Impeller entrance pre whirl characteristics research

W WU[1], Y Wang[1], Y W Han[1]

[1] National Research Center of Pumps, Jiangsu University, Zhenjiang, Jiangsu 212013, China

Email: Wuwen1122@126.com

Abstract. In order to study the effect of inlet port on the pump performance, the impeller inlet part, should be analyzed for impeller is able to extend the function of water flow to the front of the impeller for a long distance. Impeller flow of pre swirl flow is due to selection of least resistance into the impeller, but the pre swirl in the flow direction according to the impeller blade entrance angle, and the circumferential velocity of flow. The study found that lies in the external characteristic of the pump will be fell when the off-design, but in the case of large flow impeller and impeller in the direction of the front entrance fluid pre whirl steering is on the contrary, when this with little traffic is quite different .this article will study the occurrence, development, and the mechanism of the influence of flow field.

1. Introduction

The earliest research on pre-swirl pump inlet can be traced back to 1909, Steele Watt found the presence of a centrifugal pump suction line pre rotator A.J.Stepanff proposed: At a given speed, the centrifugal liquid can flow along at a certain axially into the working wheel, this flow is called nominal flow. When the flow is less than the rated flow, the direction of the liquid is the same as the rotation direction of the working wheel. When the flow is greater than the nominal flow, the direction of the liquid is opposite to the rotation direction of the working wheel. Lu Linguang, Zhu Honggeng, Chen Wei et al. A new technique for measuring the inlet swirl intensity of the pump inlet is proposed and other research on the inlet flow of the axial flow pump and the optimization of the inlet[1-2];Zhang Yongxue, Song Pengfei, et al. Study the influence of inlet pre-swirl on cavitation. Axial flow pump is a low lift pump[3], with the characteristics of large flow and low head, widely used in the field of farmland drainage, irrigation and large water conservancy projects and other fields. Pump inlet flow is very complex[4-14], the need for in-depth study of the problem is also a lot of. Although there are some studies on the pre-swirl and swirling flow of the inlet fluid of the pump, it is not yet seen in China.

Entrance pre-swirl will lead to the water pump is not working in high efficient area, more serious when can cause the pump cavitation damage. But not with a pre-swirl, there must be bad, sometimes in order to meet the needs of the pump head, is the need to import at some pre-swirl, for example, some in front of the pump to add a wheel in order to induce the want to generate their own pre-swirl. This paper will reveal the generation of the pre-swirl and its different forms in different conditions.

2.Numerical Methods

2.1.Turbulence Model

Similar SST k-ω model and standard k-ω model, but with the following improvements: deformation growth SST k-ω model and k-e model together in the mixing function and dual model. The hybrid function is designed for the near wall region. This region is effective for the standard k-ω model and the free surface, which is effective for the deformation of the k-e modeless k -ω merged model is derived from the cross diffusion equation of ω.Turbulent viscosity took into account the turbulent shear stress of the wave. Model constants of different These improvements make SST k -ω model than the standard k -ω in the field of flow in a wide range of omega has a higher precision and credibility.

Control equation:

Continuity equation: $\dfrac{\partial \rho}{\partial t}+\nabla.(\rho U)=0$ (1)

Momentum equation: $\dfrac{\partial \rho U}{\partial t}+\nabla\cdot(\rho U\otimes U)-\nabla\cdot(\mu_{eff}\nabla U)=\nabla\cdot p'+\nabla\cdot(\mu_{eff}\nabla U)^{T}+B$ (2)

B for volume in the force; μ_{eff} For the effective viscosity; p' For the correct pressure; μ_{t} For the turbulent viscosity; Its expression is：

$$\mu_{eff}=\mu+\mu_{t} \tag{3}$$

$$p'=p+\frac{2}{3}\rho k \tag{4}$$

$$\mu_{t}=C_{\mu}\rho\frac{k^{2}}{\varepsilon} \tag{5}$$

Model for both sides, for the turbulent kinetic energy and turbulent kinetic energy dissipation equation

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho Uk)=\nabla\cdot[(\mu+\frac{\mu_{t}}{\sigma_{k}})\nabla k]+P_{k}-\rho\varepsilon \tag{6}$$

