

Research Article

Variable Rotational Speed Influence to Internal Flow Field of Electric Submersible Pump with CFD

Hui-bin GU^{1*}, Ye YUAN², Huan-huan DING³
^{1,2,3}Northeast Petroleum University, Daqing, China.

*Corresponding author

Hui-bin GU

Email: 2008woweifeng@163.com

Abstract: To study the impact of variable speed operation to electric submersible pump internal flow field, the fluid dynamics numerical simulation for impeller passages is accomplished under five kinds of rotational speed. The results show that: with the rotational speed increased, the fluid static pressure is gradually increased from inlet to outlet of impeller flow passages. The impeller flow passages internal fluid flows to the periphery under the action of centrifugal force, so impeller center pressure decreased. When the fluid just flows into the flow passages, the relative velocity is small, the fluid relative velocity along radial direction of the impeller is gradually increased, reaching the maximum at impeller outlet, and the outlet flow velocity is also increased with the increased rotational speed. The revealed of the impact of variable speed operation on electric submersible pump internal flow rule, and provides theoretical basis for pump retrofit optimal design.

Keywords: Electric Submersible Pump, Rotate Speed, Internal Flow Field, Impeller.

INTRODUCTION

The usual method to adjust centrifugal pump flow rate is throttling *regulation*, but the pump rotating rotational speed is constant, so it will be loss a lot of energy on throttling wastage. With the development of frequency control of motor speed technology, pump flow rate under various conditions can be adjusted by variable rotational speed method, which greatly saves energy loss of throttling. It is few achievements that use CFD simulation technology to research electrical submersible pump variable speed operation [1]. To reduce the high cost of electrical submersible pump experimental study, this paper use the CFD software NUMECA to proceed numerical simulation for flow characteristics of electrical submersible pump impeller flow field under the different rotational speed, and it revealed the impact of the rotational speed change to the pressure field and velocity field [2-3]. The electrical submersible pump internal flow characteristics is revealed under variable rotational speed condition, and it provides scientific basis for enlarging its operation scope and retrofit design [4].

CALCULATION MODEL AND CALCULATION METHOD

Using Cfturbo software to build impeller three-dimensional model, the pump entity model diagram is shown in Fig 1. Impeller model basic parameters are: the number of blades is 6, axle diameter

is 6.37 mm, inlet diameter is 36.3 mm, outlet width is 5.73 mm, the outlet diameter is 91 mm, blade outlet angle is 21.8°, the rated speed is 2900 r/min, the rated flow rate is 230 m³/d. The working medium is water.

Finite volume method was applied to discrete Reynolds-averaged Navier-Stokes equations, S-A turbulent model and central difference scheme were selected to solve [5]. Using IGG module to mesh generation structured grid to fluid domain. FINE/Turbo was used to calculate, and used CFview software to proceed post-processing.

Use the following boundary condition calculation:

- (1) Using the velocity boundary conditions at the inlet.
- (2) Using the static pressure boundary condition at the outlet.
- (3) The wall with a no slip wall boundary condition.

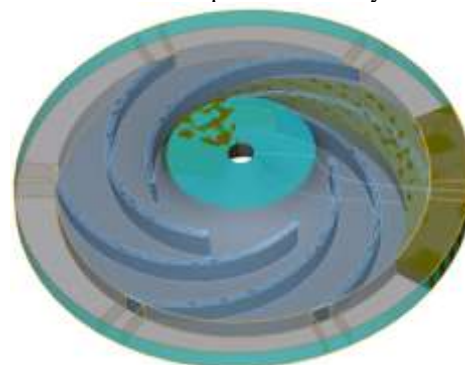


Fig 1: Geometric model impeller

SIMULATION RESULTS AND ANALYSIS

The design conditions of electric submersible pump are: the rated flow is $Q=230\text{m}^3/\text{d}$, head is $H=10\text{m}$, rotational speed is $n=2900\text{r}/\text{min}$. This paper simulated the flow field distribution on $0.6n$, $0.8n$, $1.0n$, $1.2n$, $1.4n$ five kinds of rotational speed conditions, and finally got the pressure and velocity distribution for comparative analysis. In this paper selected $0.6n$, $1n$, $1.4n$ rotational speed conditions to analyze.

