Numerical simulation and performance prediction in multi-stage submersible centrifugal pump

W J Wang, G D Li, Y Wang, Y R Cui, G Yin and S Peng

Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang, Jiangsu 212013, China

E-mail: liguidong1201@163.com

Abstract. In order to study the inner flow field of multi-stage submersible centrifugal pump, the model named QD3-60/4-1.1 was selected. Steady turbulence characteristics of impellers, diffusers and return channel were calculated by Fluent software, the SIMPLEC algorithm and RNG $\kappa$-$\varepsilon$ turbulence model with sliding mesh technology. Then, the distributions of pressure, velocity and Turbulence kinetic energy was obtained and the distributions of velocity field of a channel were analysed. The results show that the static pressure in impeller is increasing with the increasing of radius. The circumferential component of relative velocity is in the opposite direction of impeller rotating. At the same radius, the component value of pressure surface is larger than suction surface. With the increasing of flow rate, absolute velocity and relative velocity flow angle are becoming small, in opposite of the relative velocity and absolute velocity flow angle. The high turbulent zone of impeller is located in the gap of impellers and diffusers. Flow similarity and structure similarity of the multi-stage submersible pump are confirmed.

1. Introduction
The multi-stage submersible centrifugal pumps are mainly used for agricultural irrigation, water transportation and other occasions. It mainly consists of impeller, guide vane and return guide vane. In addition, the pump designed straight-linked structure has such characteristics: small flow, high head, light weight and so on. Many scholars have carried out a series of studies on multi-stage pump, such as Jianrui Liu et al.[1] carried out three-dimensional numerical simulation study of turbulent flow for model SXB with multistage fire pump. Jianjun Feng et al.[2] systematically studied the applicability of turbulence models by the LDV and PIV. Yunge Zhao et al.[3] simulated the steady turbulent flow field for the multi-stage double-suction centrifugal pumps and obtained velocity vector chart of flow field under different operating conditions of the pump. Peiru Wei et al.[4] calculated three-dimensional numerical simulation study of turbulent flow for model D450-60×7 with energy-saving optimization and analyzed the actual operation data acquisition.

Based on the Reynolds Average Navier-Stokes equation and modification RNG $\kappa$-$\varepsilon$ model using the Fluent software, this paper simulates internal flow in multi-stage submersible centrifugal pump under three kinds of different flow rates of $0.75Q_d$, $1.0Q_d$ and $1.25Q_d$, and discusses pressure distribution and velocity vector of impeller and blade. The results could provide theory for optimization design of the multi-stage submersible centrifugal pump.

2. Control Equation
The RNG $k$-$\varepsilon$ turbulence model provides an option to account for the effects of swirl or rotation by
modifying the turbulent viscosity appropriately. A more comprehensive description of the RNG theory and its applications to turbulence computation can be found in literature [5]. The steady flow is calculated by the second-order implicit time stepping method, and the turbulence model is complemented with a RNG $k$-$\varepsilon$ model [6] and standard wall functions for the near wall, consistent with the non-slip wall condition. The flow of the pump is calculated by the finite volume method to solve the Reynolds averaged momentum equation for 3D steady turbulence flow. And using RNG $k$-$\varepsilon$ equation model makes Reynolds-averaged equation closed.

3. Models and Boundary Conditions

3.1. Pump geometry mode
The simulated object is a multi-stage submersible centrifugal pump named QD3-60/4-1.1. The pump is a low-specific-speed centrifugal pump with a specific speed of 39.5. Main design parameters of simulated pump are as follows: $Q=3m^3/h$, $H=60m$, $n=2860$ r/min, impeller diameter $d_2=118mm$, impeller width at outlet $b_2=2.5mm$, blade outlet angle $\beta_2=20^\circ$, the number of impeller blades $Z=5$, the number of radial vanes $Z_d=6$ and the number of back vanes $Z_r=6$. According to hydraulic diagram of the impeller and guide vanes, three-dimensional flow model of four-stage impellers is established, as shown in figure 1.

![Figure 1. Calculation models](image)

(a) 3-D models
(b) 2-D models

3.2 Mesh generation and boundary condition
Ferziger and Peric et al. [7] pointed out performance prediction error will be gradually reduced with the improvement of the number of the grids. Therefore, the grid independence is checked to give a meaningful comparison between the CFD prediction and experiment. And in order to minimize the effects of boundary conditions and ensure numerical stability, inlet and outlet pipes should be appropriate extended. Structured hexahedral cells were used to define the inlet and outlet domain, and unstructured tetrahedral cells with strong flexibility were used for guide vanes and impellers. The grids number of each stage were 4642, 5067, 289372 and 233786, respectively, as shown in figure 2.
In this paper, the velocity of inlet is determined by the design flow, and the outlet boundary condition is set by free outflow. Boundary parameters has no effect on the upstream. The gradient of outlet parameters along the flow direction is zero. And the wall on each component of the mean velocity and fluctuation velocity are zero. The no-slip condition for the boundary layers was imposed over walls and the standard wall function was used.

