Unsteady flow characteristic analysis of a centrifugal pump under over-load operating conditions

Xiaoran Zhao¹, Yexiang Xiao¹, Zhengwei Wang¹, Jin Zhang¹
¹ State Key Laboratory of Hydroscience and Engineering & Department of Energy and Power Engineering, Tsinghua University, Beijing 100084, China

E-mail addresses: wzw@mail.tsinghua.edu.cn

Abstract. Unsteady flow phenomena, including flow separation, vortex flow, back flow and interaction between tip vortex and main flow, can occur in the centrifugal pump under over-load working conditions. Previous researches mainly focus on the flow characteristic under part-load working conditions, which is quite different from cases at over-load conditions. In this paper, flow characteristics are investigated in a centrifugal pump by CFD technology. The SST-CC turbulence model is adopted to perform steady simulations and the SAS-CC model is used in unsteady simulations of the centrifugal pump under 1.00\(Q_{\text{BEP}}\) - 1.45\(Q_{\text{BEP}}\) working conditions.

Flow characteristic is proceeded with energy loss analysis by entropy method and pressure amplitude spectra. The results indicate that there appears a significant vortex core inside the impeller when the flow rate is larger than 1.28\(Q_{\text{BEP}}\) and the flow in the diffusing part and tongue region of the volute is complicated with several swirling vortices. The main frequency and amplitude are also investigated. There exists an evident low-frequency component near the volute tongue under over-load conditions, which is related to the vortices and flow separation inside the volute.

1. Introduction

Centrifugal pump is a widely used hydraulic machine, which has been applied to various utilizations in industrial and agricultural production. With the advantages of simple structure, high work reliability and large operating range, centrifugal pump needs to be researched deeply and comprehensively. At present, most researchers are focused on the flow characteristics, like rotating stall and flow separation, in the pump impeller under part-load conditions. However, there are also many valuable flow phenomena which could affect the performance of pumps under over-load working points.

There are some papers aiming at describing the flow structures and pressure features inside centrifugal pumps at large flow rate working points through both numerical and experimental methods. Keller [1] conducted PIV measurements on a centrifugal pump at 150% of designed flow rate and observed the flow detachment inside the impeller, which eventually developed into vortices between blades. The vorticity distribution illustrated that there is a high-vorticity zone near the volute tongue, especially when the blade approaches to the tongue. Hou [2] found numerically that the energy loss in the impeller is the most important component through the whole flow passage under over-load working conditions in a centrifugal pump with diffuser. The flow structures inside the impeller could have an influence on the volute since the uneven pressure patterns can propagate downstream. Guo [3] carried out experiments on a centrifugal pump to detect the pressure and vibration and found that there is a pressure undulation circumferentially along the volute but the static pressure is low. Wang [4] compared
the pressure pulsation in a centrifugal pump volute under 60% and 120% of designed flow rate points based on LES method and stated that the amplitude is significantly larger when the flow rate is high. Gao [5] also came to the similar conclusion by showing the fact that the main frequency is the same while the amplitude is 4.77% higher at 117% of designed flow rate point than that at 78% of designed flow rate point. The obvious pressure pulsation characteristics when the flow rate is larger than the designed point could lead to vibration, noise and other problems [6][7][8]. The researches mentioned above all point out that it is necessary to study the flow characteristics under over-load working conditions. However, the energy dissipation related to the specific flow structures has not been investigated in detail. The unsteady patterns of flow inside both the impeller and volute have not been analyzed.

This paper performs simulations on a three-blade centrifugal pump under 1.00\(Q_{BEP}\) - 1.45\(Q_{BEP}\) over-load working conditions. The pressure pulsations are investigated, combining with the analysis on internal flow field and energy dissipation, to illustrate the unsteady flow characteristics.

2. Simulation model

2.1. Geometry of the centrifugal pump

The whole flow passage of a centrifugal pump is simulated in this paper. The computational flow field is shown in Figure 1. The impeller is with 3 blades and the impeller outlet diameter of the pump is 310 mm. The back and front chambers are involved in simulations with widths of 0.38 mm and 0.62 mm respectively. The rotating speed of the pump is 1000 rpm and the designed flow rate is 78.82 l/s.

