Hydraulic performance numerical simulation of high specific speed mixed-flow pump based on quasi three-dimensional hydraulic design method

Y X Zhang, M Su, H C Hou and P F Song

College of Mechanical and Transportation Engineering, China University of Petroleum, Beijing 102249, China.

E-mail: jqsumin@163.com

Abstract. This research adopts the quasi three-dimensional hydraulic design method for the impeller of high specific speed mixed-flow pump to achieve the purpose of verifying the hydraulic design method and improving hydraulic performance. Based on the two families of stream surface theory, the direct problem is completed when the meridional flow field of impeller is obtained by employing iterative calculation to settle the continuity and momentum equation of fluid. The inverse problem is completed by using the meridional flow field calculated in the direct problem. After several iterations of the direct and inverse problem, the shape of impeller and flow field information can be obtained finally when the result of iteration satisfies the convergent criteria. Subsequently the internal flow field of the designed pump are simulated by using RANS equations with RNG k-ε two-equation turbulence model. The static pressure and streamline distributions at the symmetrical cross-section, the vector velocity distribution around blades and the reflux phenomenon are analyzed. The numerical results show that the quasi three-dimensional hydraulic design method for high specific speed mixed-flow pump improves the hydraulic performance and reveal main characteristics of the internal flow of mixed-flow pump as well as provide basis for judging the rationality of the hydraulic design, improvement and optimization of hydraulic model.

1. Introduction

Mixed-flow pumps which hydraulic performance and structure lie between the centrifugal pump and axial flow pump are widely used in agricultural irrigation, hydraulic engineering, circulating water system of power station and other departments with advantages of high flow rate, wide range of high efficiency zone and stable performance in low flow rate zone. In recent years, more and more scholars pay attention to the design theory and method, structure research, numerical simulation, flow analysis and performance prediction of high specific speed mixed-pumps [1-3]. The two families of stream surfaces theory [4] is still an advance theory and research method which has been widely used in the inverse problem research of fluid machinery. In view of the development of high performance high specific speed mixed-flow pump hydraulic model, Cao Shuliang etc. use FORTRAN code achieving the hydraulic design of the impeller of high specific speed mixed-flow pump which can not only enhance the accuracy of design calculation, but also provide a platform for further research of optimization design of the mixed-flow pump impeller [5-6]. In this paper, the impeller adopts the quasi three-dimensional design method. The inner turbulent flow field of high specific speed mixed-
flow pump are calculated. The pressure and velocity distributions of the designed pump are analyzed to predict the flow-head curve and the flow-efficiency curve.

2. Quasi three-dimensional hydraulic design

The main parameters of the designed high specific speed mixed-flow pump: the designed flow rate $Q_d$ is 0.384 m$^3$/s, the designed head $H_d$ is 6.53 m, the rotational speed is 1450 r/min and the blade number is 4. Based on the two families of stream surfaces theory, the iterative calculation method of the direct and inverse problem is adopted to achieve the quasi three-dimensional hydraulic design of the impeller of high specific speed mixed-flow pump.

The velocity potential functional equation of the $S_1$ stream surface and the velocity gradient equation of the $S_2$ stream surface which satisfy both of the continuity and momentum equation are established in the direct problem. The finite-element method is applied to calculate the flow on $S_1$ stream surface and the streamline-curvature method is applied to calculate the flow on $S_2$ stream surface. Forty $S_1$ stream surfaces and an average $S_2$ stream surface between two blades are selected for iterative calculation to achieve an accurate calculation result of flow field, thus a 3D problem is converted to two 2D problems. After several times iterative calculation of two families of stream surfaces, the position difference of meridional streamline on the flow field of the $S_2$ stream surface satisfies the requirement of a given converge condition, so the quasi three-dimensional solution is obtained. The relative velocity and pressure distribution on the blade surface, the velocity distribution on the $S_1$ stream surface as well as the meridional velocity distribution on the $S_2$ stream surface are obtained.

The blade camber line differential equation and the circulation differential equation of the $S_2$ stream surface are established in the inverse problem. The characteristic-line method is applied to calculate the flow on the $S_2$ stream surface. Based on the meridional velocity distribution on the $S_2$ stream surface calculated in the direct problem, the blade camber line could be gained and blade shape is drawn by point-by-point integration with the given distribution of velocity circulation controlled by a quartic function and crowding coefficient along the meridional streamline. Then, blades are thickened on the meridional section and blade leading edges are smoothed by adopting Bezier curve lines. The 3D modelling of leading edge of blade after smoothing can be seen from figure 1.

The hydraulic design of impeller is completed after several times iteration of the direct and inverse problem. The 3D modelling of impeller after iterative design of the direct and inverse problem is shown in figure 2.