2.2.Axial flow pump model, meshing

This article select axial flow pump model are studied, Design parameters H=3.2m，Q=392/h，n=1450r/menthe fluid domain as shown in the figure below:

1-Entrance section
2-The rotor section
3-Guide vane section
4-Export section
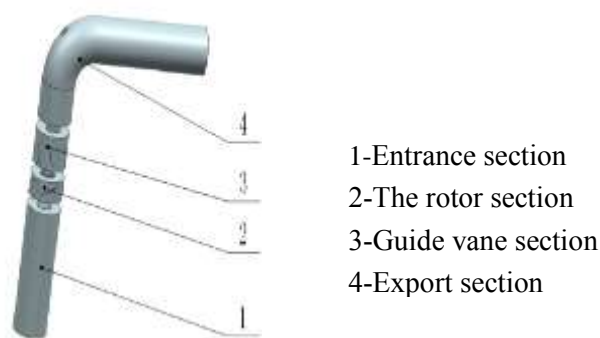
Figure 1. Axial flow pump of the fluid domain   **Figure 2.** Mesh

The imported water body diameter, water impeller diameter, the outside diameter of the guide vane water outlet water and the outside diameter is 200mm.In this image, the mesh division of the three dimensional water body of the axial flow pump, the impeller and the guide vanes are used to divide the non-structure tetrahedral mesh. According to the sort of picture, the grid number is shown in the following figure. The import and export have done a proper extension. In the grid independent inspection, the change of the local changes in the velocity gradient is greater.

Entrance section	The rotor section	Guide vane section	Export section
480758	1204493	1107034	991996

Figure 3. Each part of the fluid domain mesh number

*2.3.Boundary **conditions***

The flow field is solved by the governing equations of the rotating coordinate system. The inlet boundary conditions is mass flow, the outlet boundary condition is a standard atmospheric pressure. No-slip wall condition not consider the wall roughness conditions and turbulence model chosen   SST k-ω turbulence model.

The finite volume method is used to control the governing equations. In which the velocity and the turbulent kinetic energy viscosity coefficient are divided into two order central difference schemes. The convergence criteria for the calculation are less than the residual value10^{-4}.

3.The numerical simulation analysis and theoretical analysis

Axial pump suction chamber, also known as Inlet duct, it refers to direct toward the impeller inlet pump pipes. The smooth flow, flow rate and pressure distribution at the impeller inlet, have a great impact on the performance of the pump. Therefore, the design requirements of the suction chamber in the water loss of the minimum conditions of the liquid into the impeller inlet, and ensure that the flow of liquid into the impeller speed distribution. The experimental results show that the hydraulic loss of the suction chamber of axial flow pump is small. But although the water absorption chamber is small, it has a direct influence on the cavitation and safe operation of the pump. Because, if the flow velocity distribution is not uniform, it will greatly reduce the efficiency of the impeller and worsen the cavitation performance of the impeller. In addition, liquid flow in the suction chamber should not

produce a vortex, if there is a vortex generated, will cause the impeller inlet flow disorders, resulting in reduced efficiency, resulting in reduced efficiency of the pump, generating vibration. In order to avoid the impact of the pump, the impeller before the entrance to produce a pre rotation. In the inlet of the impeller, the flow of the vortex in the shape of the gradient, which eased the impact. When the liquid flows into the impeller at a large angle, the nature of the hydraulic loss at the entrance of the impeller is caused by the sudden expansion of the liquid or the sudden expansion of the liquid.

In the design of the pump, it is always assumed that the axial distribution of the inlet flow of the impeller is sufficiently uniform. In the derivation of the basic equation of vane pump, it is assumed that the flow is stable and the flow pattern is uniform. If the pump inlet water flow irrational, the absolute velocity of C in the circumferential direction of the Cu1 is zero, and the Euler equation of the vane pump is:

$$H_t = \frac{U_2 C_{U2} - U_1 C_{U1}}{g} = \frac{U_2 C_{U2}}{g} \tag{7}$$

The above equation: H_t-Pump design head; C_{U1}、C_{U2}-Respectively for the impeller import and export of absolute velocity in the circumferential direction of the projection; U_1、U_2-Impeller respectively in the import and export of circumferential velocity of water flow; g-Acceleration of gravity.

