PAPER • OPEN ACCESS

Effect of gas volume fraction on vortex motion in hydraulic turbine

To cite this article: G T Shi *et al* 2018 *IOP Conf. Ser.: Earth Environ. Sci.* **163** 012117

View the article online for updates and enhancements.

IOP e**books**™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

Effect of gas volume fraction on vortex motion in hydraulic turbine

G T Shi[1], Y Liu[1], K Luo[1], X P Jiang[2] and X T Yao[1]

[1] Department of Power Machinery, University of Xihua, Chengdu, China
[2] School of Mechanical Electronic & Information Engineering, China University of Mining &Technology(Beijing), Beijing ,China

shiguangtai_1985@126.com

Abstract. In order to analyze the vortex motion in the flow channel of the hydraulic turbine impeller during the gas volume fraction change. Now, the pump with a specific speed of 55.7 is chosen as hydraulic turbine. On the basis of considering the gas compressibility, to take numerical calculation on the model under different flow rates and different gas volume fraction, to analyze the influence of gas volume fraction on vortex motion law in the impeller flow channel. Findings: When the flow rate is small, the relative velocity distribution in the impeller flow channel is uneven, the velocity field is chaotic, and there are obvious vortices, with the increase of the gas volume fraction, the vortices in the impeller flow channel gradually move to the inlet direction of the blade; With the increase of the flow rate, the flow in the channel of the hydraulic turbine impeller is unstable. Both the pressure surface and the suction surface of the blade appear vortices, the vortex region in the impeller flow channel is enlarged, and all of them are concentrated on the back of the blade. The results provide a theoretical basis for the optimal design of hydraulic turbine structures.

1. Introduction

Hydraulic turbines were the first studied to recover pressure energy, and have a wide range of applications, the hydraulic turbine recovery of liquid energy research on the national economy and social development will have an important role in promoting [1-5]. In engineering practice, when the hydraulic turbine recovery of synthetic ammonia, petrochemical and other industries of high pressure liquid, these high pressure liquids often contain a certain amount of gas [6], the presence of gas can cause instability in the flow channel of the impeller and form a vortex, resulting in increased hydraulic losses, energy recovery efficiency becomes low, stability deteriorated and other issues.

At present, the research on the flow mechanism in the hydraulic turbine mainly focuses on the velocity distribution law, the law of pressure distribution, the static and dynamic interference. In the literature [7], the velocity field and pressure field of pump as turbine are analyzed by numerical simulation and external characteristic test, and the feasibility of pump as turbine is verified. In the literature [8], the flow mechanism inside the hydraulic turbine is analyzed in detail, and get some conclusions. In the literature [9], the numerical simulation method is used to study the external characteristics and the internal flow field of the hydraulic turbine under pure water condition and gas-liquid two-phase condition. The literature [10]

analyzes the hydraulic turbine unsteady pressure pulsation. In the literature [11], the effect of guide vanes on hydraulic turbine performance and pressure pulsation in overcurrent components was studied by adding vanes. In the literature [12-14] the hydraulic loss within the impeller is the main loss inside the hydraulic turbine. It can be seen that the formation and development of vortex in the impeller flow channel is one of the main reasons leading to the increase of hydraulic turbine hydraulic loss and the decrease of hydraulic performance.

It can be seen that the study of the vortex motion law in the hydraulic turbine will help to improve the flow of the hydraulic turbine, reduce the hydraulic loss and improve the hydraulic turbine hydraulic performance. However, the current research on the vortex in the hydraulic turbine is rarely reported in the literature, and it is rarely involved in gas-liquid two-phase condition. Therefore, the numerical simulation method is used to explore the influence of the volume fraction on the vortex motion and distribution in the hydraulic turbine.

2. The main parameters of the hydraulic turbine

In this paper, the single-stage and single-suction pump is selected as the turbine, and the design parameters under pump conditions are as follows: flow $Q_p=90m^3/h$, head $H_p=93.6m$, blade number $Z=6$，speed $n=2900r/min$, $n_{sp}=55.7$; The basic parameters under Turbine conditions: flow $Q_t=128.34m^3/h$，head $H_t=131.44m$, $n_{st}=50.9$.

