Scholars Journal of Engineering and Technology (SJET) **ISSN 2321-435X (Online)**
Sch. J. Eng. Tech., 2015; 3(4C):477-481 **ISSN 2347-9523 (Print)**
©Scholars Academic and Scientific Publisher
(An International Publisher for Academic and Scientific Resources)
www.saspublisher.com

Research Article

Numerical simulation study of flow field of centrifugal pump based on variable speed

Ma Ming-Yi
Northeast Petroleum University, School of Mechanical Science and Engineering, Daqing, China

***Corresponding author**
Ma Ming-Yi
Email: mapaper@163.com

Abstract: In order to study the variable speed influent to the inner flow field of centrifugal pump, we selected three pump speed, 1500r/min, 2200r/min, 3200r/min, as working condition to conduct on fluid dynamics numerical simulation. The simulation results indicated that with the speed increasing or decreasing, the inner flow field of centrifugal pump has different degrees of changing. Furthermore, at the working condition of 1500r/min and 3200r/min, the inner flow passage have very obvious difference, there are bigger pressure and velocity gradient in the local region. This centrifugal pump has a good performance of working in this speed range.
Keywords: Centrifugal Pump, Variable Speed, Flow Field, Numerical Simulation

Introduction

The centrifugal pump is the widest using fluid machinery in the engineering region, so it is very important for the reasonable operation with high efficient. If the centrifugal pump works in the unreasonable working condition, the cavitation easily happened that is a general damage. In the fact, the centrifugal pump usually was working in the variable speed, its' performance could not be exactly calculated. This paper select a certain type of electrical centrifugal pump as prototype model. And we use the NUMECA software as the technical tool to simulate the inner flow passage of centrifugal pump and analysis the simulated results, such as the pressure field and velocity field. The results indicated that CFD is a good technical tool to test the flow performance of the centrifugal pump, and CFD should be using the fluid machine in a wide scope.

Theory Method of Fluid Dynamics

Navier-Stokes Equations

The general Navier-Stokes equations written in a Cartesian frame can be expressed as:

$$\frac{\partial}{\partial t}\int_{\Omega} U d\Omega + \int_{S} \vec{F}_I \cdot d\vec{S} + \int_{S} \vec{F}_V \cdot d\vec{S} = \int_{\Omega} S_T d\Omega$$

Where Ω is the volume and S is the surface, U is the vector of the conservative variables. $\vec{F}_I$ and $\vec{F}_V$ are respectively the in viscid and viscous flux vectors. E and q_i the total energy and the heat flux components. κ and S_T are respectively the laminar thermal conductivity and the source terms.

Transport Equations for the Standard $k-\omega$ Model

The turbulence kinetic energy k, and its rate of dissipation, ω, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + G_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

In these equations, G_K represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms.

The turbulent (or eddy) viscosity μ_t, is computed by combining k and ε as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Where C_μ is a constant.

The model constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k$ and σ_ε have the following default values:

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

1. Geometry model of pump and numerical simulation method

This paper uses Solid works software and CF turbo software to build a three-dimensional model of centrifugal pump, Fig.1 shows the geometry model of the centrifugal pump. The basic parameters of Impeller model basic parameters are: the number of blades is 6, the axle diameter is 6.5 mm, the inlet diameter is 37.2 mm, the outlet width is 6.2 mm, the outlet diameter is 98 mm, the blade outlet angle is 22°. The flow rate is 200-250 m^3/d. The fluid is water.

This paper uses the finite volume method to discrete Reynolds-Averaged N-S equations, $k-\omega$ turbulent model and the central difference scheme are selected to apply for this simulation. We use NUMECA/AUTOGRID software to generate the structured grid of the inner fluid passage domain as shown in Fig.2. FINE/Turbo was used to calculate the fluid field, and used NUMECA/CFVIEW software to proceed post-processing for centrifugal pump.

