Numerical Simulation of Unstable Flow in Cooling Pump of Internal Combustion Engine

Chunhao Shen¹, Haoyang Zhang¹, Qiaorui Si¹, Shuijing Xia², Kaile Huang¹ and Jianping Yuan¹*

¹ National Research Center of Pumps, Jiangsu University, Zhenjiang, Jiangsu, 212013, China;
² Huayu Pilberg pump Technology Co., Ltd, Shanghai, 201900, China;
*Corresponding author’s e-mail: yh@ujs.edu.cn

Abstract. Cooling pump is an important device to ensure the reliable and stable working of the internal combustion engine. As the size of the structure is limited by space, its internal flow is complex. To explore the internal flow of cooling pump, the numerical simulation of the full flow field of the model pump under different flow conditions is carried out by using the Detached-Eddy Simulation (DES) model, and the unstable flow characteristics in the flow-path are analyzed. The results show that the intensity of pressure fluctuation near the tongue is the highest and the flow deterioration is the most serious. Under the condition of small flow rate, the area of flow separation is from the tongue to the outlet, return vortices and rotating stall occur in the impeller flow-path, and the outlet vortices dominate the vibration characteristics of the cooling pump. The lowest non-uniformity coefficient of impeller inlet corresponds to the 0.7Qd working point in high efficiency area, but the vertical degree of outflow is same, which is close to the ideal flow direction, and the velocity distribution of flow-path section becomes more uniform with the increase of flow rate. The change of flow rate will affect the unstable degree of the inlet flow, thus affecting the flow rate of the working fluid flowing into the impeller. The unstable degree of the inlet is more serious when the cooling pump works under the condition of small flow rate. The research results can provide some theoretical reference for the structural design of cooling pump.

1. Introduction

These Internal combustion engines are widely used in various national fields, such as transportation, agricultural production, industrial machinery and so on. As an efficient water-cooled cooling component, cooling pump is the key to ensure the stable working of internal combustion engine. The most common malfunctions in internal combustion engines, such as cylinder scoring and detonation, are mostly caused by poor cooling system condition. Improving the internal flow of cooling pump and ensuring the stability and quietness of internal combustion engine working has become a technical problem restricting the development of products in the market. Therefore, it is necessary to study the internal flow characteristics of cooling pump of internal combustion engine.

Cooling pump of internal combustion engine is a kind of centrifugal pump. A lot of experiments and simulations have been carried out on the internal flow characteristics of centrifugal pump by scholars at home and abroad. Paone et al. [1] studied the phenomenon of internal rotation stall of centrifugal pump by combining particle image test method and pressure pulsation method and found that the reflux and external jet phenomenon would occur when the low pass filter pressure was changed under the condition...
of stall. Sinha et al. [2-3] used the PIV technique to study the unsteady flow in the centrifugal pump, and it was found that, when the centrifugal pump produces a stall and was in the stall condition, the leakage and the reflux of the stall in the diffuser were more serious with the decrease of flow rate. Miyabe et al. [4] tested the centrifugal pump under the design and non-design conditions respectively. It was found that the jet-wake phenomenon produced by the relative velocity field exhibits unsteady characteristics under the design conditions, while in the non-design condition, the relative velocity field has a rotating stall phenomenon. Westra et al. [5] used PIV technology and Computational Fluid Dynamics (CFD) method to find that the secondary flow is the cause of the production of the low-speed area of the suction side of the blade and the front shroud area, and the range of secondary flow decreases with the increase of the flow rate. Pedersen et al. [6] used the numerical method to find that the working instability of the pump was increased when the pump works at less than the design flow rate. Feng et al. [7] carried out numerical simulation on the diffuser centrifugal pump by four turbulence models, and found that the SST K-ω and DES turbulence models can better predict the stall phenomenon. Through CFD simulation, Barrio et al. [8] found that "jet-wake", secondary flow and other unstable flow phenomena were the cause of fluctuation of pulsation intensity near the tongue, and there were vortices which are opposite to the rotation direction of the impeller between the impeller outlet and the tongue. Byskov et al. [9] compared the LES and RANS simulation results with the PIV test results, and obtained the conclusion that the LES simulation method has higher accuracy. Based on the SST K-ω turbulence model, Jiang et al. [10] analyzed the unsteady flow field in the diffuser centrifugal pump and found that the pressure distribution in the impeller was mainly affected by the interaction between the impeller and the diffuser. In view of the complexity of hydraulic performance optimization design of traditional centrifugal pump, Zhang [11] proposed a hybrid agent model method by Proper Orthogonal Decomposition and Radial Basis Function (POD-RBF) to reconstruct and analyze the flow field in impeller of centrifugal pump. Through the comparative simulation of open impeller and shrouded impeller, Zhang et al. [12] found that the efficiency of shrouded impeller is higher and the flow state is better. Liu et al. [13] found that the method of changing the height of the volute can effectively improve the internal unstable flow. Dou et al. [14] found that the impeller outlet and tongue were the two most unstable positions, and with the decrease of flow rate, the unstable area shifted from the impeller to the outlet. To sum up, the unstable phenomena such as reflux, secondary flow and rotating stall are particularly prominent when the centrifugal pump deviates from the design condition, but so far there is no much paper about the degree of internal unstable flow and its influence on the performance.