3. Numerical calculation method

3.1 Compute domain selection and meshing

The hydraulic model of the internal flow area of the hydraulic turbine is established by Pro / e software. In order to improve the calculation precision, the inlet and outlet pipe sections are extended to twice the pipe diameter. Then, the model is divided into unstructured grid, and the grid is checked for irrelevant. When the total number of grids is more than 2.4 million, the range of the efficiency is less than 0.45%, so the total number of grids is larger than 2.4 million is more appropriate. In this paper, the total number of grids is 2400629, and the number of grids, volute, outlet extension, impeller and impeller inlet extension is 546832, 618030, 482319 and 753448 respectively.

3.2 Selection of multiphase flow model

There are two numerical methods to deal with multiphase flow: Euler-Lagrangian method, Euler-Euler method. There are three Euler-Euler multiphase models in FLUENT: Mixture, VOF, and Euler model. In this paper, the gas is a continuous compressible ideal gas, according to the principle of multi-phase flow and the stability of the solution, economic and other factors, Mixture model more in line with the calculation requirements. The Mixture model uses a single fluid method that allows the phase to move at different speeds, allowing the phases to cross each other.

The continuous equation of the Mixture model

$$\frac{\partial}{\partial t}\left(\rho_m\right)+\nabla\square\left(\rho_m\,\vec{\upsilon}_m\right)=0$$

(1)

Here υ_m is the mass average velocity: $\vec{\upsilon}_m=\dfrac{\sum_{k=1}^{n}\alpha_k\rho_k\vec{\upsilon}_k}{\rho_m}$, ρ_m is the mixture density，α_k is the Kth phase volume fraction.

The momentum equation of the Mixture model

$$\frac{\partial}{\partial t}\left(\rho_m \vec{\upsilon}_m\right)+\nabla\square\left(\rho_m \vec{\upsilon}_m \vec{\upsilon}_m\right)=\nabla\square\left[\mu_m\left(\nabla\vec{\upsilon}_m+\nabla\vec{\upsilon}_m^T\right)\right]-\nabla p+\rho_m \vec{g}+\vec{F}+\nabla\square\left(\sum_{k=1}^{n}\alpha_k \rho_k \vec{\upsilon}_{dr,k}\vec{\upsilon}_{dr,k}\right) \tag{2}$$

$\vec{F}$ is the volume force, u_m Mixture viscosity coefficient, n is the number of phases, $\vec{\upsilon}$dr, k is the drift velocity of the second phase k.

The energy equation of the Mixture model

$$\frac{\partial}{\partial t}\sum_{k=1}^{n}\left(\alpha_k \rho_k E_k\right)+\nabla\cdot\sum_{k=1}^{n}\left(\alpha_k \vec{\upsilon}_k\left(\rho_k E_k+p\right)\right)=\nabla\cdot\left(k_{eff}\nabla T\right)+S_E \tag{3}$$

k_{eff} is the effective thermal conductivity, S_e contains all volumes of heat, EK=hk- $\frac{p}{\rho_k}$ +$\frac{\upsilon_k^2}{2}$ is for compressible phase，for incompressible phase EK=hk，hk is the apparent enthalpy of the Kth phase.

3.3. Parameter settings

In this paper, the working medium of the model is the water and the ideal gas, and make the following assumptions:
- The water is the main phase, the gas is the secondary phase;
- The water is continuously incompressible and the gas is continuously compressible;
- There is no phase change and mass transfer between gas and liquid phases.

Using a pressure-based solver, choose steady state method to solve; Using SIMPLEC algorithm, the turbulence model is chosen as the standard k-ε turbulence model, set the convergence accuracy to10^{-4}; The non - slip boundary condition is adopted at the solid wall, and the standard wall function is used near the wall; The boundary condition of the hydraulic turbine inlet is set to the mass flow inlet and the outlet boundary condition is set as the pressure outlet; the mass flow rate of the clear water and the ideal gas is adjusted by changing the inlet flow rate and the gas volume fraction.