![Figure 1. The computational flow field of the centrifugal pump.](image)

2.2. Mesh generation

In simulations, it is necessary to improve the mesh quality in order to ensure both the accuracy and efficiency of the simulations. As shown in Figure 2, the hexahedral structural meshes are generated in suction pipe, volute, back blades, front and back chambers domains. The tetrahedral unstructured meshes in the impeller are produced. All of the mesh elements are generated in ANSYS ICEM 16.0.
2.3. Turbulence model and boundary conditions

A commercial CFD software ANSYS CFX 16.0 is used in simulations. SST $k$-$\omega$ turbulence model is applied in steady simulations while SAS-SST turbulence model is used in unsteady simulations. SAS-SST method is an efficient methodology to predict the unsteady feature of complicated flow structures, including the swirling and separating flow. In the meanwhile, it costs less computational time and source than the LES method. The governing equations are expressed as follows:

Continuity equation
\[ \frac{\partial \rho C_i}{\partial x_i} = 0 \]  

Momentum equation
\[ \frac{\partial}{\partial t} \left( \rho C_i \right) + \rho \frac{\partial}{\partial x_j} \left( \rho C_i C_j \right) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \tau_{ij} - \rho C_i C_j \right] \]  

Total energy equation
\[ \frac{\partial \rho h}{\partial t} + \frac{\partial (\rho h C_i)}{\partial x_j} = \frac{\partial p}{\partial x_j} + C_i \frac{\partial p}{\partial x_j} + \rho C_i \frac{\partial C_j}{\partial x_j} + \tau_{ij} \frac{\partial C_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \rho h C_i - q_i \right) \]

For the three-bladed centrifugal pump, the blade is highly curved. Therefore, curvature correction [9] is added as an advanced turbulence control.

Mass flow rate boundary condition is adopted at the suction pipe inlet with 25°C temperature, in order to be consistent with actual situations. Outlet boundary condition at the volute outlet is set as static pressure. No-slip and adiabatic wall conditions are used in solid boundaries. In unsteady simulations, the selected time step is $2 \times 10^{-4}$ s, which is 1/300 of a rotation period. The unsteady simulations include 15 impeller revolutions, among which the latter 10 revolutions are used for pressure pulsations analysis.

2.4. Monitor points arrangement

Monitor points are distributed in the volute to record pressure signals and to conduct pressure pulsation analysis. There are 18 monitor points in the volute circumferentially and in the diffusing part, as shown in Figure 3.
3. Numerical Results

3.1. Inner Flow Structure Analysis

In order to detect the flow characteristics under over-load conditions, the flow structures in the impeller and volute are explored at $1.00Q_{BEP}$, $1.07Q_{BEP}$, $1.15Q_{BEP}$, $1.22Q_{BEP}$, $1.29Q_{BEP}$, $1.33Q_{BEP}$, $1.39Q_{BEP}$ and $1.45Q_{BEP}$ working points. The flow passes the impeller passages smoothly at the designed point with an even velocity distribution, while there appear vortexes when the flow rate increases. Figure 4 displays the inner flow behavior inside the impeller under three typical working conditions ($1.00Q_{BEP}$, $1.22Q_{BEP}$ and $1.45Q_{BEP}$) for examples. It can be seen that when the flow rate is larger, two obvious low velocity areas appear and eventually turn out significant vortices in the middle of flow passages.

The vortices in the impeller would propagate downstream and affect the flow patterns in the volute under over-load conditions. The interaction between the rotating flow and stationary part could intensify the disorder in the volute. According to the second law of thermodynamics, entropy is a measurement of the thermodynamic irreversible process and the irreversibility is the original source of energy dissipation. As shown in Figure 5(a), the entropy distributes uniformly in the volute at the designed point. However, at $1.22Q_{BEP}$ working point, an unsmooth distribution appears at the spiral section, as displayed in Figure 5(d). When the flow rate is larger, the high entropy area exists at both the spiral section and diffusing part near the volute tongue.
For the purpose of investigating the flow characteristics inside the volute, the velocity vector distributions are used to display the flow structure under 1.00, 1.22 and 1.45 working points, as shown Figure 6. It is detected that the low-velocity area accounts for more space under larger flow rate working conditions with evident swirling flow at the spiral section near the volute tongue. Under the largest flow rate condition, the swirling flow exists and a flow separation appears in the diffusing part, which could both lead to pressure pulsations.