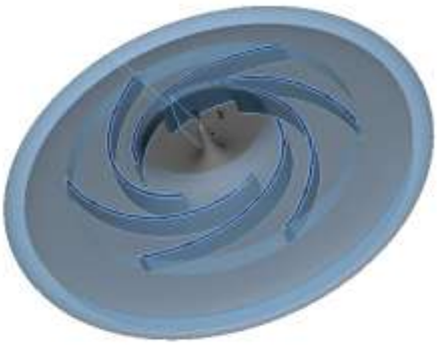


Fig-1:Geometry Model of Centrifugal Pump

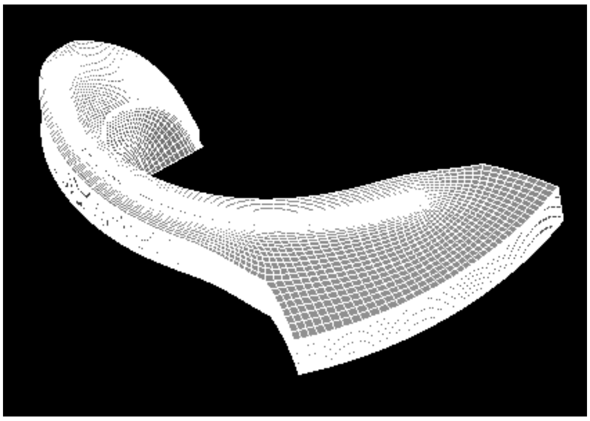


Fig-2: structure mesh of centrifugal pump

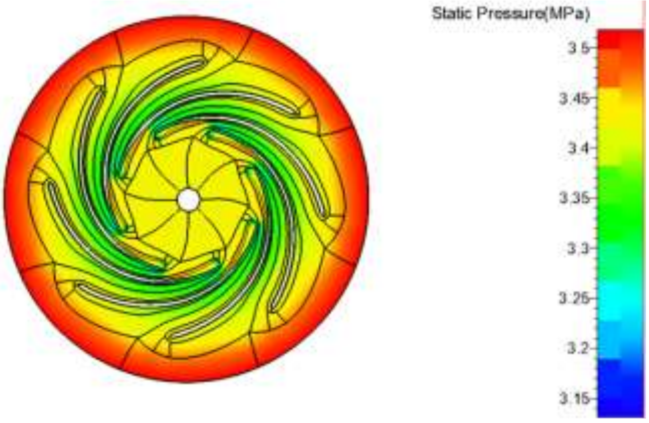
RESULTS AND DISCCUTION

The design working condition is as follows; rated flow rate is 200m3/d~250m3/d, the lift is 8.5m, the rotate speed are 1500r/min, 2200r/min, 3200r/min respectively. This paper respectively simulates the three working condition, then analysis the pressure field.

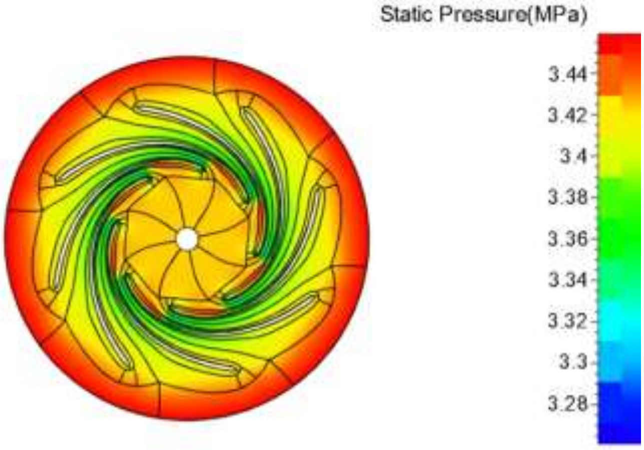
The inner flow of pump impeller is a complex three dimension flow. According to the numerical simulated results, the inner pressure distribution and velocity distribution of pump can be drawn by the post processing software CFVIEW. So we can analysis the inner flow field distribution and flow loss.

Fig.3 was shown the impeller surface static pressure distribution at different rotate speed. From this picture, we can found out the pressure distribution is even very much not only the blade surface but also flow passage. From the entrance of impeller to the discharge, the water pressure increased slowly, and the pressure of pressure surface was bigger than suction surface at the same concyclic.

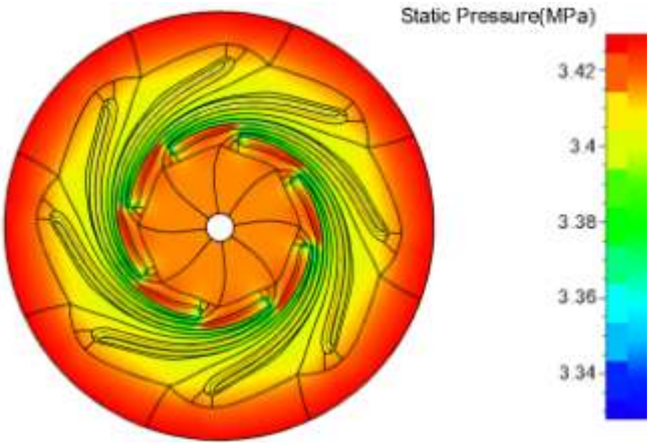
From the Fig.3-a, the Fig.3-b and the Fig.3-c, with the rotate speed decreasing, the static pressure in the impeller passage field decreased little by little. Seeing from the total pressure distribution, the pressure distribution trends have the same law. But with decreasing of rotate speed, the negative pressure scope in the entrance of pump was bigger, the rate of flow was smaller and the cavitation would much more easily happen.