4. Calculation result analysis

4.1. Analysis of flow field in Hydraulic Turbine with different gas volume fractions

4.1.1. Streamline distribution in the PAT under small flow conditions. In order to facilitate the analysis, the flow path in the impeller is divided into I, II, III, IV, V, VI, and figure. 1 shows the distribution of the streamline in the hydraulic turbine overcurrent component with different gas volume fraction and small flow rate. It can be seen from the figure that the relative velocity distribution in the impeller flow channel is not uniform and the velocity field is chaotic. There are four obvious vortices and each is different, mainly due to the asymmetry of the volute structure. The direction of rotation of the vortex is opposite to that of the impeller and is clockwise, and both are located in the blade pressure surface. With the increase of the gas volume fraction in the small flow condition, the vortex is formed in the flow path II of the impeller and gradually becomes larger. Under the pure water condition, there are two vortices in the flow path III in the same direction of rotation. As the gas volume fraction increases, the two vortices gradually become a larger vortex. It can be seen from the figure that as the gas volume fraction increases, the vortex in the impeller gradually moving toward the blade inlet direction, the vortex area gradually increases.
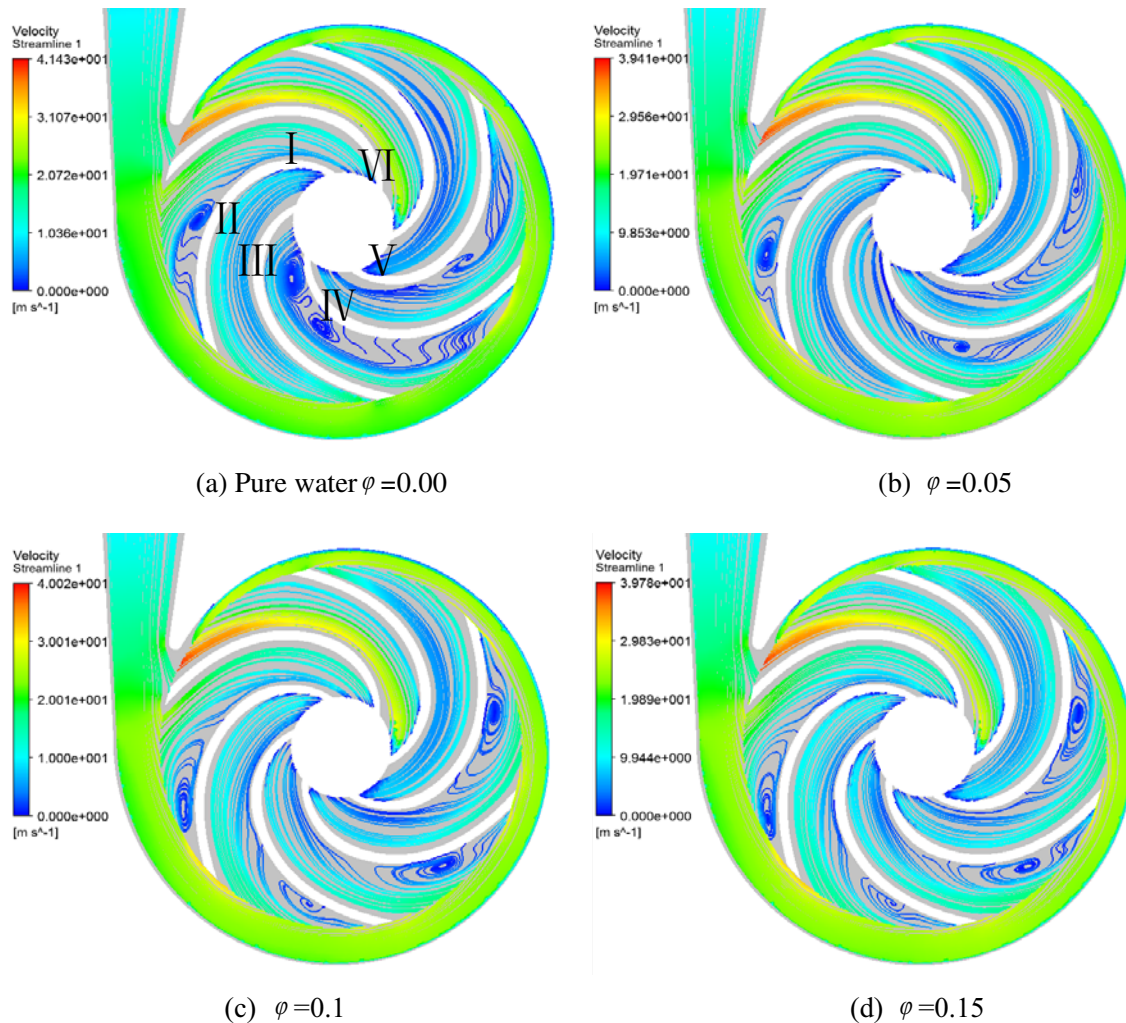
(a) Pure water φ =0.00