Liquid flow to the impeller, and after the outflow of the impeller, is due to the energy gradient to lower than the zero flow of numerical value. Through the impeller, the energy gradient so that the liquid can be gradually increased in the head. According to the principle of energy gradient, liquid flow to choose the path of least resistance impeller and after the outflow of the impeller. The liquid is obtained by the pre rotation so as to enter the impeller flow channel with the smallest obstacle. The inlet angle of the vane of the impeller is related to the flow of the impeller and the circumferential speed of the impeller. All of these three quantities determines the inlet velocity triangles. Obviously, if the liquid is close to the angle of the blade angle, the flow of the liquid is the least resistance. Under the given working wheel speed, only a flow, the liquid can along the axial plane into the wheel. See figure below, when the flow rate of the pump is much smaller than the nominal flow, the direction of the liquid is the same as that of the working wheel, so it may be possible to enter the work in the direction of the β. But when the flow is greater than the nominal flow rate, the pre rotation must be in the opposite direction, in order to meet the requirements of "minimum resistance". Within the actual pump, liquid flow in accordance with the above flow spectrum, just because of the influence of short suction tube and the suction line structure and slightly change. It should be pointed out that, at the entrance of wheel suck liquid prewhirl is not because of the influence of the wheel, it is clear that the wheel can't push liquid to its own direction of rotation in the opposite direction, and when the flow more than nominal flow, often see this kind of situation.
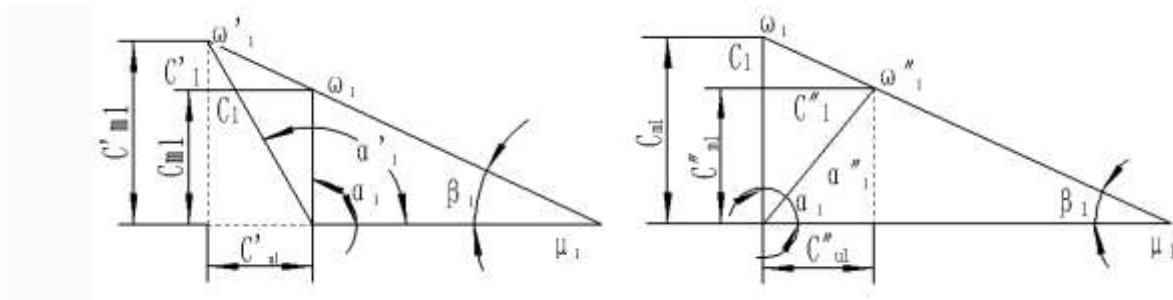
Figure 4. Non-design conditions of the pump inlet velocity triangle

Take two of axial flow pump inlet cross section, respectively to observe two cross section component of circular velocity, respectively.
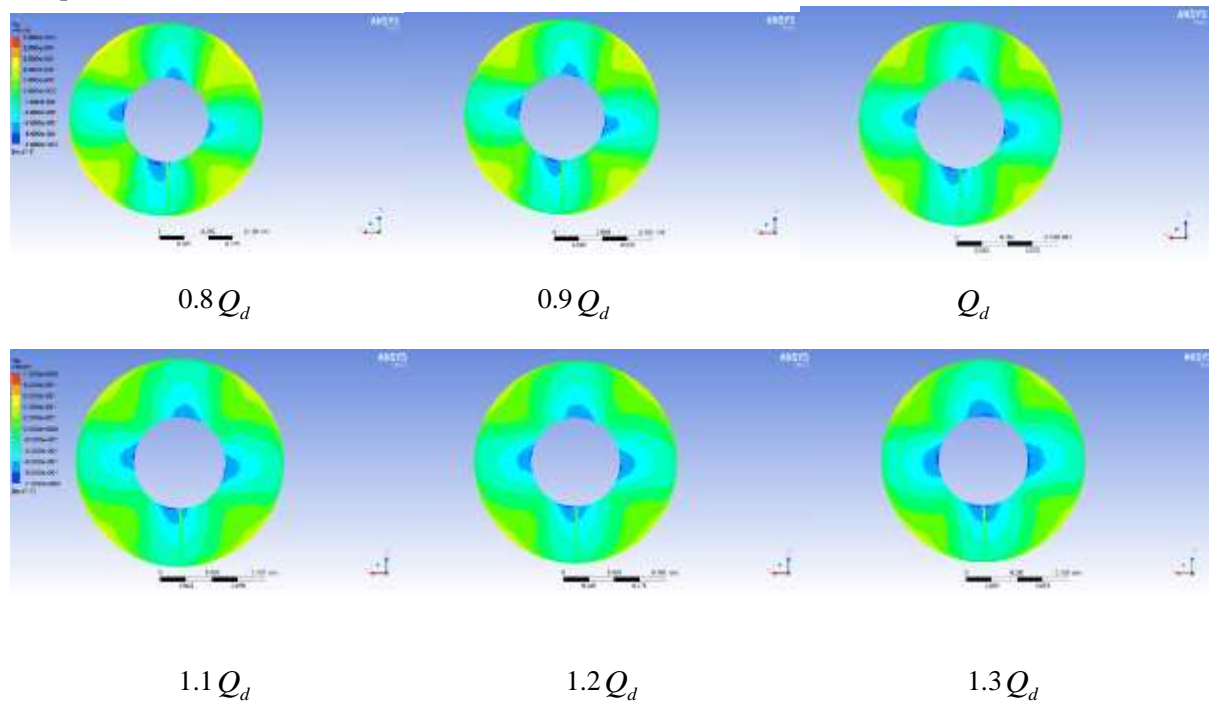