3.2. Pressure Pulsation Analysis

Unsteady simulations of the centrifugal pump are performed under 1.00, 1.22 and 1.45 over-load conditions. Pressure pulsation spectra of all the circumferential monitor points are obtained
by FFT methods, as shown in Figure 7. The normalization $c_p$ is defined as equation (4), in which $U_{out}$ is the circumferential velocity at the impeller outlet:

$$c_p = \frac{P - \overline{P}}{0.5 \rho U_{out}^2}$$

(4)

The main frequencies under three conditions are the same as $3f_n$ ($f_n$ is the rotating frequency), while the amplitude of main frequency is higher with larger flow rate. The main frequency can be demonstrated due to the rotor-stator interaction (RSI) between the impeller and volute tongue in the three-bladed pump. At the designed point, there is no evident low-frequency component. At 1.22$Q_{BEP}$ condition, the low-frequency pulsation is found at pbt7, which is the monitor located at the volute tongue. At 1.45$Q_{BEP}$ condition, the pressure pulsation near the tongue is more complicated with several obvious frequencies lower than $3f_n$. Besides, at this working point, the pulsation is significant at monitor point pbt8, which is near the diffusing part in the volute.

The pressure spectra of monitor points pout5~8 are shown in Figure 8. The amplitudes of frequency $3f_n$ are not conspicuous because these points are located far away from the impeller outlet and the influence of RSI is not significant. The frequency components smaller than $8f_n$ is obvious with complexity, which could be related to the flow separation in the diffusing part.

In general, the pressure pulsation characteristics act in accordance with the flow features in the volute. The pressure analysis in pout1~4 are not presented in this paper since the amplitudes are quite small in the volute outlet.

![Figure 7. Pressure pulsation spectra of monitor points pbt1~10.](image-url)
4. Conclusion
This paper presents numerical simulations on a three-bladed centrifugal pump with front and back chambers under 8 over-load working conditions. The flow structures and energy dissipation distributions are analyzed in the impeller and volute. Pressure pulsation spectra under three typical working conditions are analyzed through FFT method and some representative dominant frequencies are found in the volute. The characteristic of dominant pressure frequencies and amplitudes agree with the flow feature.

(1) There are vortex structures with low velocity in the middle of impeller flow passages under over-load conditions, which spread downstream to the volute.

(2) The energy dissipation is uniformly distributed in the volute at the designed point while high entropy area appears near the volute tongue when the flow rate is larger than $1.22Q_{BEP}$. This phenomenon is related to the swirling flow in spiral section and separating flow in diffusing part of volute.

(3) The main frequency of pressure pulsation in the volute is $3f_{e}$ due to the rotor-stator interaction between the impeller and the volute tongue. The amplitude increases with the flow rate.

(4) The frequency components smaller than $3f_{e}$ are more complicated near the volute tongue and in the diffusing part of the volute under over-load conditions. This phenomenon refers to the flow structures in the tongue region.

Acknowledgments
Special thanks are due to the State Key Program of National Science of China (Grant No. 51439002), National Natural Science Foundation of China (No. 51279083), the National Key Research and Development Program of China (No. 2017YFC0404200) for supporting the present work.

References

[1] Keller J, Blanco E, Barrio R., Parrondo J 2014 PIV measurements of the unsteady flow structures in a volute centrifugal pump at a high flow rate. Exp Fluids. 55 (10), 1820.

[2] Hou H, Zhang Y, Xu C, Zhang J, Li Z 2016 Effects of radial diffuser hydraulic design on a double-suction centrifugal pump. IOP Conf Ser-Mat Sci. Vol. 129, No. 1, p. 012017.

[3] Guo S, Maruta Y 2005 Experimental investigations on pressure fluctuations and vibration of the impeller in a centrifugal pump with vaned diffusers. JSME Int J B-Fluids T, 48 (1), 136-143.

[4] Wang W, Cui Y, Wang Y, Li G, Liang Q, Yin G 2013 Analysis on the blade inlet pressure fluctuation of the centrifugal pump based on LES. IOP Conf Ser-Mat Sci. Vol. 52, No. 2, p. 022025.

[5] Gao Z, Zhu W, Lu L, Deng J, Zhang J, Wuang F 2014 Numerical and experimental study of unsteady flow in a large centrifugal pump with stay vanes. ASME J Fluids Eng. 136 (7), 071101.

[6] Yuan S, Yang J, Yuan J, Luo Y, Pei J 2012 Experimental investigation on the flow-induced noise under variable conditions for centrifugal pumps. Chin J Mech Eng, 25 (3), 456-462.

[7] Si Q, Dupont P, Bayeul-Lainé A, Dazin A, Roussette O, Yuan S 2015 An experimental study of the flow field inside the diffuser passage of a laboratory centrifugal pump. ASME J Fluids Eng. 137 (6), 061105.

[8] Gao M, Dong P, Lei S, Turan A 2017 Computational Study of the Noise Radiation in a Centrifugal Pump When Flow Rate Changes. Energies, 10 (2), 221.

[9] Spalart P, Shur M 1997 On the sensitization of turbulence models to rotation and curvature. Aerosp Sci Technol. 1 (5), 297-302.