(b) φ =0.05

(c) φ =0.1

(d) φ =0.15

Figure 1. Streamline distribution in the PAT with different gas volume fraction under small flow conditions

4.1.2. Streamline distribution in the PAT under optimum conditions. The figure 2 shows the distribution of the streamline in the hydraulic turbine overcurrent component with different gas volume fraction and design condition. It can be seen from the figure that under the design condition, the vortex area in the impeller flow channel is reduced and the flow of the fluid is improved. The rotation direction of the vortex is clockwise and all appears on the blade pressure surface. The streamline is relatively uniform at the outlet of the blade; With the gas volume fraction increased, the vortex gradually moved toward the blade inlet direction, and finally disappeared, and flow separation are formed at the inlet.
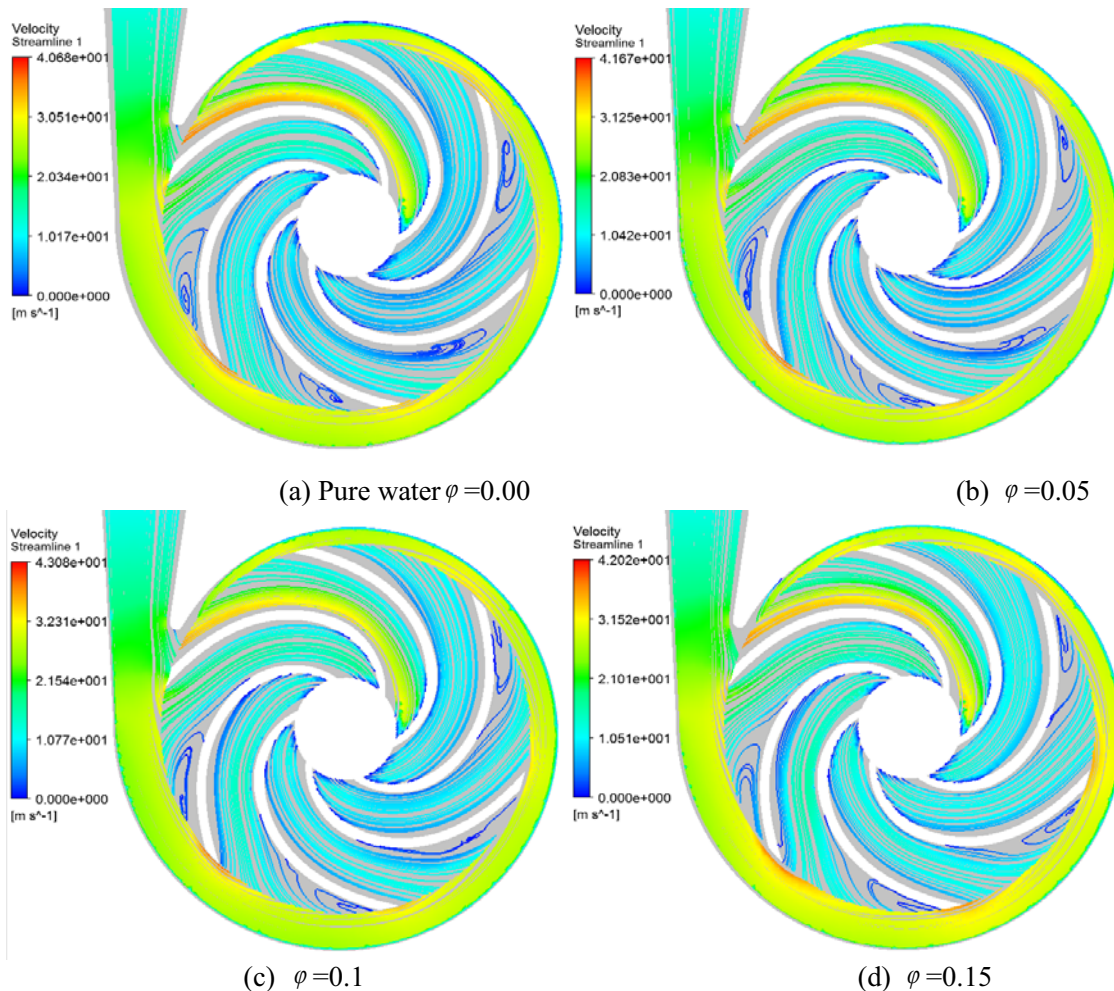
(a) Pure water φ =0.00   (b) φ =0.05