Pressure Distribution

As you can see from Fig 2, regardless of the blade and the flow passages, the pressure distribution is relatively uniform. From the impeller inlet to outlet, as the work of the impeller, the pressure of the water is gradually increased and the pressure rose. It can be seen from the a, b, c three figure of Fig 2, with the decrease of the rotational speed, the static pressure of impeller flow field is gradually reduced. Look at from the overall the distribution of pressure field, the pressure distribution trend is basically same.

As shown in Fig 3, the left side of the figure is the work surface of the blade, and on the right is the back of the blade. As can be seen from the figure, the pressure distribution on the blade work surface and the back is not uniform. This is due to the three-dimensional flow of fluid in the impeller and different flow velocity at different radius. In the work face, fluid from the inlet flows into the impeller passages, and it causes a shock, so the leading edge of the blade pressure is relatively higher, then because of the fluid flow will be affected by the viscous friction resistance, and caused a part pressure loss along blade, this part of the pressure loss consumes the pump energy, so the pressure decreased. After this, the fluid along the passage flows to the trailing edge, and the pressure increased with the passages radius increases gradually. The blade back pressure is also increased with the passages radius increases gradually. And on the same radius, the static pressure of the working surface is greater than the back. From Fig 3, the rotational speed is larger, the negative pressure range will be greater at the outlet of blade back. This shows that the greater the flow velocity, the easier to produce cavitation phenomenon on the blade back.