Because of its complex flow-path structure, narrow spatial distribution and specific working environment, the experimental method is not suitable for observation and the numerical calculation method is used to predict the internal flow of the cooling pump. In this paper, based on the Standard k-ω turbulence model, the steady numerical simulation of the whole flow field of the cooling pump is carried out. On this basis, the internal flow field of the cooling pump is simulated by using the Detached-Eddy Simulation (DES), and its internal flow characteristics are obtained, and the pressure fluctuation characteristics and the inlet quality of the impeller is analyzed, and the relationship between this and the flow field in the pump is established.

2. Calculation model

2.1 Study object and parameters
In this paper, the cooling pump of internal combustion engine is studied, as shown in Fig. 1, which adopts centrifugal impeller. The structure of the model pump includes pump shell, impeller and base. The designed flow rate is 10.2 m³/h, the head is 14 m, and the rotating speed is 6750 r/min.

2.2 Calculation domain and grid division
The flow field calculation domain includes inlet, impeller, volute, as shown in Figure 2. In this paper, ICEM software is used to grid the water body in the calculation domain. This study adopts a tetrahedral grid with a strong ability to adapt to the complex geometric structure and fast grid generation taking the
complexity of the geometric model of the cooling pump into consideration [15]. The calculation domain of tetrahedral unstructured grid is divided and the tongue is encrypted. Under the design condition, the prediction results of the external characteristics of the pump with different grid numbers are shown in Figure 3. It can be seen from the diagram that when the grid number is greater than 3183636, the change of the design head of the model pump is less than 1%, so the number of the grid is determined for subsequent calculation.

![Fig. 1 Three-dimensional structure of cooling pump](image)

![Fig. 2 Domains of flow field calculation](image)

(a) pump shell (b) impeller (c) base

(a) inlet (b) impeller (c) volute

2.3 Setting of monitoring points

In order to study the pressure change of important positions from the inlet of the cooling pump to the outlet of the volute, 16 monitoring points are set in the middle section, the inlet and outlet of the pump. As shown in Figure 4, P1, P2, P3, P4 are located on the circumference of the impeller outlet, P5 is located at the impeller inlet, P6, P7, P8 are arranged in a position away from the tongue opening in proper order. Wherein the monitoring point P6 is the nearest from the tongue opening, P8 is the farthest. P9, P10 are on the outlet position and P11-P16 are located in the impeller flow-path.