(c) φ =0.1   (d) φ =0.15

Figure 2. Streamline distribution in the PAT with different gas volume fraction under optimum conditions

4.1.3. Streamline distribution in the PAT under large flow conditions. The figure3 shows the relative velocity distribution in the impeller. The figure 4 shows the distribution of the streamline in the hydraulic turbine overcurrent component with different gas volume fraction and large flow condition. It can be seen from figure 4 : Under this condition, the flow in the flow channel of the hydraulic turbine impeller is the most unstable, the speed field is the most chaotic, the blade pressure surface and suction surface are vortex, especially the back of the blade, and there are a large number of vortex, as shown in figure. 3, the blade pressure surface and the back of the vortex rotate in the opposite direction, and the vortex on the back of the blade is counter clockwise. With the increase of the gas volume fraction, the vortex is gradually formed on the back of the I, II, III, IV channel, the blade pressure surface produces a backflow to the blade inlet, and the vortex area in the impeller channel is enlarged and concentrated on the back of the blade.
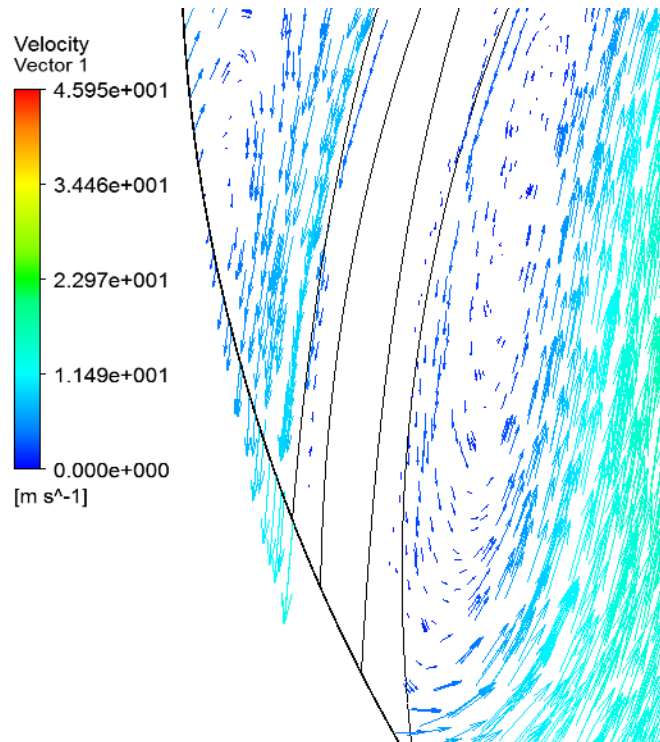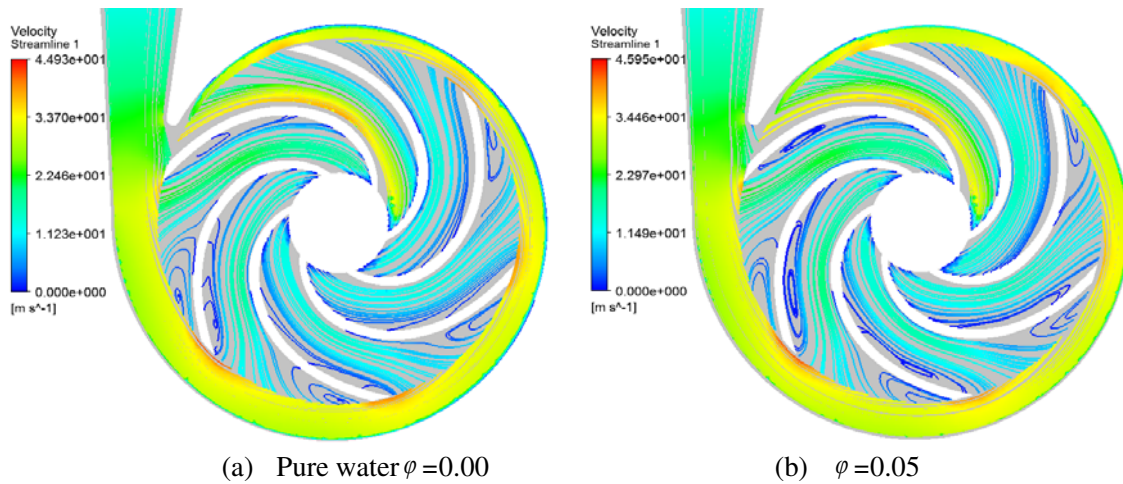
Figure 3. Impeller relative velocity distribution

