Study on internal flow field simulation accuracy of centrifugal impellers based on different meshing types

X Q Zheng, Y Wang, Z H Gui and YN Du
Technology center of State Grid Xinyuan Company Ltd., Beijing, 100161, China.

Abstract. Meshing is an important preprocessing part of numerical simulation for water pump, which has important influence on numerical results. In order to simulate internal flow field in water pump more accurately, three different types of grids with hexahedron, polyhedron and tetrahedron are applied to simulate the internal flow field of impeller by using improved k-ω SST turbulence model. Besides, simulating results are compared with PIV testing results by evaluating their accuracy. Numerical simulation results show that the proposed simulation method reduces the requirement for computer configuration. Internal flow field of the reflux pump will be captured more accurately using polyhedral meshes with improved k-ω SST turbulence model, which has a high accuracy and high computational efficiency as well as strong adaptability.

1. Introduction
In recent years, along with the rapid development of the computer technology and computational fluid dynamics (CFD), more and more numerical simulation is applied in there search and optimization design of flow inside the pump fluid machinery [1-3]. Meshing as an important preprocessing part of numerical simulation, it has an important influence on the numerical results.

Due to water pump’s complicated structure, the hexahedron meshing is relatively time-consuming, and therefore, non-structural tetrahedral meshes with good adaptability are now mostly used. However, the tetrahedral grid may be less efficient and less accurate than the hexahedral grid. In recent years, as a new kind of grid technology, with the accuracy of hexahedral grid, polyhedron grid has the characteristic of tetrahedral grid that is easy to generate. Especially for complex structure, with good adaptability, it has been widely used in engineering [4-6]. Some application in the numerical simulation of centrifugal pump is compared for the structural mesh, non-structural mesh, and hybrid mesh in the [7] of the literature. However, at present, research of the mesh unit type on the influence of water pump inner flow prediction accuracy is very few in the numerical simulation of water pump, and there is no literature that has a quantitative analysis with test results. Consequently, it is necessary to carry on the related research.

Milovan Peric [4] used a polyhedron mesh to simulate the flow of engine water jacket and demonstrated the advantages of polyhedron mesh. Frederic Kuznik [5] simulated with polyhedron grid for indoor air flow and heat transfer comparing with the test results, which showed that the polyhedral grid can better simulate the interior flow field. Ben Diedrichs [6] applied polyhedron mesh to analyze the horizontal wind stability of a high-speed train and obtain accurate results. In this paper, take a centrifugal pump for example, three mesh type is applied with improved k-ω SST turbulence model to simulate internal flow, where results are compared with PIV test results to evaluate their accuracy. Numerical simulation results show that the proposed simulation method reduces the requirement for...
computer configuration. Internal flow field of the reflux pump will be captured more accurately using polyhedral meshes with improved $k-\omega$ SST turbulence model, which has a high accuracy and high computational efficiency as well as strong adaptability.

2. Research object of centrifugal pump

In order to study performance of grid cell with hexahedron, polyhedron and tetrahedron in calculation of interior flow for centrifugal pump, a commonly used industrial low specific speed centrifugal pump impeller is selected in this paper. This pump has a relatively abundant experimental data, and its structure form is shown in figure 1. The design flow of this pump is $Q_d = 3.06$ L/s, and the design of single stage head is $H_d = 1.75$ m. The internal flow of impeller in water pump plays a leading role in the overall characteristics of the pump, therefore, flow inside the impeller mainly will be studied in this paper. More detailed geometric parameters can be seen in references [8].

![Figure 1. Schematic diagram of impeller structure for centrifugal pump](image1)

3. Numerical simulation method

3.1. Calculation domain and grid

Appropriate extension is implemented for the upstream and downstream flow in the impeller import and export in order to weaken the influence of import and export boundary conditions on the calculation accuracy. Due to the complicated computational domain shape, adaptable unstructured grids with unstructured tetrahedron grid, hexahedron grid and polyhedron grid are adopted in the process of meshing throughout the computational domain, which improves the orthogonality of the grid to obtain grid with higher quality. In the model of $k-\omega$ SST turbulence model, the near wall area is required fine mesh, therefore, mesh refinement is carried out on the near wall of the blade where the whole computing domain is divided into 1.2 million grids. In order to compare the calculation results of three kinds of grid units, the grid scale is consistent, and the calculation domain and grid are shown in figure 2.

![Figure 2. Grid diagram](image2)
3.2. Boundary Conditions and solving methods

Rotating coordinate system is used in the calculation, and in the impeller imports, velocity-inlet is applied. The speed consists of average speed and pulsating speed where the average velocity direction is along the inlet surface normal, and the average import speed is set as 0.914m/s. The velocity pulsation at the inlet is calculated from the estimated turbulence intensity $I$ and the turbulence length scale $l$ \[^{10}\], where the turbulence intensity can be calculated based on $I = \frac{u'}{\bar{u}} = 0.16(Re)^{-\frac{1}{8}}$ with the calculated value of 4%. The turbulence length scale $l$ is calculated according to $l = 0.07L$ \[^{11}\], and its value is 0.00497m. The rotational direction periodic boundary is used in the direction of the flow direction. The relative pressure value is 0 Pa at the outlet and the Neumann boundary condition is used for the exit velocity. No slip wall boundary condition is applied. Taking the curvature effect of the impeller into account, the improved $k$-$\omega$ SST turbulence model is adopted to simulate the internal flow of the impeller.