Figure 5. Left for the cross section of two positions

Below for the axial flow pump under different conditions, the entry interface 1 absolute velocity component in the circumferential direction:

$0.8Q_d$	$0.9Q_d$	$1Q_d$

$1.1Q_d$	$1.2Q_d$	$1.3Q_d$
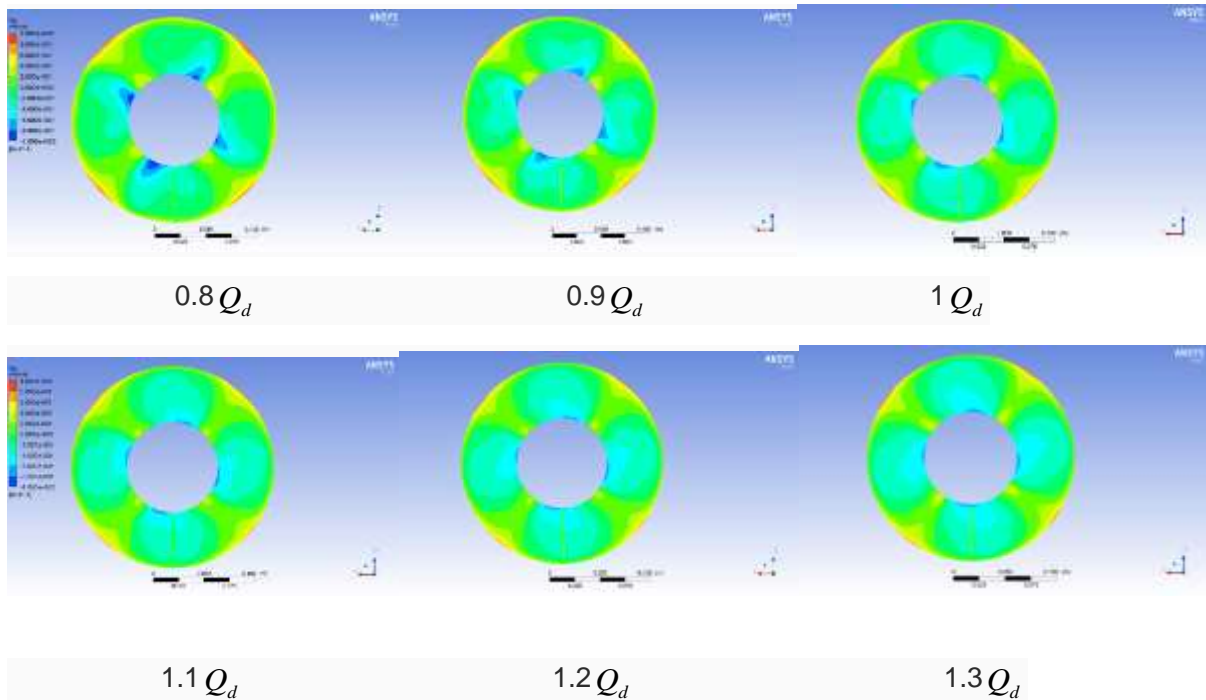
Figure 6.under different conditions, the entry interface 1 absolute velocity component in the circumferential direction；