(a)   Pure water φ =0.00   (b) φ =0.05

<table>
<tr><td>(c) φ =0.1</td><td>(d) φ =0.15</td></tr>
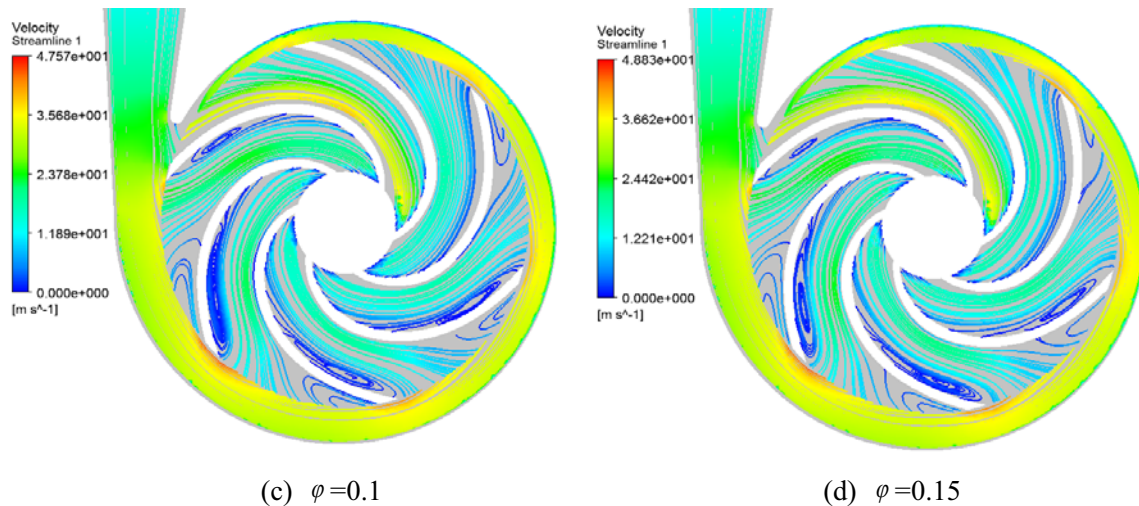</table>

Figure 4. Streamline distribution in the PAT with different gas volume fraction under large flow conditions