4. Pressure Distribution and Velocity Vector of the flow field

4.1 Pressure field analysis

Figure 3 successively shows that the static pressure distribution of middle section for single stage impeller, guide vane and return guide vane under multi-conditions (the small flow condition $0.75Q_d$, the design condition $1.0Q_d$ and the large flow condition $1.25Q_d$). As can be seen from the figure, static pressure distribution evenly increases along the radial direction of the impeller. At the same radius of the impeller, pressure surface of the blade is greater than suction surface about the static pressure. Under each operating condition, the minimum pressure within the impellers is located suction surface of the first-stage blade inlet and the low pressure of suction surface area is growing with the increase of the flow. It is a place that the first-stage impeller is prone to cavitation. When the fluid enters the secondary impeller, the inlet pressure is approximately equal to the static pressure of the first-stage impeller. The inlet of the second stage does not occurs cavitation, because the inlet pressure is much larger than the vaporizing pressure. The rotor-stator interaction generated by the relative movement between the guide vanes and the impellers, resulting in static pressure uneven distribution of the impeller outlet, and the kinetic energy is gradually transformed into the potential energy when liquid flows from the impeller into the guide vane. To the next stage impeller, the potential energy remained constant, and the pressure is evenly distributed, as shown in figure 3 (b), (d), (f).

![Figure 3](image)

**Figure 3.** The distribution of static pressure of every impeller and diffuser under different conditions (Pa)

4.2 Velocity field analysis

Figure 4 successively shows that the absolute velocity and circumferential component of the relative velocity for impellers and vanes under multi-conditions (the small flow conditions $0.75Q_d$, the design conditions $1.0Q_d$ and the large flow conditions $1.25Q_d$). The low velocity region near the inlet of the
impeller is shown in figure 4 (a), (c). Circumferential component of the absolute velocity in the small flow condition is greater than that in the large flow condition. There is a lower velocity part of the high velocity zone. This is because at the larger radius, the impeller passage gradually expanded, the binding capacity of the flow is reduced and the liquid flow occurs offset in the circumferential direction. This phenomenon has also been confirmed in the literature of the PIV experiments [8] [9]. As can be seen from figure 4 (b), (d), circumferential component of the relative velocity is opposite to the direction of the Impeller rotation. Its value is negative at the same radius and the absolute value of the pressure surface is greater than the suction surface. Regardless of what kind of operating conditions, the velocity distribution and the flow pattern in each stage of the impeller are almost the same.

Figure 4. The distribution of $v_u$ and $w_u$ of every impellers and diffusers at middle surface under off-designed conditions (m/s)

Figure 5 shows the velocity triangle of the impeller outlet of the pressure surface under three different conditions. Where $u_2$ is circumferential velocity of the impeller outlet, $v_2$ is absolute velocity of the impeller outlet, $w_2$ is relative velocity of the impeller outlet. From figure 5, you can know as the flow increases, absolute velocity of the impeller($v_2$), circumferential component of the absolute velocity($v_{2u}$) and flow angle of the relative velocity($\beta_2$) are reduced. And radial component of the absolute velocity($v_{2r}$), flow angle of the absolute velocity($\alpha_2$), relative velocity($w_2$), circumferential component of the relative velocity($w_{2u}$) and radial component of the relative velocity($w_{2r}$) are increased. However, the reduced amplitude of flow angle of the relative velocity is small than the increase amplitude of flow angle of the absolute velocity.

Figure 5. Velocity triangle of PS at impeller outlet under different conditions

Figure 6 is an enlarged view of the relative velocity vector of the flow channel of NO.1 under $0.75Q_d$. As can be seen from the figure 6, the phenomenon of stall can be obviously observed in the flow channel of NO.1, 2 and 3. The phenomenon of segregation can be observed in the flow channel of NO.4 and NO.5. The mainstream generates a strong disturbance, due to the flow channel has a large whirlpool area and more severe reflux. This is why the pump performance is low under the small flow conditions.
4.3 Turbulent kinetic energy analysis
The turbulent kinetic energy distribution at impellers, guide vanes and return guide vanes under the design condition can be seen from figure 7 and figure 8. It can be seen that the small changes of turbulent kinetic energy at impellers of No.1, 4, 5, while a high turbulent kinetic energy change region exists at pressure surface and suction surface of No.2, 3 [10][11]. The large gradient of the turbulent kinetic energy area exists at the outlet and causing larger energy loss is due to the rotor-stator interaction. The diffusion tube of turbulent kinetic energy distribution is consistent. The turbulent kinetic energy of guide vane gradually reduced to zero with the increase of diffuser, and the turbulent kinetic energy distribution of the diffuser agree with each other. Figure 8 shows in return guide vanes inlet position, turbulent kinetic energy region has a larger gradient, and return guide vane of the turbulent kinetic energy at the outlet trend to increase, this is because that the secondary impeller inlet disturbance caused. It can be seen, the turbulent kinetic energy distribution of each-stage impeller are almost the same. And because of at all stages in multi-stage pump, the impeller, guide vane and return guide vane are consistent in structure, so it can be considered that internal flow is similar at all levels in the multi-stage pump.