a. 3200r/min



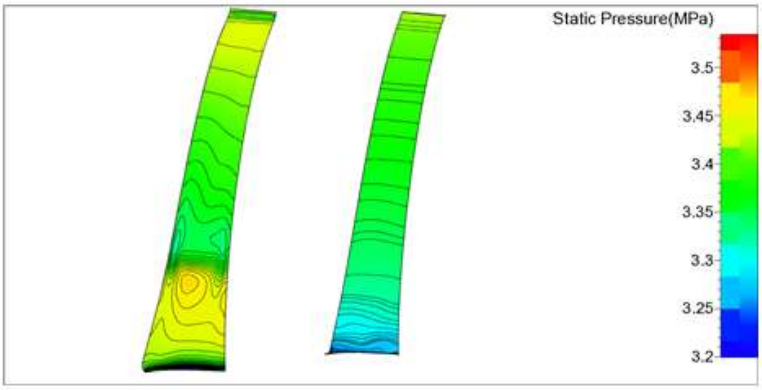
b. 2200r/min



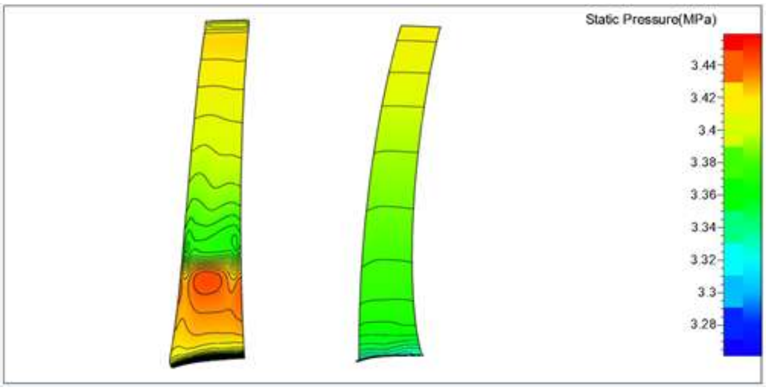
c. 1500r/min

Fig-3: Static pressure distribution of impeller at different rotate speed

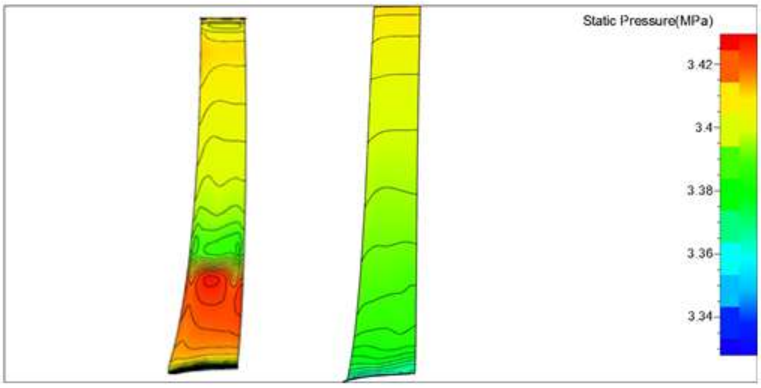
Seeing from Fig.4, the pressure distribution of pressure surface and suction surface were not uniform. In the pressure surface, the fluid flows from the entrance into the impeller passage and cause certain impact, so the pressure of blade leading was bigger than other position. Then the water flow would be suffered viscous frictional drag, there were some pressure loss along the passage, the part of pressure loss consumed the energy of pump, so the pressure decreased. After then, the fluid flowed from passage to the trailing edge of blade, the pressure increased with the increasing of passage radius.



a. 3200r/min



b. 2200r/min



c. 1500r/min

Fig-4: Static pressure distribution in the pressure surface and suction surface at different rotate speed

CONCLUSION

This paper uses NUMECA software to simulate the flow field of the electrical centrifugal pump. The simulated results indicated that the calculated results are nearly conformed to the universal law of centrifugal pump working characteristics, that is to say, with the rotate speed increasing, the energy consumption increases. This indicated the NUMECA software can be fit for simulating the flow field of the centrifugal pump and the calculated results is correct and feasible.

Comparing to the inner flow fields of centrifugal pump at some kinds of rotate speed condition, with the rotate speed increasing or decreasing, the pressure and velocity in the flow passage of pump have different changes. At the small and big rotate speed working condition, the flow field of the inner flow passage has an obvious difference; there are big pressure and velocity gradient in the local region.

The numerical simulated software can really reflect the complex flow phenomenon, and can foresee the new phenomenon that the theoretical analysis and test study can’t find.

REFERENCES

1. Kaewnai S, Chamaoot M, Wongwises S; Predicting performance of radial flow type impeller of centrifugal pump using CFD. Journal of Mechanical Science and Technology, 2009; 236.
2. Liu Dm, Liu Xb, Li J; Numerical simulation of turbine inner flow field based on the numeca software. Water Sciences and Engineering Technology, 2007; 05: 52-55.
3. Li-Ping NFFX; Application of one-equation spalart-allmaras turbulence model in the numerical simulation of internal flows. Journal of Engineering Thermophysics, 2001; 03: 304-306.
4. Xue-qin HUANG, Shu-yun LI, Yu-feng ZHU; Influence of the change of centrifugal pump rotating speed on the pump-unit efficiency. Journal of He Bei Institute of Chemical Technology and Light Industry, 1995; 04: 38-42.