Below for the axial flow pump under different conditions, the entry interface 2 absolute velocity component in the circumferential direction:

$0.8\,Q_d$ $0.9\,Q_d$ Q_d

$1.1\,Q_d$ $1.2\,Q_d$ $1.3\,Q_d$

Figure 7. under different conditions, the entry interface 2 absolute velocity component in the circumferential direction；

From the above we can see from the above, the position of the 1 section of the radius of the impeller wheel position is bigger, relative to the area of cross flow 2 reduced. Axial surface speed increase, according to the velocity triangle, it can be seen in small flow section of Cu value relative to the section 2 Cu value is small, and the big traffic 1 section of Cu is bigger.

People often put the work wheel at the entrance and exit at the loss of the impact. However, this term is not appropriate, because the term "impact" or "crash" in mechanics does not necessarily mean a loss. From the fluid dynamics, it is known that if a collision occurs in the liquid flow direction, most of the energy is recycled (impulse).In the pump, in order to avoid the impact, the liquid flow in the working wheel before the entrance to produce a pre rotation. In the working wheel exit, in the case of a velocity gradient, which will ease the impact. When the liquid flows into the working wheel with a larger impact angle, the nature of the hydraulic loss at the entrance of the working wheel is the loss caused by sudden expansion or sudden diffusion of the liquid after the desulfurization. Wheel at the outlet of the loss is mainly due to the guide leaves lower mean velocity and wheel at the outlet of the high velocity induced in the liquid jet friction loss. It should be noted that, even at maximum efficiency conditions and guide leaves the average speed is still than the circumference of the wheel at the outlet of the absolute velocity points much lower rate. As this is the best working condition, it can not be used to change the area of the guide vane to improve the situation. In addition, the impact damage at the entrance of the guide vane, these losses and the loss of the working wheel inlet, which is

the same as the loss of diffusion.

If it is assumed, wheel structure can in flow is rated flow (this time without impact), guarantee in the flow direction and the blade inlet angle, outlet angle consistent, so in this two there is no additional loss. When the flow rate is greater than or less than the nominal flow rate, flow velocity will suddenly change direction and magnitude. Through the above analysis, the axial flow pump is a large flow and the flow of small flow, the entrance and exit of its working wheel will generally have a diffusion loss.
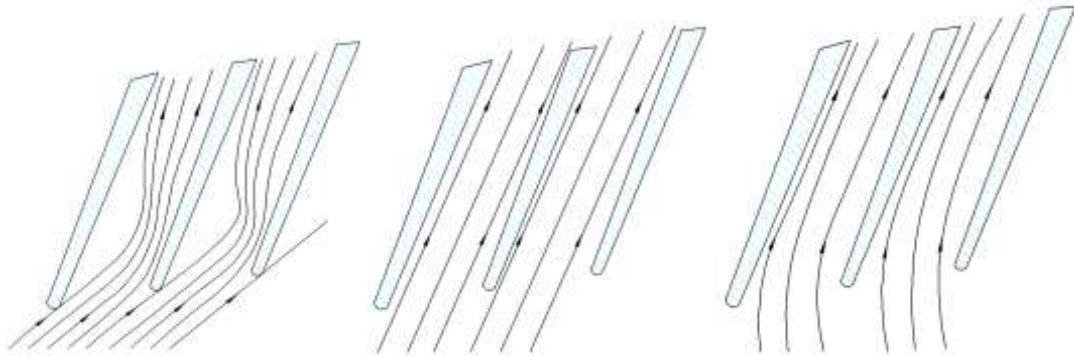