![Figure 6. Relative velocity magnitude of first impeller and enlarge drawing under 0.75Q_d](image)

**Figure 6.** Relative velocity magnitude of first impeller and enlarge drawing under 0.75Q_d

![Figure 7. Turbulence kinetic energy k distributions of different impellers and diffusers at Q_d (m^3/h)](image)

**(a) The first stage (b) The second stage (c) The third stage (d) The fourth stage**

**Figure 7.** Turbulence kinetic energy k distributions of different impellers and diffusers at Q_d (m^3/h)

![Figure 8. Turbulence kinetic energy k distributions of different return channels at Q_d (m^3/h)](image)

**(a) The first stage (b) The second stage (c) The third stage (d) The fourth stage**

**Figure 8.** Turbulence kinetic energy k distributions of different return channels at Q_d (m^3/h)

5. Conclusions
Numerical simulation has the features of short cycle, low cost and high efficiency. Through the
internal flow analysis and performance prediction of multistage submersible pump, part of the test can be replaced by numerical simulation to a certain extent and the special performance requirements of pump design can be optimized in the short term. Numerical simulations of flow field based on multi-stage submersible centrifugal pump, some conclusions are obtained:

1) In the interior of the impeller, static pressure distribution evenly increases along the radial direction. And the static pressure of guide vanes and return guide vanes remained unchanged. To the next stage impeller, the static pressure of return guide vanes remained constant, and the pressure is evenly distributed.

2) The low velocity area is located in the inlet of impeller, which rise with the increase of the radius. At the same radius, the absolute value of the pressure surface is greater than the suction surface.

3) High turbulent kinetic energy in the impeller is located in between the impeller and the positive guide vane relative interference area, the high turbulent kinetic energy of the ABM leaf blade at the inlet; as to the flow disturbance of the secondary impeller inlet, the outlet of turbulent change in kinetic energy of ABM leaf blades is obvious.

4) Flow and structure similarity analysis shows, the impellers structure of multi-stage submersible pump are same, and internal flow pattern is basically the same. In order to reduce the workload of the numerical simulation, the analysis can be performed only on single-stage impeller.

**Acknowledgments**

The present study was supported financially by National Science and Technology Support Program(2011BAF14B01), a Project Funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions(PAPD,BK2009218) and Graduate Training Innovation Project of Jiangsu province(CXLX13_662), China. Their guidance and assistance are gratefully acknowledged.

**References**

[1] Liu J R, Zhang L S, Xiang H G, et al. 2010 *J. Drain. and Irrig. Mach. Eng.* **28**(5) 394-97

[2] Feng J J, Benra F K and Dohmen H J 2010 *J Forsch Ingenieurwes* **74**(3) 123-33

[3] Zhao Yunge et al. 2012 *J. Drain. Irrig. Mach. Eng.* **30**(3) 300-3

[4] Wei P R, Liu W W and Jian W 2010 *J. Fluid Mach.* **38**(09) 31-34

[5] Wang F J 2004 *Computational Fluid Dynamic Analysis-CFD Principle and Application* (Beijing: Tsinghua University Press) p 114

[6] Launder B E and Spalding D B 1974 *Computer Methods in Applied Mechanics and Eng.* **3**(2) 269-89

[7] Ferziger J H and Peric M 1996 *Computational Methods for Fluid Dynamics* (Berlin: Springer)

[8] Pedersen N, Larsen P S and Jacobsen C B. 2003 *J. Fluids Eng.* **125**(1) 61-72

[9] Sinha M, Katz J and Meneveau C 2000 *Trans. ASME J. Fluids Eng.* **122** 108-16

[10] Yuan J P, Fu Y X, Liu Y, et al 2010 *J.Drain. Irrig. Mach. Eng.* **28**(4) 310-14

[11] Solis M, Bakir F and Khelladi S 2009 *Trans. ASME J. Fluids Eng.* **121** 1-13