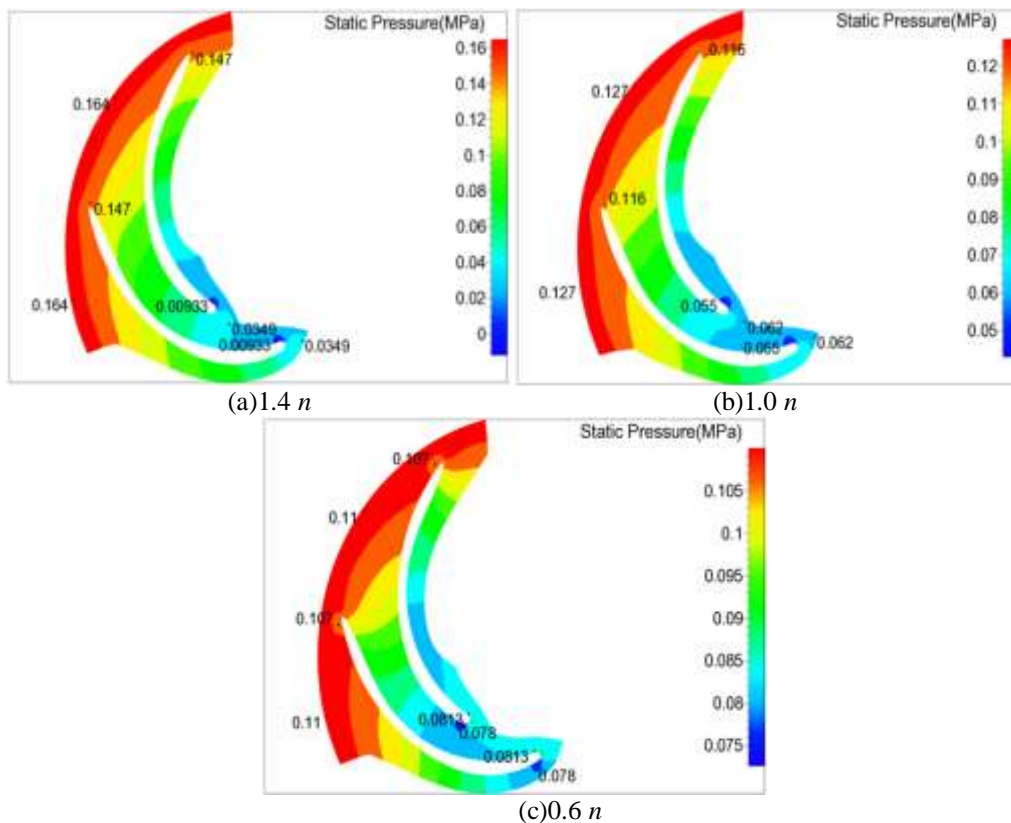


Fig 2: Impeller pressure distribution under different speed

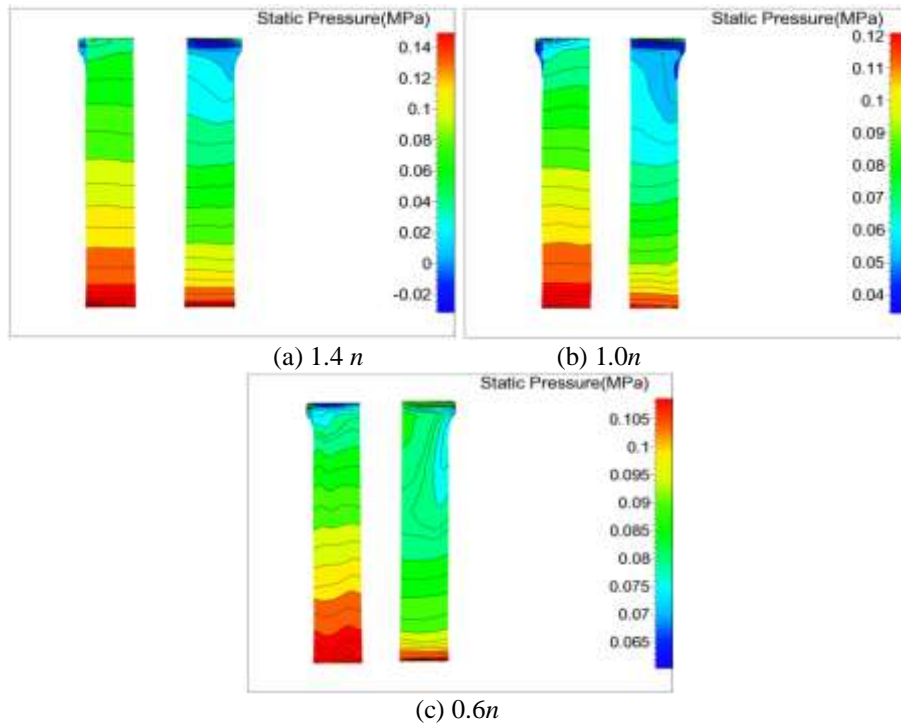


Fig 3: Blade working face and back static pressure distribution under different rotational speed

Velocity Distribution

As shown in Fig 4 for the relative velocity distribution of the impeller surface under the different rotational speed conditions. The fluid relative velocity gradually increases along the radial direction of the impeller, reaches the maximum at the impeller outlet. It can be seen from Fig 4, with the decrease of the rotational speed, the flow velocity at the pump outlet is

also reduced. But at the outlet, the relative velocity is rapid decline, it indicates that the fluid converging in flow passages produces a shock, it will cause great loss of energy, and this ultimately will impact the efficiency of electrical submersible pump. However, with the decrease of the rotational speed, the lower flow velocity range is also decreased.

