-J, Guo P-C and Luo Y 2013 Numerical prediction on over-all performance of double suction pump reverse running as turbine *J Drain Irrig Mach Eng* **31** 195-9 (In Chinese)

[19] Zhang K-Y 2017 Numerical Simulation and Experimental Study on the Internal Flow Characteristics of Centrifugal Pump in Gas-Liquid Two-Phase Flow Conditions. In: Jiangsu University, (Zhenjiang: Jiangsu University)