Pressure fluctuation characteristics of centrifugal pump at low flow rate

X F YAN¹ and G Q Q G MENG¹

¹Inner Mongolia Vocational College of Chemical Engineering, Hohhot 010070, Inner Mongolia, China

E-mail: erenmangha2012@163.com

Abstract. The current investigation is aimed to simulate the unsteady internal flows in a centrifugal pump impeller with seven twisted blades by using a three-dimensional Navier-Stokes equations with Scale-adaptive simulation (SAS) turbulence model. A detailed analysis of the results at design flow rate \( Q_0 \) and low flow rate \( 0.5Q_0 \) and \( 0.3Q_0 \) is presented. Unsteady flow analyses in centrifugal pump are focused mainly on the volute. The calculation results of pump head and efficiency at different flow rate conditions is in good agreement with the experimental data. Nine monitor points are positioned in the volute. The frequency of pressure fluctuations on all monitor points are analyzed and discussed. Under different flow rate conditions, the dominant frequency of pressure fluctuation in volute are the blade pass frequency and its integer multiples, the blade pass frequency always played the dominant role, and the largest pressure fluctuation maximum amplitude occurs near the volute tongue zone. With the flow rate decrease, the intensity of pressure fluctuation gets stronger, especially in spiral zone is the most obvious and the amplitude value of pressure fluctuation is the most dramatically increased. The maximum amplitude of pressure fluctuation in spiral zone is about 2-3 times of design flow rate. Moreover, the streamline distribution in pump flow passage at flow rate \( Q_0 \), \( 0.5Q_0 \) and \( 0.3Q_0 \) is analyzed. The results demonstrate that with the flow rate decrease, the vortex in the impeller flow passage is generating, developing and even blocking the whole impeller flow passage, which is disturbs the internal flow field and induces the strong pressure fluctuations in volute spiral zone. This research provided a reference for expanding the operation range of centrifugal pump and optimization design of volute in centrifugal pump.

1. Introduction
The major sources of centrifugal pump pressure fluctuation is the rotor-stator interactions. In recent years, many studies have been done on the pressure fluctuation induced by the rotor-stator interactions in centrifugal pump [1-4]. The centrifugal pump is working under off-design flow rate condition, especially low flow rate, it is easy to occur the flow separation [5], rotating stall [6], secondary flows [7], cavitation [8,9,10], and so on. It causes the centrifugal pump internal flow field instabilities. The flow field instabilities can cause various losses in the pump, resulting in the unstable operation of the centrifugal pump. The centrifugal pump flow rate is less than 50% of the design flow rate, the rotating stall occurs [11,12]. Cavitation has large influence on pressure fluctuations in centrifugal pump impeller under low flow rate than under large flow rate [13]. Duplaa et al. [14] tested a cavitating centrifugal pump during fast startups and observed the water hammer phenomena under low flow rate.
Zhang et al. [15] tested the unsteady pressure fluctuation in a centrifugal pump, and obtained that under low flow rate, the low frequency components of the high amplitude in the centrifugal pump volute. Because in the design of centrifugal pump, the first consideration is the reliability of operation. Hence, under low flow conditions, the future study of the internal flow field of centrifugal pump is very necessary.

In this paper, the unsteady internal flows in a centrifugal pump calculate by using a three-dimensional Navier-Stokes equations with Scale-adaptive simulation (SAS) turbulence model, and verify the calculation results through test. Under the design flow rate \((Q_0=25\text{m}^3/\text{h})\) and low flow rate \((0.5Q_0\text{ and } 0.3Q_0)\), the pressure fluctuation characteristics in volute is analyzed, the streamline distribution in the pump flow passage is also discussed.

2. Problem statement

2.1. Computational model

The continuity and momentum equations is based on Reynolds' average Navier-Stokes equations, as follows:

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m u_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (\rho_m u_i)}{\partial t} + \frac{\partial (\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu_m + \mu_t \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \right]
\]

Where \(\rho\) and \(\mu\) are the density and dynamic viscosity respectively; subscript \(m\) denote the mixture, subscripts \(i\) and \(j\) denote the axis directions; \(p\) is the pressure; \(u\) is the velocity; \(\mu_t\) is the turbulent viscosity.

In the present study, the Scale-adaptive simulation (SAS) turbulence model [16,17] is selected.