Figure 8. Inlet flow conditions in different conditions

The axial flow pump can reduce the separation of the boundary layer in the large flow rate, and reduce the loss caused by boundary layer separation. The following axial flow pump is given under different operating conditions, the impeller under the different proportions of flow distribution.
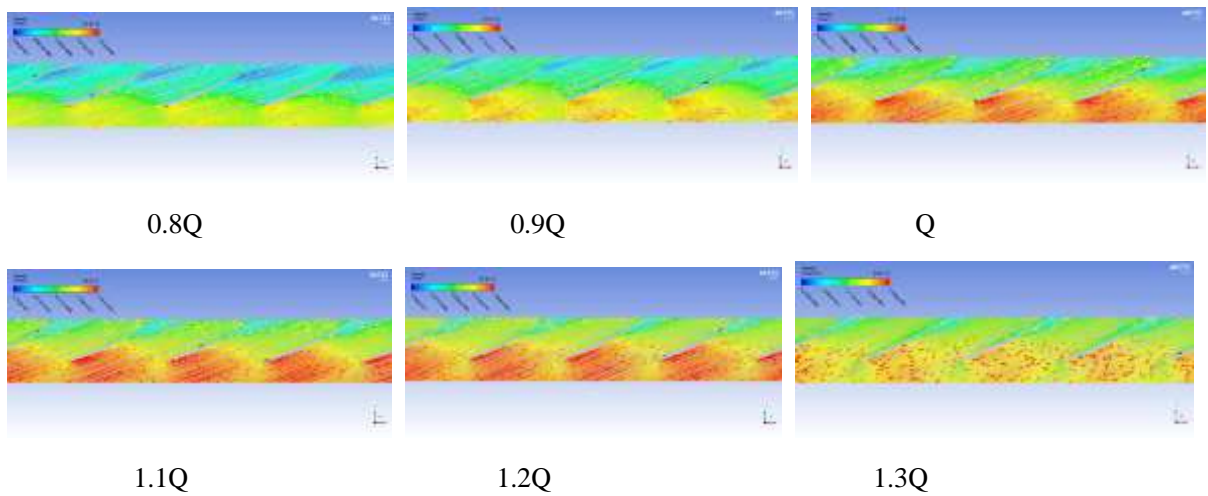

| 0.8Q | 0.9Q | Q |

| 1.1Q | 1.2Q | 1.3Q |
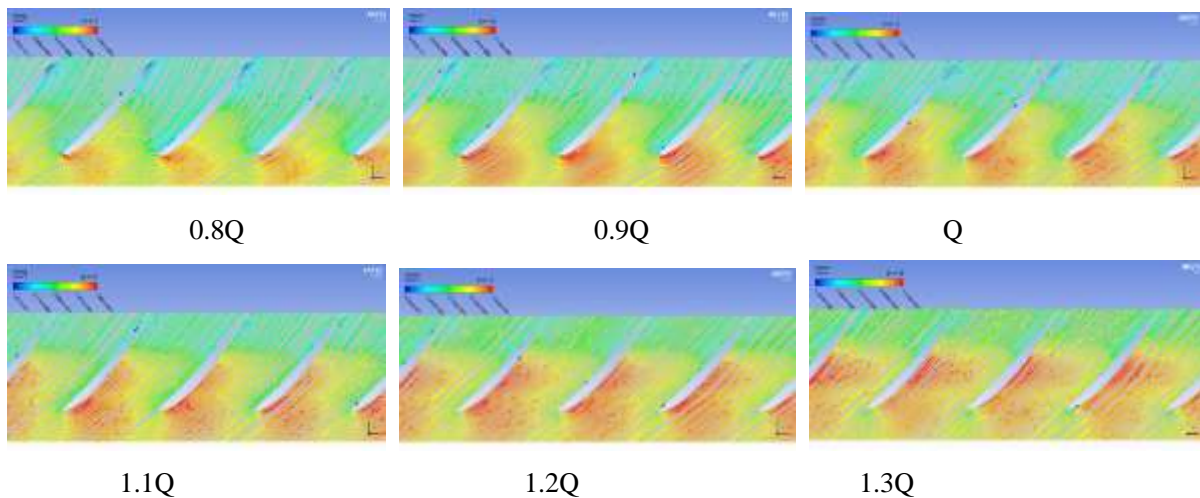
Figure 9. $(r_i - r_h)/(r - r_h) = 0.1$ The radial size ratio of 0.1 on the cylindrical surface of streamline