![Fig. 3 Analysis of mesh independence](image)

![Fig. 4 Monitoring points position in cooling pump](image)

(a) Vertical view (b) End view

3. Numerical calculation method

3.1 Turbulence model

The steady calculation of the model pump adopts the standard $k - \varepsilon$ turbulence model, which takes into account the vortex viscosity coefficient, turbulent kinetic energy and turbulent kinetic energy dissipation, and it is a typical eddy viscosity model. The Detached-Eddy Simulation (DES) [16] is used for dynamic flow field analysis, which is a hybrid model of RANS/LES. Compared with the large vortex simulation, the Detached-Eddy Simulation saves more time, and the near wall region avoids the small scale energetic vortex that the LES model needs to solve, this can predict the turbulence fluctuation better in the boundary layer. The governing equations are as follows:
\[
\frac{D\nu}{Dt} = c_{bl}\nu + \frac{1}{\sigma}\left[\nabla \cdot \left((\nu + V)\nabla \nu + c_{b2}\left(\nabla \nu\right)\right)\right] - c_{w1}f_{w}\left[\frac{\nu}{d_w}\right]^2
\]  
\tag{1}

\[
v_i = \nu f_{v1}, \quad f_{v1} = \frac{x^3}{x^3 + c_{v1}}, \quad x = \frac{\nu}{\nu}
\]  
\tag{2}

\[
S = f_{v3}S + \frac{v}{k^2d_w^2}f_{v2}, \quad f_{v2} = \left(1 + \frac{x}{c_{v2}}\right)^3, \quad f_{v3} = \frac{(1+x f_{v1})(1-f_{v2})}{x}
\]  
\tag{3}

\[
f_{w} = g\left(\frac{1+c_{w3}^6}{g^6 + c_{w3}^6}\right), \quad g = r + c_{w2}\left(r^6 - r\right), \quad r = \frac{v}{Sk^2d_w^2}, \quad c_{w1} = \frac{c_{bl}}{k^2} + \frac{1 + c_{b2}}{\sigma}
\]  
\tag{4}

Where \(d_w\)—the length of the turbulence model. \(\nu\)—the viscosity coefficient of molecular motion, \(S\)—the absolute value of the local vorticity, \(c_{bl} = 0.1355, c_{b2} = 0.622, \sigma = 2/3, \) Carmen constant \(k = 0.41, c_{w1} = 0.3, c_{w3} = 2, c_{v1} = 7.1.\) DES, based on the S-A model, replaces the feature length with:

\[
d = \min\left(d_w, C_{DES}\Delta\right)
\]  
\tag{5}

Where \(\Delta = \max\left(\Delta x, \Delta y, \Delta z\right),\) it is the maximum grid distance. \(C_{DES}\) is generally 0.65.

3.2 Boundary conditions

Based on the commercial CFD software Ansys CFX, the cooling pump is calculated. The standard turbulence model \(k - \varepsilon\) is used to calculate the external characteristics. The medium adopts 25 °C water, the inlet boundary is set with the total pressure inlet, and the outlet boundary condition is set as the mass flow outlet. The rotor–stator interface between impeller and inlet, impeller and volute are also set up. Grid connection method choose GGI. Steady calculation result file is selected as the starting file in the numerical simulation of flow field, and the Transient Rotor Stator mode is set as dynamic and static domain coupling method, and the convergence accuracy is set to \(10^{-4}\). Taking 3° as a time step, the time step is \(\Delta t = 7.4 \times 10^{-5}\) s, and the calculated results of 10 rotation cycles after convergence is saved.

4. Results and analysis

4.1 Flow analysis

4.1.1 The analysis of the external characteristics. Figure 5 shows the comparison of simulation results and experimental results of external characteristics. It can be seen from the diagram that from the closed point to the design flow rate condition, the prediction curve of head and efficiency shows the same trend as the experimental curve, and the numerical value is very close. Under the design flow condition \(1.0Q_d\), the simulated values are almost consistent with the experimental values. Under the condition of full flow rate, the maximum deviation between the test value and the simulated value of the head is not more than 5.5%. From the simulation efficiency curve, the high efficiency area appears near \(0.7Q_d\), and the experimental value of efficiency is very close to the simulated value under the condition of small flow rate. With the increase of flow rate, the simulated value is slightly larger than the experimental value, but the maximum deviation is no more than 4.3%.