4.2. Analysis of flow field in Hydraulic Turbine with different flow

The figure 5 shows the distribution of the streamline in the hydraulic turbine overcurrent component with different flow and gas volume fraction of 15%(due to the larger the gas volume fraction, the more unstable the flow within the impeller, so selected gas volume fraction of 15% of the conditions to study). It can be seen from the figure: As the flow increases from $1.2Q_p$ to the design condition $1.4Q_p$ and then increases to $1.6Q_p$, the velocity field in the hydraulic turbine impeller flow channel is improved and gradually becomes chaotic. The vortex area in the impeller flow channel decreases first and then increases, and the number of vortices decreases first and then increases. The vortex of the blade pressure surface disappears and the vortex on the back of the blade gradually forms, the hydraulic loss of the turbine is reduced first and then increased, which causes the hydraulic efficiency to increase first and then decrease, which is consistent with the known hydraulic turbine efficiency curve.
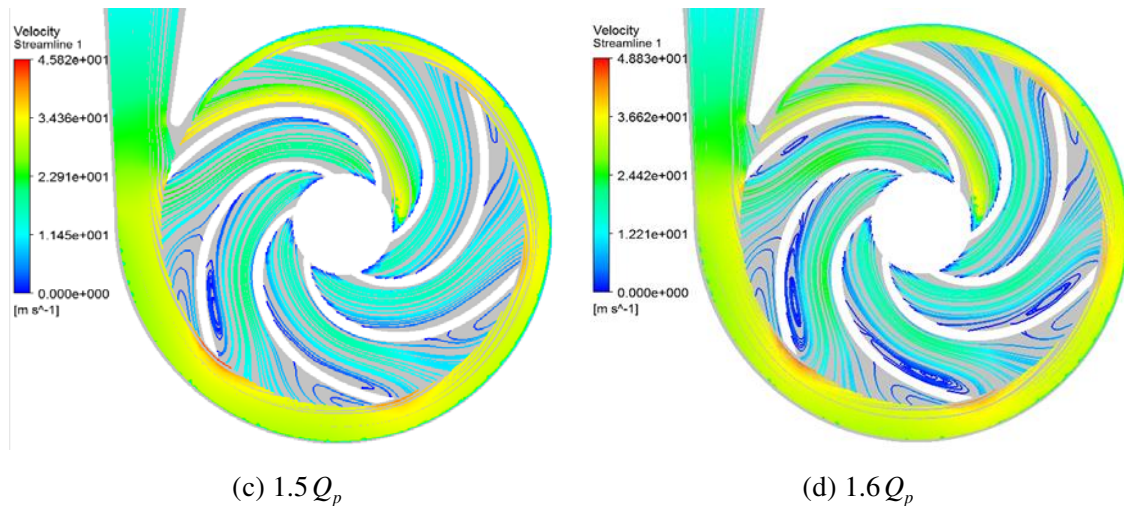

<table>
<tr><td>(a) $1.2Q_p$</td><td>(b) $1.4Q_p$</td></tr>
</table>

(c) $1.5 Q_p$ (d) $1.6 Q_p$
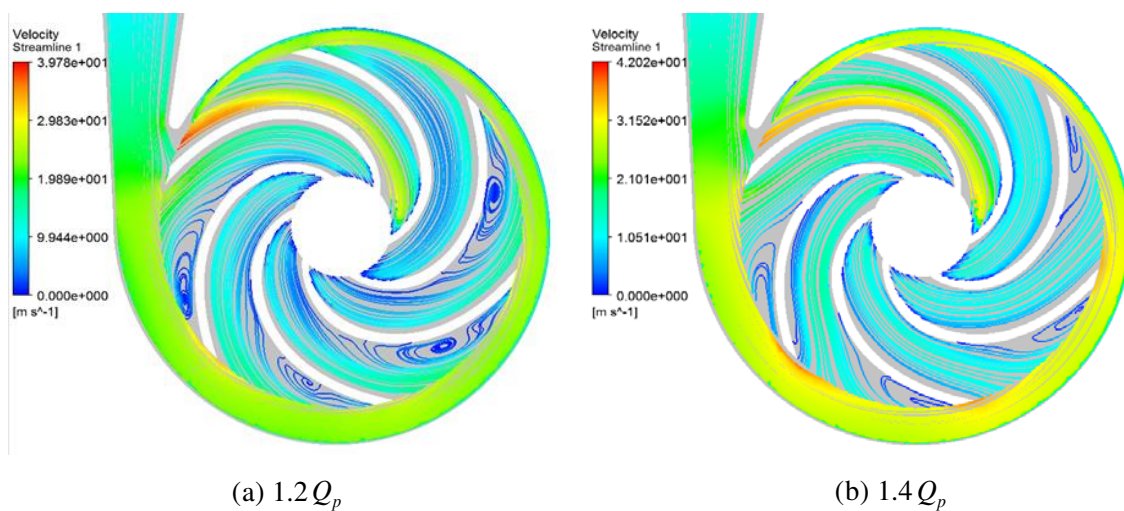
Figure 5. Streamline distribution in the PAT with different flow under gas volume fraction of 15%

5. Conclusions

- When the flow rate is small, the relative velocity distribution in the impeller flow channel is uneven, the velocity field is chaotic, and there are obvious vortices, and they are all on the blade pressure surface. With the increase of the gas volume fraction, the vortices in the impeller flow channel gradually move to the inlet direction of the blade, and the vortex region in the impeller flow channel is enlarged.