2.2. Computational method

The general purpose CFD code Ansys CFX14.0 is used for numerical simulation. The total pressure at the inlet and the mass flow rate at the outlet which value is determined by the flow rate of different conditions. The time step is \(\Delta t=1.85\times10^{-4}\text{s}\). First of all, steady calculation is carried out at the design flow rate \((Q_0=25\text{m}^3/\text{h})\) and low flow rate \((0.5Q_0\text{ and } 0.3Q_0)\). Then, the results of the steady calculation are taken as the initial flow field in unsteady flow calculation. Basically, the calculations are conducted at least 12 impeller rotational periods in order to obtain a quasi-periodic solution.

2.3. Parameters and computational domain mesh of centrifugal pump

The centrifugal pump used in the present study is a conventional single suction pump with a shrouded impeller and seven twisted blades. The main parameters of centrifugal pump are listed in table 1. The computational domain including three modules, the inlet section, impeller and volute, as shown in figure 1(a). The whole passage of centrifugal pump adopts structured hexahedral meshes, the volute mesh, as shown in figure 1(b). The meshes independence has been verified by using the five mesh densities to calculate, and based on the calculate results, the meshes is selecting \(1.56 \times 10^6\) elements [18].

(a) Whole computation domain

(b)Volute mesh

Figure 1. Centrifugal pump computation domain and mesh.
Table 1. Parameters of centrifugal pump.

| Parameter                        | Value |
|----------------------------------|-------|
| Design flow rate $Q_0$ (m$^3$/h) | 25    |
| Head $H$ (m)                     | 7     |
| Rotational speed $n$ (r/min)     | 1450  |
| Number of blades $Z$             | 7     |
| Impeller outlet diameter $D_2$ (mm) | 160   |
| Wrap angle of pump               | 98°   |

3. Results and analysis

3.1. Comparison between numerical results and experimental data

The centrifugal pump head and efficiency calculation results is compared with the experimental data, as shown in figure 2. The calculated head and efficiency of the pump is comparatively good agreement with the experimental results, which demonstrates that the numerical simulations can predict the pump performance accurately.

![Figure 2. Centrifugal pump performance.](image)

3.2. Analysis of the pressure fluctuation frequency domain in volute

In the unsteady calculation, the 3 different flow rate condition $Q=1.0Q_0$, $0.5Q_0$ and $0.3Q_0$ are selected to analysis. The FFT (Fast Fourier Transform) is used to obtain the characteristics of pressure fluctuation frequency domain. The blade pass frequency is $f_{BPF}=169.17$Hz.

In order to analysis the pressure fluctuation characteristics in the pump volute, 9 monitoring points in volute are set at the middle span of the centrifugal pump and setting along the direction of flow direction, volute tongue zone (points $p_1$ and $p_2$), spiral zone (points $p_2$, $p_3$, $p_4$, $p_5$ and $p_6$), volute diffusion tube (points $p_7$, $p_8$ and $p_9$), as shown in figure 3.

![Figure 3. Monitoring points position in volute.](image)

Figure 4 shows the frequency characteristics of pressure fluctuations under $1.0Q_0$, $0.5Q_0$ and $0.3Q_0$. The pressure fluctuations of each monitoring point appears greatest peak at 169.17Hz. There are also obvious peak at 338.34Hz and 507.51Hz. The pressure fluctuation amplitude on each monitoring point is gradually decreasing with the increase of frequency.
3.3. Analysis of the maximum amplitude of pressure in volute

Figure 5 shows the maximum amplitudes of pressure fluctuations under $1.0Q_0$, $0.5Q_0$ and $0.3Q_0$. Compared with $1.0Q_0$, under low flow rate, the pressure fluctuation maximum amplitudes in the volute, in addition to point $p1$, $p7$, $p8$ and $p9$, increase basically with the flow rate decreasing. Under design flow rate $1.0Q_0$, the maximum amplitude is biggest at point $p1$ which value is 6703.4Pa. Under low flow rate $0.5Q_0$, the maximum amplitude is biggest at point $p2$ which value is 10023.6Pa, about 1.6 times of $1.0Q_0$, and at points $p3$, $p4$, $p5$, $p6$ which values are 3834.1Pa, 1265.2Pa, 549.5Pa, 1434.4Pa, about 3.6, 3.4, 1.0, 1.6 times of $1.0Q_0$, respectively. Under low flow rate $0.3Q_0$, the maximum amplitude is biggest at point $p2$ which value is 12845.4Pa, about 2.0 times of $1.0Q_0$, and at points $p3$, $p4$, $p5$, $p6$ which values are 4964.1Pa, 1582.3Pa, 1093.8Pa, 1377.3Pa, about 4.7, 4.3, 2.0, 1.6 times of $1.0Q_0$, respectively.