4.1.2 Pressure field distribution. Figure 6 shows the static pressure cloud diagram of cooling pump under different flow rates conditions at rotating speed of 6750r/min. It can be seen from the diagram that the pressure at the inlet of the impeller is low under different flow rates, which is affected by the pressure gradient and is easy to cause the reflux phenomenon at the inlet of the pump. The maximum pressure
appears near the tongue and the fluid at the outlet of the impeller is stagnant at the tongue. The kinetic energy is converted into pressure energy, which can reveal the cause of separation of flow under the condition of small flow rate. Under the condition of small flow rate, the pressure gradient of the whole flow-path is much larger than that of large flow condition and pressure gradient is especially significant near the tongue, which indicates that the degree of flow deterioration is more serious. At the same time, the low pressure area on one side of the tongue will block the fluid, reduce the overcurrent area of the fluid, increase the flow rate sharply, and aggravate the instability of the flow field. With the increase of flow rate, the low pressure area at the inlet of impeller increases, but the pressure gradient decreases.

4.2 Flow analysis of impeller flow-path under partial working condition
Under the condition of $0.5Q_d$, the flow distribution diagram of the middle section of the impeller at different times are shown in Figure 6. The interval $T$ corresponds to a rotating period of impeller. It can be seen from the figure that the pressure gradient in flow-path 1, 2 and 6 is very large, and the internal flow is also very complex. The low pressure appears most in the flow-path 3 and the flow-path 4, and reflux occurs at a small area of the inlet of the impeller in the flow-path 4. In the flow-path 3, the low pressure area spreads to the blade pressure surface or even the trailing edge, and the external strong pressure easily causes reflux in a large area of one side of the flow-path 3 near the blade working side, and the reflux area is relatively disturbed with the normal flow fluid on the other side, and a reflux vortex is formed at the outlet.

From the perspective of time change, stall vortexes appear in all flow-paths, but the shape and size of stall vortexes in each flow-path are different, and there is relative motion with the rotating direction of impeller: the stall vortexes in the impeller are produced on the pressure surface of the blade, absorbing the flow energy around them and moving closer to the suction side. It will block the flow-paths, transfer and develop to the next flow-path in the opposite direction of impeller rotation. Because the vortex is produced on the pressure surface of the blade and is close to the outlet, the fluid will directly impact on the surface of the blade, resulting in the vibration of the tail edge of the blade. This vortex mass is also called the "outlet vortex", which may dominate the vibration characteristics of the cooling pump under the condition of small flow rate.
Fig. 6 Flow distribution in impeller domain under 0.5 flow rate

4.3 Comparative analysis of pressure fluctuation in frequency domain

The pressure pulsation frequency domains of the monitoring points in the cooling pump under the design flow and the 0.5 times of design flow rate are respectively shown in Figure 7 and 8. It can be seen from the comparison of the two figures that the pressure pulsation amplitude corresponding to the blade frequency and the frequency multiplication of the monitoring points (P6, P7, P8) in the vicinity of the tongue is obviously higher than that of the impeller inlet area under the design condition, which is easy to generate hydraulic excitation. The reason is that the fluid in the vicinity of the tongue is affected by the rotor–stator interaction of the impeller and the tongue, and the shape of the tongue is generating a turbulent flow, and the unstable pulse is increased. After the intermediate frequency range of 5f0, the amplitude decreases rapidly. Under the 0.5 times of design flow rate, the pulse intensity of the monitoring point P6 near the tongue is obviously higher than that of the other monitoring points, and the peak of main frequency and the frequency multiplication of the monitoring point are obviously increased, indicating that the flow deterioration close to the tongue is more serious.