- Under the design condition, the vortex region in the impeller flow channel of hydraulic turbine is reduced, the flow condition of the fluid has been improved, the rotation direction of the vortex is clockwise and all appears on the blade pressure surface, and the streamline at the blade outlet is relatively uniform; With the increase of gas volume fraction, the vortex gradually moved closer to the blade inlet and finally disappeared, and flow separation are formed at the inlet.

- Under the condition of large flow rate, the flow in the channel of the hydraulic turbine impeller is most unstable, and the velocity field is in the most chaotic, both the pressure surface and the suction surface of the blade appear vortices. In particular, a large number of vortices appear on the back of the blade. The blade pressure surface produces a back-flow near the blade inlet, the vortex region in the impeller flow channel is enlarged, and they are all concentrated on the back of the blade.

References

[1]   Yang Junhu, Zhang Xuening，Wang Xiaohui，et al. Overview of research on energy recovery hydraulic turbine [J].Fluid Machinery, 2011,39(6):29—33.

[2]   Carravetta A, Antipodi L, Golia U, et al. Energy Saving in a Water Supply Network by Coupling a

Pump and a Pump As Turbine (PAT) in a Turbopump[J]. Water, 2017, 9(1): 62.

[3] Date A, Akbarzadeh A. Design and cost analysis of low head simple reaction hydro turbine for remote area power supply[J]. *Renewable Energy*, 2009, 34(2): 409-415.

[4] De Marchis M, Milici B, Volpe R, et al. Energy Saving in Water Distribution Network through Pump as Turbine Generators: Economic and Environmental Analysis[J]. *Energies*, 2016, 9(11): 877.

[5] Gong R Z, Wang H J, Chen L X, et al. Application of entropy production theory to hydro-turbine hydraulic analysis[J]. *Science China Technological Sciences*, 2013, 56(7): 1636-1643.

[6] WILLIAMS A. The turbine performance of centrifugal pumps: a comparison of prediction methods [J]. *Proc Instn Mech Engrs*, 1994，208(A1):59-66.

[7] Yang Sunsheng, Kong Fanyu, Su Xianghui,et al. Numerical simulation and performance experiment on pump and pump as turbine [J]. *Journal of Xi'an Jiaotong University*, 2012,46 (3):37—41.

[8] Shi Guangtai, Yang Junhu. Calculation method of slip factor for impeller outlet of centrifugal pump as hydraulic turbine [J]. *Transactions of the Chinese Society of Agricultural Engineering*, 2014,30(13):68—77.

[9] Yang Junhu. XuTing, LiuHaoran. Numerical Simulation of Whole Flow Passage for a Multi-stage HydraulicTurbine with Gas-liquid Two-Phase Media[J]. *Journal of Xihua University · Natural Science*, 2014,(05):103-108.

[10] Yang Sunsheng, Kong Fanyu, Zhang Xinpeng, et al. Simulation and analysis of unsteady pressure fluctuation in hydraulic turbine [J]. *Transactions of the Chinese Society of Agricultural Engineering*, 2012,28(7):67—72.

[11] Shi Guangtai, Liu Xiaobing, et al. Pressure pulsation characteristics in pump as hydraulic turbine with guide vanes [J].*Journal of drainage and irrigation machinery engineering (JDIME)*,2017,35(1):6—12,55.

[12] Yang Sunsheng, Zhang Hui, Kong Fanyu, et al. Analysis on blades curved shape of centrifugal pump as turbine[J]. *Journal of Shanghai Jiaotong University*, 2016,(09):1467-1472+1479.

[13] Wang Tao, Kong Fanyu, Yang Sunsheng, et al. Effects of blade angle distributions on performance of pump as turbine[J].Transactions of the Chinese Society for Agricultural Machinery, 2015,10:75-80.

[14] Yang Sunsheng, Li Qiang,Huang Zhipan, et al. Research on Different Specific Speed Pumps Used as Turbines[J]. *Journal of Shanghai Jiaotong University*, 2013,(03):69-72.