3.3. Improved turbulence model

3.3.1. $k$-$\omega$ SST turbulence model. $k$-$\omega$. SST model is based on BSL $k$-$\omega$ model, and according to the characteristics of the boundary layer flow, Menter proposed a kind of stress transport model conforming to the shear transport in shear layer flow. For incompressible flow, transport equation of BSL $k$-$\omega$ model is:

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j k) = \frac{\partial}{\partial x_j} \left[ \frac{\partial k}{\partial x_j} + \frac{\partial(k \omega)}{\partial x_j} \right] - \frac{2(1-F_1)\sigma_{\omega2}}{\omega} \left( \frac{\partial \omega}{\partial x_j} \right)_j \frac{\partial \omega}{\partial x_j} - \frac{\omega}{\omega} \frac{\partial \nu_t}{\partial x_j}$$

and

$$\frac{\partial \omega}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j \omega) = \frac{\partial}{\partial x_j} \left[ \frac{\partial \omega}{\partial x_j} + \frac{\partial(k \omega)}{\partial x_j} \right] - \frac{2(1-F_1)\sigma_{\omega2}}{\omega} \left( \frac{\partial \omega}{\partial x_j} \right)_j \frac{\partial \omega}{\partial x_j} - \frac{\omega}{\omega} \frac{\partial \nu_t}{\partial x_j}$$

Where, $k$ is turbulent kinetic energy; $P_k$ is the formation of turbulent kinetic energy $k$.

The expression of the weight function $F_1$ is:

$$F_1 = \tanh \left( \arg \frac{\partial \nu_t}{\partial x_j} \right)$$

Where

$$\arg = \min \left[ \max \left( \frac{\sqrt{k}}{\rho \omega y}, \frac{500v}{y^2} \right), \frac{4\sigma_{\omega2}k}{CD_{\omega2}y^2} \right]$$

and

$$CD_{\omega2} = 2\sigma_{\omega2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

Different from the BSL $k$-$\omega$ model, the eddy viscosity coefficient of the SST $k$-$\omega$ model is defined as:

$$\nu_t = \frac{a_k}{\max (a_{\nu_t}, SF_x)}$$

Where,
\[ F_2 = \tanh \left( \arg_z^2 \right) \]  
\[ \arg_z = \max \left\{ 2 \frac{\sqrt{\kappa}}{\beta \omega_y}, \frac{500v}{y^2 \omega} \right\} \]  
\[ S = \sqrt{S_\theta S_\eta} \]

3.3.2. Improved k-\( \omega \) SST turbulence model. Based on equation (4), Hellsten defined the correction coefficient of rotation and curvature:

\[ F_4 = \frac{1}{1 + C_n R_i} \]  

Where, recommended value of \( C_n \) is 3.6.

On the basis of Hellsten, this paper considers the rotation influence of the production term \( P_k \) of turbulent kinetic energy \( k \), namely:

\[ G_k = 2\nu_j |S_\theta||\Omega_\eta| \]  

Therefore, the modified k-\( \omega \) SST model expression is as follows:

\[ \frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = G_k - \beta \omega k + \frac{\partial}{\partial x_j} \left[ (u + \sigma_j \nu_j) \frac{\partial k}{\partial x_j} \right] \]  

\[ \frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} = 2\gamma S_\omega^2 - F_a \omega^2 + \frac{\partial}{\partial x_j} \left[ (u + \sigma_j \nu_j) \frac{\partial \omega}{\partial x_j} \right] \]

\[ -2(F_i - 1)\sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \]  

4. Calculation results and analysis

Based on the above calculation model, boundary condition and solving method, three kinds of grid units are applied to simulate the internal flow field of centrifugal pump, and different predicting flow field results are obtained. The calculation results will be analyzed. The flow of centrifugal pump impeller is most concerned, therefore, the flow field in the middle section of the impeller (\( Z / b_2 = 0.5 \)) is analyzed in the post-processing.

Figure 3 shows the static pressure nephogram of the middle section in impeller with the simulation of different grid types. As can be seen from figure 3, the static pressure distribution trend with three kinds of grids is basically the same. The static pressure value of blade face is gradually increasing from inlet to outlet of the impeller (radial direction). Static pressure value in pressure side of the blade is greater than blade back. In the impeller inlet, due to the prerotation, there is a partial low pressure area on the blade back, which is easy to induce flow separation. It can be seen that the low pressure area calculated by the three grids is slightly different, the area of low pressure calculated by the tetrahedron and hexahedron is larger, and the low pressure area with polyhedron model is smaller. At the impeller outlet, wake flow is formed due to the circumference in the blade tail. The results of wake flow in blade tail with three kinds of mesh models are similar. In figure 4, relative velocity nephogram in the middle of the section with different grid type are simulated. With the minimum speed in impeller inlet, the average relative speed values reduce gradually from the suction surface to the pressure side, which is in line with the slip-line field theory. In the process of flowing to the downstream, the average relative speed gradually tends to be more uniform. All of the three kinds of grid models can predict the flow phenomenon of "jet - wake". The further comparison shows that high-speed area with tetrahedron model and hexahedron model is smaller, and the high-speed area in
the flow path of the polyhedron model is larger. Wake flow area with tetrahedron and hexahedron model is large, where in the flow path of the polyhedron model is smaller.