3. Numerical simulation

The inner flow of high specific speed mixed-flow pump is complex three-dimensional incompressible turbulent flow accompanied by reflux, secondary flow, cross flow, flow separation and other phenomena. The three-dimensional flow field of the designed high specific speed mixed-flow pump
are numerically simulated at seven typical operating conditions (including the designed flow rate) within the scope of 0.6~1.2 designed flow rate by using FLUENT software.

3.1. Three-dimensional modeling and grid partition

The domain for flow filed calculation is the entity—the subtraction of pump overall shell and impeller, shift where fluid cannot pass, as shown in figure 3. An enough long straight pipe which is extended at the impeller inlet is to avoid the influence of rotated flow field of impeller on the designed pump inlet. Meanwhile, considering the influence of calculation convergence and outlet boundary on the inner flow field, a bending pipe is extended to the impeller outlet. The UG software is carried on modelling the impeller, straight pipe and bending pipe. The domain is partitioned by grid generation module ICEM. The unstructured tetrahedral grid with strong flexibility is adopted for the mesh. The quality of grid is controlled greater than 0.3. In addition, in order to reduce the influence of grid number on calculation results, the grid independence analysis of the designed condition is carried, as show in table 1. The correlation of efficiency is less than 1% after calculation, so the influence of grid number on calculation results can be ignored. The final determined total grid number is 1 860 000.

| Grid numbers (ten thousand) | Efficiency (%) |
|-----------------------------|----------------|
| 1                           | 92.12          |
| 2                           | 92.03          |
| 3                           | 91.89          |

3.2. Calculation method and boundary conditions

FLUENT software is adopted to calculate the inner flow fields by assuming that the fluid is steady, viscous and incompressible. The RNG \( k-\varepsilon \) turbulent model and SIMPLE algorithm are applied to solve the RANS equations. The Multiple Reference Frame (MRF) model is applied to take into account the interaction between stationary parts and rotating impeller. The internal flow field is simplified by the instantaneous flow field of impeller fixed in a position, thus the unsteady problem is approximately regarded as steady problem. For rotating impeller, the control equations are solved under the relative coordinate. While, for the stationary parts (the straight and bending pipe), the control equations are solved under the absolute coordinate. The interface of the two sub-regions satisfies the condition of the equal absolute velocity. This method integrates the stationary parts and rotating impeller of the designed mixed-flow pump as a whole domain to achieve numerical simulation as well as using the steady calculation method to solve unsteady problem. For such calculations, standard wall functions based on the logarithmic law have been used. Standard Scheme has been used for pressure terms and second-order upwind discretization scheme has been used for convection terms. The boundary conditions are as follows: inlet is the velocity inlet and assumed as a uniform distribution, outflow is given as boundary condition at outlet, and the solid walls with non-slip condition such as blade surface, hub and shroud are given the moving wall.

3.3. Numerical results and analysis

Figure 4 shows the whole static pressure distribution at symmetrical cross-section of the designed pump period. From the figure 4, it can be seen that the pressure at inlet of pump is relatively low and the distribution is quite uniform. With the increase of flow rate, the pressure tends to be increasing. After the fluid flows into the impeller, the flow along the streamline of blade and the velocity achieve to the maximum at the inlet of impeller due to the high speed rotating blade. Therefore, a low pressure area is formed at the inlet of impeller. And with the increase of flow rate, the minimum pressure value is higher. The pressure changes from low to high along the normal direction of pressure surface of blade. When liquid flows into the bending pipe, the pressure on outboard of pipe is relatively higher.
Figure 4. Whole Static pressure distribution of the designed pump period

From four groups of figure 5 as follows at different flow rate, it can be seen that there is an obvious negative pressure area at the inlet of blade where can easily induce cavitation. At low flow rate conditions, the static pressure distribution of pressure surface increases gradually from hub to shroud, the static pressure of suction surface increases gradually from inlet to outlet and the growth rate is gentle. At the designed flow rate and high flow rate conditions, the static pressure distribution of pressure surface increases gradually from inlet to outlet along pressure surface and the static pressure of suction surface occurs a relatively low area at the mid of the blades. At the four flow rate conditions, as the length of the blade at the shroud is larger than that of at the hub, the flow fluid can get more work energy at the shroud which causes the higher pressure significantly at the outlet near the shroud. In addition, the low pressure zone at blade inlet of pressure surface gradually expands and pressure continuously reduces with the increase of flow rate from low to high. While, the corresponding increase of flow rate causes the rise of pressure at blade outlet of suction surface.

Figure 5. Static pressure distribution of pressure surface and suction surface

Figure 6 shows the relative velocity vector distribution of blades on the impeller at multi-operating conditions. Some results can be got as follows: (1) the relative velocity increases gradually from inlet to outlet. Compared with low flow rate, the minimum value and maximum value of relative velocity on blade both increase with the increase of flow rate. (2) The flow trend is well and the distribution is uniform at multi-operating conditions. There is no obvious cut-off and eddy at the designed condition. However, at the condition of 0.6 designed flow rate, a small scale of back flow appears at inlet of blade very close to the hub and the distribution of velocity is relatively disorder.