4.4 The analysis of the flow quality into the impeller

In order to accurately measure the uniformity of flow at the outlet of flow-path [17-18], the velocity inhomogeneity coefficient \( \xi \) and verticality coefficient \( \vartheta \) are introduced to quantify the quality of pipeline outflow (pump inflow). \( \xi \) represents the uniformity of flow velocity. For the outlet section, \( \xi = 0 \) is the most ideal outlet state. The larger the value of \( \xi \) is, the more uneven the flow velocity distribute, and the greater the influence on the efficiency of the cooling pump is. The expression is as follows:

\[
\xi = \frac{1}{Q} \int_{\frac{dA}{A}} |U_a - U| dA
\]

Where \( Q \)—the volume flow of the outflow section; \( U_a \)—the axial velocity on each unit \( dA \) of the outflow section; and \( U \)—the average speed on the outflow section.

The verticality coefficient \( \vartheta \) represents the uniformity of the flow direction and reflects the vertical degree between the outflow direction and the cross section. As the outlet section, \( \vartheta = 90^\circ \) is the most ideal
outflow direction. The closer to 90° the outflow angle is, the better the outflow direction is, and the better use of the flow energy the cooling pump make. The expression is as follows:

\[
\bar{\theta} = 90^\circ - \frac{1}{Q} \int_{A_1} v_z \arctan \left( \frac{v_u}{v_z} \right) dA
\]

(7)

Where \( Q \)—the volume flowrate of the outflow section; \( v_z \)—the axial velocity of the outflow section; and \( v_u \)—the tangential velocity of the outflow section.

Considering the size of the inlet flow-path, the section at the 0.125\( D_0 \) before the inlet of the impeller is selected as the outlet section of the inlet pipeline, where \( D_0 \) is the inlet diameter of the impeller. The changing curves of inhomogeneity coefficient and verticality coefficient with flow rate are shown in Figure 9 and Figure 10 respectively. It can be seen from the diagram that the coefficient of inhomogeneity decreases sharply at first and then increases with the increase of flow rate, and the lowest value appears at the working point of 0.7\( Q_d \).

The velocity vector diagram of the section at 0.125\( D_0 \) before the impeller inlet is shown in Figure 11. As can be seen from the diagram, when the flow rate is under the condition of 0.3\( Q_d \), a large area of the vortex appears near the center of the shaft surface. As the flow rate increases, the area of the vortex is gradually reduced, indicating that the flow condition of the flow field tends to improve. It can be seen from the distribution of speed, the increase of flowrate makes the velocity distribution of the cross-section of the flow-path more uniform. Figure 12 is a cross-sectional velocity vector diagram of the inlet side. As can be seen from the diagram, as the flow rate increases, the relative expansion of the volume of the fluid in the inlet pipeline produced a squeeze, this breaks the original pressure gradient, and the velocity distribution of the cross section of the flow-path becomes more uniform.
5. Conclusion

(1) Under the condition of small flow rate, the pressure gradient of the whole flow-path increases, and the area of flow separation even spreads from the tongue to the outlet, which seriously weakens the hydraulic performance of the pump. There is a certain phase deviation in the time domain cycle of different flow-paths, which indicates that the phenomenon of "rotating stall" appears in the impeller flow-path.

(2) The vortex is produced on the pressure surface of the blade and is close to the outlet, which leads to the vibration of the tail edge of the blade. The "outlet vortex" may dominate the vibration characteristics of the cooling pump under the condition of small flow rate. With the decrease of flow rate, the pulsation intensity of the monitoring point near the tongue is significantly higher than that of other monitoring points, and the main frequency and its frequency multiplication peak value increase significantly, which indicates that the closer the flow near the tongue, the more serious the deterioration of the flow.

(3) The change of flow rate affects the non-uniformity coefficient, and the lowest point is the $0.7Q_d$ working point in the high-efficiency area, which indicates that the axial flow rate of the outlet cross-section is more uniform than other flow working conditions. Under different flow working conditions, the difference of the outflow verticality is small, and the velocity distribution of the cross section of the flow-path becomes even more uniform from the condition of the distribution of the flow velocity of the shaft surface.