According to the figure 5 analysis, for 3 different conditions, the biggest pressure fluctuation maximum amplitudes appears near volute tongue zone. Under low flow rate condition, the maximum
amplitudes in volute tongue zone (point p2) and spiral zone (points p3, p4, p5 and p6) increase obviously, and especially at 0.3Q0.

Figure 6 shows the streamline distribution in pump middle span section. Under design flow rate 1.0Q0, the streamline distribution in pump, in addition to the flow separation near the blade pressure side of some flow passage, it is relatively smooth. Under low flow rate 0.5Q0, in impeller flow passage, the vortex generates, growths, and blocks some flow passage inlet. Under low flow rate 0.3Q0, in the impeller flow passage, the vortex develops, the vortex number increases, and even blocks some whole flow passage, which induces the internal flow field instabilities and the amplitude of pressure fluctuations increase in volute spiral zone.

4. Conclusion

In order to investigate the centrifugal pump internal flow field under low flow rate condition, a three-dimensional Navier-Stokes equations and SAS turbulence model has been employed to calculate. The comparison of the predicted head and efficiency between calculation results and experimental data shows a good agreement. Based on the experiment and calculation results, concluding remarks as follows are obtained.

Under 3 different flow rate 1.0Q0, 0.5Q0 and 0.3Q0, the dominant frequency of pressure fluctuation in volute are the blade pass frequency and its integer multiples, the biggest pressure fluctuation maximum amplitudes occur near the volute tongue zone.

The low flow rate condition, the intensity of pressure fluctuation gets stronger with the flow rate decreasing. Compared with design flow rate 1.0Q0, the pressure fluctuation maximum amplitudes in spiral zone increase obviously, especially at 0.3Q0, about 2~3 times of 1.0Q0.

The streamline distribution in the pump flow passage, under design flow rate 1.0Q0, it is relatively smooth; under 0.5Q0, the vortex generates, growths, and blocks some impeller flow passage inlet; under 0.3Q0, the vortex develops, the vortex number increases, and even blocks some whole impeller flow passage.

Acknowledgments

This work was supported by the Inner Mongolia Higher School Scientific Research Program [Grant number NJZY17450].

References

[1] Guo S J and Okamoto H 2003 Int. J. of Rotating Machinery 9(2) 135-144
[2] Rodriguez C G, Egusquiza E and Santos I F 2007 J. Fluids Eng. 129 1428-1435
[3] Yuan S Q, Ni Y Y, Pan Z Y and Yuan J P 2009 Chinese J. Mech Eng 22(1) 64-69
[4] Pei J, Wang W J and Yuan S Q 2014 J. Cent. South Univ. 21 1017-1024
[5] Chen K W, Lee T S, Winoto S H and Zhao R M 2007 Int. J. of Rotating Machinery 83641 1-8
[6] Lucius A and Brenner G 2011 J. Fluids Eng. 133 081102-8
[7] Brun K and Kurz R 2005 Int. J. of Rotating Machinery 2005(1) 45-52
[8] Tan L, Zhu B S, Cao S L and Wang Y M 2013 Chinese Sci Bull 58(8) 949-52
[9] Tan L, Cao S L, Wang Y M and Zhu B S 2012 Chinese Phys Lett 29(1) 014702
[10] Tan L, Zhu B S, Wang Y C, Cao S L and Gui S B 2015 Eng Computation 32(6) 1549-66
[11] Wang H and Tsukamoto H 2003 J. Fluids Eng. 125(5) 767-778
[12] Krause N, Pap E and Thevenin D 2006 Pro. of the 4th WSEAS Int. Conf. on Fluid Mechanics and Aerodynamics (Elounda, Greece, August 21-23, 2006)
[13] Tan L, Zhu B S, Cao S L, Wang Y C and Wang B B 2014 P I Mech Eng C-J Mec 228 1994-2006
[14] Duplaa S, Coutier-Delgosha O, Dazin A, Roussette O, Bois G and Caignaert G 2010 J. Fluids Eng. 132 021301-1
[15] Zhang N, Yang M G, Gao B, Li Z and Ni D 2015 J. Fluids Eng. 137(6) 061103-10
[16] Lucius A and Brenner G 2010 Int. J. Heat Fluid Flow 31 1113-1118
[17] Sedlar M, Sputa O and Komarek M 2012 Int. J. Fluid Machinery and Systems 5(1) 18-29
[18] Meng G Q Q G, Tan L, Cao S L, Wang Y C, Xu Y and Qu W S 2014 Pro. Of the 6th Int. Sym. on Fluid Machinery and Fluids Engineering (Wuhan, China, October 22-25, 2014)