| 0.8Q | 0.9Q | Q |

| 1.1Q | 1.2Q | 1.3Q |
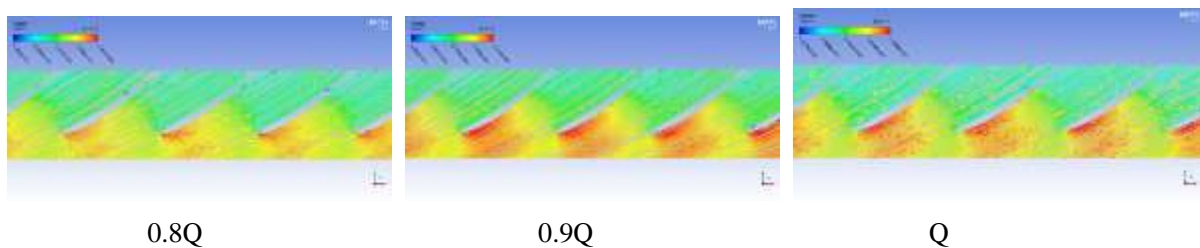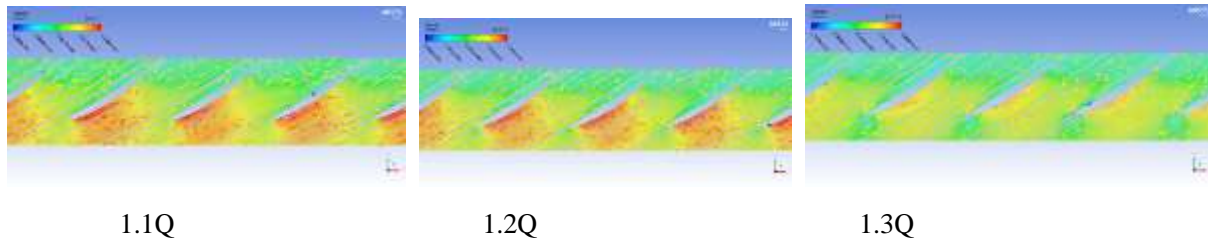
Figure 10. $(r_i - r_h)/(r - r_h) = 0.5$ The radial size ratio of 0.5 on the cylindrical surface of streamline

| 0.8Q | 0.9Q | Q |

| 1.1Q | 1.2Q | 1.3Q |

Figure 11. $(r_i - r_h)/(r - r_h) = 0.9$ The radial size ratio of 0.9 on the cylindrical surface of streamline

r_i – The radius of the cross - section

r – Working wheel radius

r_h – Wheel hub in radius

When flow is greater than or less than the design flow, the direction of the flow velocity will change, and this can be expressed in the type change caused by the loss.

At the entrance of the loss： $h_{s1} = K_1 \dfrac{\Delta C_{u1}^2}{2g}$ （7）

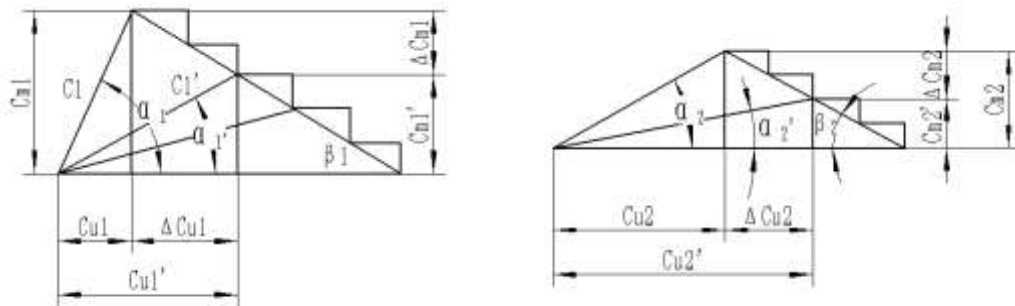The exit of the loss： $h_{s2} = K_2 \dfrac{\Delta C_{u2}^2}{2g}$ （8）