![Figure 3](image)

**Figure 3.** Static pressure nephogram of the middle section \((Z/b_2=0.5)\)

![Figure 4](image)

**Figure 4.** Average relative velocity nephogram of the middle section \((Z/b_2=0.5)\)

In order to further analyze impeller flow performance quantitatively with three kinds of grid models, figure 5 shows the radial velocity and the tangential velocity distribution as well as testing value in the position with axial direction \(Z/b_2=0.5\), radial direction \(r/R_2=0.5\) and \(r/R_2=0.9\) by three kinds of predicting grid model. As can be seen from figure 5 (a), in the area of axial direction \(Z/b_2=0.5\), radial direction \(r/R_2=0.5\), good results with three models on the predictions of a radial velocity are obtained, however, three types of grid model all get a unreal low speed. In comparison, the prediction of radial velocity is closer to the experimental results with polyhedral model. It can be seen from the figure 5 (b), in the area of axial direction \(Z/b_2=0.5\), radial direction \(r/R_2=0.5\), predictions of tangential velocity with three types of grid model are relatively good, and the tangential velocity prediction with polyhedral model is more close to experimental results. Besides, in the area of
axial direction $Z/b_2 = 0.5$, radial direction $r/R_2 = 0.5$. Predicting radial velocity and tangential velocity with three kinds of models are also close to the experimental values, and polyhedron performance is more close to experimental values (in figure 5 (c)) compared with other two kinds of mesh models. From figure 5 (d), it can be seen that the simulated values of the polyhedral model are closer to the LDV experimental values near the pressure surface.
5. Conclusion
In this paper, the model of $k$-$\omega$ SST is improved, and the internal flow field of impeller in a centrifugal pump is studied by the type of hexahedron, polyhedron and tetrahedron.

The polyhedron grid probably need only 1/4 of tetrahedral mesh count with good calculation precision, greatly reducing the number of grid cell. Polyhedron grid can get better results as tetrahedron and hexahedron grid with fewer cell. On the premise of ensuring the accuracy of the calculation, computer configuration and consumption of computing resources are reduced, and computational efficiency is improved.

On the basis of Hellsten, this paper considers the rotation influence of the production term $P_k$ of turbulent kinetic energy $k$, and improved $k$-$\omega$ SST model is built.

Impeller flow performance is analyzed from the perspective of quantitative with three kinds of grid predicting model. Polyhedron grid can well predict the static pressure and relative speed distribution as well as trend in impeller from the suction surface to the pressure surface, which can more accurately capture the reflux of internal flow field. Polyhedron grid has good feasibility and accuracy in the impeller flow numerical simulation.
Reference

[1] Yang Z, Wang F, Zhou P. Evaluation of subgrid-scale models in large-eddy simulations of turbulent flow in a centrifugal pump impeller [J]. Chinese Journal of Mechanical Engineering, 2012, 25(5): 233-242.

[2] Haochen Ma, Rong Ding, Dongsheng Yang. CFD analysis of hump phenomena in low-ratio rotating centrifugal pumps [J]. Fluid machinery.2013, 41(12): 43-47.

[3] Hongjie Wang, Lengfeng Shu, Junlong Zhao, et al. Numerical study on pressure pulsation of turbine pump in non-design conditions [J]. Propulsion technology. 2014, 1: 8.

[4] Peric M. Flow simulation using control volumes of arbitrary polyhedral shape [J]. ERCOFTAC bulletin. 2004, 62: 25-29.

[5] Jasak H, Beaudoin M. Openfoam turbo tools: From general purpose cfd to turbomachinery simulations[C]. American Society of Mechanical Engineers, 2011.

[6] Bayeul-Lainé A, Simonet S, Bois G, et al. Two-phase numerical study of the flow field formed in water pump sump: influence of air entrainment[C]. IOP Publishing, 2012.

[7] Houlin Liu, Mingming Liu, Liang Dong, et al. Influence of grid type on the numerical calculation accuracy of centrifugal pump [J]. Journal of huazhong university of science and technology: natural science edition.2013(10): 64-67.

[8] Pedersen N, Larsen P S, Jacobsen C B. Flow in a centrifugal pump impeller at design and off-design conditions: Part I: Particle image velocimetry (PIV) and laser Doppler velocimetry (LDV) measurements[J]. Journal of fluids engineering. 2003, 125(1): 61-72.

[9] Mathey F, Cokljat D, Bertoglio J P, et al. Specification of LES Inlet Boundary Condition Using Vortex Method[Z]. Antalya, Turkey: 2003.

[10] Fujun Wang. Computational fluid dynamics analysis - CFD software principles and applications [M]. Beijing: Tsinghua university press, 2004.