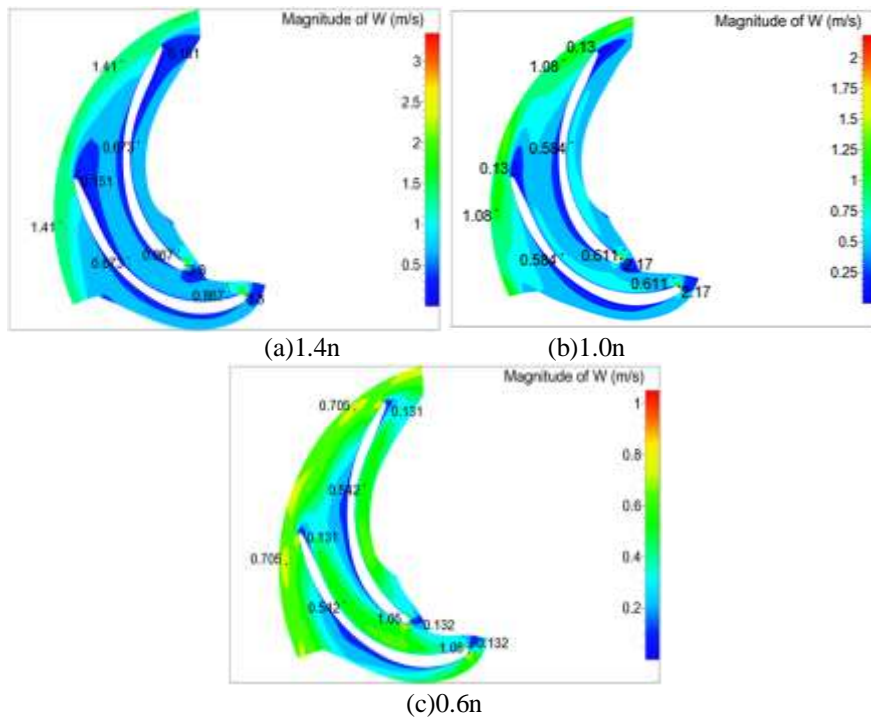


Fig 4: Impeller relative velocity distribution under different rotational speed

CONCLUSION

Using numerical simulation method, under 0.6n, 0.8n, 1.0n, 1.2n, 1.4n five rotational speed conditions, and comparative analysis of the impeller flow field pressure and relative velocity distribution, it gets the following conclusions:

(1)As can be seen from the pressure distribution, with the rotational speed increased, the static pressure is gradually increased from inlet to outlet of impeller flow passages. The larger rotational speed, the lower static pressure at inlet, and the negative pressure range is possible greater. This shows that the greater velocity, the easier to produce cavitation at inlet. The larger rotational speed, the negative pressure range will be greater at the outlet of blade back, and the easier to produce cavitation phenomenon.

(2)As can be seen from the relative velocity distribution, fluid has a shock to pump while it flows into the flow passages, and relative velocity at the inlet small range is large. The fluid relative velocity gradually increases along the radial direction of the impeller, reaches the maximum at the impeller outlet. With the increase of rotational speed, outlet flow velocity is also increased.

(3)As the rotational speed increased or decreased, the pressure and relative velocity of flow field inside electrical submersible pump also has different degree of change. Near the impeller inlet and outlet, fluid pressure and relative velocity near the blade has an obvious change, and this is also one of the factors resulting the electrical submersible pump internal flow field instability.

REFERENCES

1. Suthep Kaewnai, Manuspong Chamaoot, Somchai Wongwiset; Predicting performance of radial flow type impeller of centrifugal pump using CFD, *Journal of Mechanical Science and Technology*, 2009; 23: 1620-1627.
2. LIU De-min, LIU Xiao-bing, Li Juan; Numerical simulation of turbine inner flow field based on the numeca software, *Water Sciences and Engineering Technology*, 2007; 05: 52-55.
3. ZHENG Ying, LIU Xiao-bing, ZENG Yong-zhong; 3-D numerical simulation through the whole flow passage of francis turbine that based on CFD, *Water Sciences and Engineering Technology*, 2008; 06: 70-72.
4. HUANG Xue-qin, LI Shu-yun, ZHU Yu-feng; 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.
5. NING Fang-fei, XU Li-ping; Application of one-equation spalart-allmaras turbulence model in the numerical simulation of internal flows, *Journal of Engineering Thermophysics*, 2001; 03: 304-306.