Figure 12.The speed of the pump triangle

ΔC_{u1} and ΔC_{u2} are negative values when the flow is greater than the design flow. In the following figure, when the flow rate is designed, the working speed of the working wheel is C_{m1}, And the absolute velocity and the circumference velocity of the liquid flow is the angle of the α.When the flow is reduced （$C'_{m1} < C_{m1}$）, the liquid flow enters the blade angle of the blade, and the circumferential velocity component is C'_{u1} and $\Delta C_{u1} = C'_{u1} - C_{u1}$. Similarly, the flow Q_s, wheel export of axial plane speed C_{m2}, and absolutely velocity on the circumferential is C_{u2}.When the flow rate is reduced, the circumferential velocity increases to C'_{u2}, and the circumferential velocity increment is the difference between the two values of the appeal, that is $\Delta C'_{u2} = C'_{u2} - C_{u2}$.When the flow rate is greater than the Q_s, ΔC_{u2} and ΔC_{u1} are all negative values. It should be pointed out that in the next two graphs, if the C_{m1} increment of the axial velocity and the flow rate is equal to that of the ΔC_{u1}.Similarly, with an increment corresponding to the C_{m2}, ΔC_{u2} increased an equal value.

When the flow is greater than or less than Q_s, losses have increased, and are proportional to the square of the flow. In this way, can put the above two losses a formula: the formula of synthesis, this is quadratic parabolic equations.

4. Conclusions

4.1. In heavy traffic and small flow rate, according to the velocity triangle deduction, entrance prewhirl direction is different. The direction of the heavy traffic, prewhirl general contrary to work wheel steering. Prove the import prewhirl is not caused by fluid viscosity

4.2. Reasonable control entry conditions to reduce inlet diffusion loss, improve the efficiency of the

pump, provide reference for the design of the pump.

4.3. Import flow caused by flow separation of streamline on the blade surface, and impact on the rotating stall in the pump is critical, this article provides for improving the inlet flow to pump provide reference to some extent avoid rotating stall.

References

[1]   Lu L G, Zhu H G et al. 2003 A new technique for measuring the inlet swirl intensity of the pump inlet [J]. *Journal of Irrigation and Drainage*, **22**(3):57~60

[2]   Lu L G, Chen W et al. 2012 The influence of axial flow pump operating condition on the inlet of the impeller chamber [J]. *Journal of hydroelectric engineering*, **33**(1):214~219

[3]   Zhang Y X, Song T F, Xu C et al. 2014 Experimental and numerical simulation of the influence of the pre swirl adjustment on the cavitation of centrifugal pump [J], *Agricultural Machinery Science*, **45**(9):131~137

[4]   Stepanff A J 1980 *Centrifugal and Axial Flow Pumps* [M]. Beijing: China Machine Press, 1980:41~42. (in Chinese)

[5]   Sun J Z, Liu D K, Ji S L 1984 Influence of flow pattern at the impeller chamber inlet on axial-flow pump operating [J] *Pump Technology* , (1):30~34. (in Chinese)

[6]   Qian J R 1981 Research for flow patterns at the impeller chamber inlet of an axial-flow pump [J]. *Drainage and Irrigation Machinery*, (5):8~12. (in Chinese)

[7]   Zhang D S , Shi W D， Guan X F et al. 2009 Measurement on flow field at inlet and outlet of axial-flow pump impeller [J]. *Drainage and Irrigation Machinery*, **27**(4):210~214. (in Chinese)

[8]   Vu T C and Shyy W 1990 Navier-Stokes flow Analysis for hydraulic turbine draft tubes [J].*Fluids engineering*, (112):199~204

[9]   Li Y J, WANG F J 2007 Numerical investigation of performance of an axial-flow pump with inducer [J]. *Journal of Hydrodynamic*, **19**(6):705~711

[10]   Lu L G, Wu K P, Leng Y et al. 2005 Measurement for hydraulic loss of outlet conduit of pumping station[J]. *Drainage and Irrigation Machinery*, **23**(5):23~26. (in Chinese)

[11]   Constantinescu G S, Patel V C and Luan Z G, Khonsari M M 2007 Numerical model for simulation of pump-intake flow and vortices [J]. *Journal of Hydraulic Engineering*, (2):123~134

[12]   Blazek J 2005 *Computational fluid dynamic: principles and application (Second Edition)* [M]. Oxford: Elsevier Ltd .2005:271~302

[13]   Muggii F A, Eisele K, Casey M V 1997 Flow analysis in a pump diffuser-part 2: validation and Limitations of CFD for diffuser flows [J]. *Journal of Hydraulic Engineering*, (11):978~984

[14]   Liu Z X, Liu J Z 2008 *Pumps and Pumping Station* [M].Beijing: China and Hydropower press (in Chinese)