References

[1] Paone, N., Riethmuller, M.L., Braembussche, R.A.V.D. (1989) Experimental investigation of the flow in the vaneless diffuser of a centrifugal pump by particle image displacement velocimetry. Experiments in Fluids, 7: 371-378.

[2] Sinha, M., Katz, J. (2000) Quantitative visualization of the flow in a centrifugal pump with diffuser vanes-I: on flow structures and turbulence. Journal of Fluids Engineering, 122: 97-107.

[3] Sinha, M., Pinarbasi, A., Katz, J. (2001) The flow structure during onset and developed states of rotating within a vaned diffuser of a centrifugal pump. Journal of Fluids Engineering, 123: 490-499.

[4] Miyabe, M., Ffurukawa, A., Maeda, H., et al. (2006) On the unstable pump performance in a low specific speed mixed flow pump. In: 23rd IAHR Symposium on Hydraulic Machinery and Systems, Yokohama. pp. 115-120.

[5] Westra, R.W., Broersma, L., Van, A.K., et al. (2009) Secondary flows in centrifugal pump impellers: PIV measurements and CFD computations. In: ASME Fluids Engineering Division Summer Meeting. Colorado.

[6] Larsen, P.S., Pedersen, N., Jacobsen, C.B. (2003) Flow in a centrifugal pump impeller at design and off-design conditions. Part 1: PIV and LDV measurements. Journal of Fluids...
Engineering, 125: 61-72.

[7] Feng, J.J., Benra, F.K., Dohmen, H.J. (2010) Application of different turbulence models in unsteady flow simulations of a radial diffuser pump. Forschung im Ingenieurwesen, 74: 123-133.

[8] Barrio, R., Parrondo, J., Blanco, E. (2010) Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. Computers and Fluids, 39: 859-870.

[9] Byskov, R.K., Jacobsen, C.B., Condra, T., et al. (2004) Large eddy simulation for flow analysis in a centrifugal pump impeller. Advances in LES of Complex Flows, Springer Netherlands.

[10] Jiang, W., Li, T., Wang, Y.C., et al. (2017) Numerical Simulation and Experiment of Flow Field in Centrifugal Pump with Vane Diffuser. Transactions of the Chinese Society for Agricultural Machinery, 48: 121-128.

[11] Zhang, R.H., Chen X.B., Guo G.Q., et al. (2018) Reconstruction and Modal Analysis for Flow Field of Low Specific Speed Centrifugal Pump Impeller. Transactions of the Chinese Society for Agricultural Machinery, 49: 143-149.

[12] Zhang, Q.H., Yang X.Y., Xu, Y.H., et al. (2018) Influence of different impeller structure on the performance of engine cooling water pump. Fluid Machinery, 46: 27-31.

[13] Liu, T.T., Wang, T., Yang B., et al. (2009) Numerical simulation and structure improvement for a car pump with opened centrifugal impeller. Journal of Engineering Thermophysics, 30: 961-963.

[14] Dou, H.S., Jiang, W., Zhang, Y.L., et al. (2014) Flow instability in centrifugal pump based on energy gradient theory. Transactions of the Chinese Society for Agricultural Machinery, 45: 88-92.

[15] Lin, G. (2017) Study of Flow Induced Radiated Noise in a Multi-Stage. Jiangsu University, Zhenjiang.

[16] Jiang, X.Q., Li, Y.B., Li, J.Z., et al. (2018) Numerical study on transient flow in centrifugal pump based on DES. Pump Technology: 20-23.

[17] Liu, R.W., Huang, G.F. (2011) Numerical study on effect of inlet lip on hydrodynamics for waterjet propulsion. Shipbuilding of China, 52: 39-45.

[18] Shi, W.D., Zhang G.J., Zhang D.S., et al. (2014) Effects of non-uniform suction flow on performance and pressure fluctuation in axial-flow pumps. Journal of drainage and irrigation machinery engineering, 32: 277-282.