Figure 6. Relative velocity vector distribution of blades
Figure 7 shows the relative velocity vector distribution of hub and local amplification of relative velocity vector distribution at inlet of blade. (1) There exists impact phenomenon at inlet of blade. At low flow rate and designed flow rate conditions, the range of impact is small. At high flow rate conditions, there is an obvious impact phenomenon which can cause drastic vibration resulting in bad influence on operating efficiency and lifetime of pump. (2) With the increase of flow rate, the velocity is more uniform. The alteration of velocity has no phenomenon of too emergency or too slow, the energy transformation is uniform which helps to improve the performance of pump. The velocity on pressure surface is bigger than suction surface.

![Figure 7](image1)

Figure 7. Relative velocity vector distribution of hub and local amplification of relative velocity vector distribution at inlet of blade

Figure 8 shows the streamline distributions of inner flow field in the designed pump. It can be seen that at low flow rate conditions, there exists eddy phenomenon close to the axis. At 0.6 design flow rate condition, there is relatively serious reflux at outlet for the separation of flow.

![Figure 8](image2)

Figure 8. Streamline distribution of inner flow field

4. Hydraulic performance analysis

According to the results of numerical simulation, the flow-head curve and flow-efficiency curve can be drawn directly at seven flow rate operating condition points from 0.6~1.2 \( Q_d \). Using the surface integral function provided by FLUENT software, the total pressure of inlet and outlet of pump can be obtained. In the pump, the head of pump is the difference between the total head at outlet and the total head at inlet. The head of pump is showed in equation (1) and the hydraulic efficiency is calculated as in equation (2).

\[
H = \frac{p_{\text{outlet}} - p_{\text{inlet}}}{\rho g} + \Delta Z \tag{1}
\]

\[
\eta = \frac{\rho g Q H}{N} \times 100\% \tag{2}
\]

As it can be seen from figure 9 that with the increase of flow rate, the head of the designed pump declines gradually and the efficiency increases gradually at low flow rate conditions while it drops substantially at high flow rate around 1.2 \( Q_d \). The efficiency of the designed pump is high (the highest
efficiency can reach 92.03%). The range of high efficiency zone is wide. When the efficiency is greater than 83%, the flow rate ranges from 0.269 m$^3$/s to 0.422 m$^3$/s, the head ranges from 3.97 m to 11.30 m.

While, compared with the mixed-flow pump (the highest efficiency can reach 87.15%) designed based on 2D theory [7] which has the same parameters with the designed pump, When the efficiency is greater than 83%, the flow rate ranges from 0.290 m$^3$/s to 0.458 m$^3$/s, the head ranges from 4.90 m to 8.44 m. It can be seen that the head range of the designed pump in the high efficiency zone is wider than the pump designed based on 2D theory. And the highest efficiency of the designed pump in this paper achieves an increase of 4.88%.

Figure 9. The flow-head curve and flow-efficiency curve of the designed pump

5. Conclusions

(1) The quasi three-dimensional design method can successfully apply to the high specific speed mixed-pump impeller and compared with the pump designed based on 2D theory, the hydraulic performance of the designed pump improves, the head range of the designed pump in the high efficiency zone is wider and the efficiency increases significantly.

(2) An obvious negative pressure area appears at the inlet of blade where can easily induce cavitations.

(3) There exist reflux eddy close to the rotational axis and impact phenomenon at inlet of blade. At low flow rate and designed flow rate conditions, the reflux eddy produces heavily while the range of impact is quite small; at high flow rate conditions, there exist only an obvious impact phenomenon which needs to be paid more attention.

Acknowledgement

This research was supported by the Science Foundation of China University of Petroleum, Beijing (NO.KYJJ2012-04-11) and Petro-China Innovation Foundation (2012D-5006-0611).

References

[1] Kim J H, Ahn H J and Kim K Y 2010 Science China Technological Sciences 53(1) 24-27
[2] Li J D, Zeng YZ, Liu X B and Wang H Y 2012 J. Procedia Eng. 31 187-95
[3] Xie R, Shan YJ and Wang X F 2010 J. Drain. Irrig. Mech. Eng. 28(4) 295-99
[4] Wu C H 1952 Trans. ASME 75 p1363
[5] Lin K, Cao S L, Zhu B S and Lu L 2008 J. Tsinghua University: Science and Technology 48 (2) 219-23
[6] Bing H, Cao S L and Tan L 2001 J. Drain. Irrig. Mach. Eng. 29(4) 227-81
[7] Guan X F, Yang J J, Yuan J P, et al. 2002 J.Pump Technology 4 3-8