Numerical Simulation of Unsteady Flow in A Centrifugal Pump

Xudan Ma ¹, Kefeng Lyu ^{2*}, Shu Zhang ¹, Ping Zhou ¹, and Lei Huang ¹

¹School of mechanical and electrical engineering, Anhui Jianzhu University, Hefei, China

²School of environmental and energy engineering, Anhui Jianzhu University, Hefei, China

Abstract. In this paper, the computational fluid dynamic (CFD) method was used to simulate the three-dimension, unsteady turbulent flow in a primary pump. Moving mesh method was used to simulate the rotation of the impeller. Sufficient monitoring points in the impeller channel, diffuser channel and interaction zone are set to get the instantaneous pressure. Fast Fourier Transformation (FFT) method was used to get the fluctuation frequency and amplitude. Steady and unsteady calculations were carried out in small and nominal flow rates. Unsteady flow in the flow channel at off design condition was revealed. The vibration frequency variation of the pump in different flow rates was analysed. The pressure fluctuation amplitude in different positions of the pump was shown. The results showed that in the clearance between impeller and diffuser, it has the largest fluctuation and higher frequencies. The fluctuation transports downstream. In off design flow rate, the pressure fluctuation is larger than in nominal flow conditions. The dominant frequency is 1 BPF in small flow rate and 3 BPF in large flow rate. The pressure fluctuation happened in the pump is caused by the stator-rotor interaction. This is clearly shown in the internal flow field.

1 Introduction

Pressure fluctuation was one of the most important characteristics of a pump. The pressure fluctuation could lead to vibration and noises. Even worse, if the pressure fluctuation frequency was harmonious to the pump assembly natural frequency, it would increase the vibration which made the pump working in an unsteady state and the life would decrease much faster. So, it was important to get to know the pressure fluctuation characteristics when designing a pump.

The pressure fluctuation caused by rotor-stator interaction is widely existed in turbine machines. This has been studied in pumps and turbines ^[1-2]. Kitano Majidi ^[3] numerically studied the pressure fluctuation in a volute pump and found that the largest pressure fluctuation happened in the outlet of the impeller. Specifically, in a diffused pump, N. Arndt, et al. ^[4-5] conducted experiments and found that fluctuating lift decreased strongly when the radial gap was increased. The close spacing of impeller and diffuser strongly increased the unsteadiness of the flow. The same conclusion was validated with CFD results by Tarek A. Meakhail, et al. ^[6]. C.G. Rodriguez, et al. ^[7] theoretically analyzed the characteristics in the frequency domain of the vibration originated in the rotor-stator interaction. And H. Wang, et al ^[8]. also theoretically analyzed the pressure fluctuation with vortex method and validate with CFD results. R.P. Dring et al ^[9] analyzed two mechanisms, the potential and the wake interaction. Akinori Furukawa et al ^[10] analyzed the inviscid flow and found that the potential interaction between the

impeller and the diffuser blades appears more strongly than the impeller-wake interaction.

In this paper, a three-dimensional pump was simulated. Pressure fluctuation characteristics were obtained. The pressure fluctuation in small flow rate condition is investigated. This condition is chosen because in the pump operation, smaller conditions are more frequently worked on.

2 Computational model

The numerical model consists of the impeller, diffuser, casing, front and back clearance, inlet and outlet pipe. The 3-D model is shown in figure 1. In order to get the pressure variation with time, several monitoring points are set. Some of the most important monitoring points are shown in figure 2.

Fig. 1. 3D model of simulation

*Corresponding author: lkf2010@mail.ustc.edu.cn

Fig. 2. Monitoring points in the axial plain

The model was meshed with ICEM. Structural grid is adopted for all the components. The mesh grid for the impeller and diffuser are shown in figure 3. In order to make the first node of the grid locate in the logarithmic layer, the boundary mesh is refined.

Fig. 3. An overall view of the structure mesh

3 Computational model

3.1 Boundary conditions

The inlet condition was set as velocity inlet and the outlet was pressure outlet. The impeller and diffuser were connected by interface. For steady state calculation, the rotation of the impeller was realized by moving reference frame (MRF) method. For unsteady state calculation, the rotation of the impeller was realized by sliding mesh (SM) method. RNG k- ϵ model was used to simulate the turbulence flow.

3.2 Numerical methods

The simulation work was conducted with Fluent 6.3, which contained plenty of physical models. SIMPLEC algorithm was used as a relationship between velocity and pressure corrections to enforce mass conservation. The second order upwind scheme was used to discrete the convective terms. The time step was set as the time when the impeller turns one third of one degree. The total computational time was set as the impeller rotates at least 3 turns.

4 Results and Discussions

4.1 Steady performance

The following equations defined the dimensionless head and flow rate parameter of the pump. In these equations, the subscript 0 stands for the head and flow rate under nominal condition.

Experiments were conducted with water in the room temperature. The steady state pump performance was compared between numerical results and experimental results, as was shown in figure 4. The numerical results showed good agreement to the experimental ones. The numerical method was reliable.

Fig. 4. Comparison between numerical and experimental results

4.2 Pressure fluctuation

The pressure fluctuations in the monitoring points were transformed using Fast Fourier Transformation (FFT). The Blade Passing Frequency (BPF) is 36.17 Hz.

The main frequencies shown inside the pump are the BPF and its higher harmonics. The pressure fluctuation is generated because of the interaction between the rotational impeller and the stationary diffuser. In the clearance between the impeller and the diffuser, the vibration amplitude is the largest. The pressure fluctuation in monitoring 1 is shown in figure 5.

Fig. 5. Pressure fluctuation in monitoring point 1 (Q_0)

To investigate the transportation of the pressure fluctuation, the first ten characteristic frequencies of different monitoring points in three flow rates are compared as shown in figure 6, 7 and 8.

Fig. 6. Pressure fluctuation in all monitoring points (0.6Q₀)

In the clearance near the impeller outlet and in the diffuser channel inlet, there also exist very large pressure fluctuation. As the pressure fluctuation transported downstream, in the outlet pipe, we get the smallest amplitude. Meanwhile, inside the clearance of the impeller and the diffuser, higher frequencies, which may be more than 10 times of the base BPF, are presented. But in the downstream positions, only low frequencies are shown. Except the pressure fluctuation in the clearance between the shroud and the casing, the hub and the casing, basically, the amplitude is decreasing in the nominal flow rate. So it is harmful to the pump operation.

Fig. 7. Pressure fluctuation in all monitoring points (Q₀)

Fig. 8. Pressure fluctuation in all monitoring points (1.4Q₀)

The flow rate influences the dominant pressure fluctuation in the clearance between the shroud and casing and the hub and casing the most. The following figure 9 shows the pressure fluctuation in monitoring 5 which located in the front clearance near the hub. In small flow rate, the dominant frequency is 1 BPF and 2 BPF. In nominal flow rate and especially in large flow rate, the dominant frequency is 3 BPF.

Fig. 9. Pressure fluctuation in monitoring point 5

The same situation also existed in the clearance of the hub and the casing. Comparing figure 9 with figure 10 and figure, in small flow rate, the dominant frequency is 1 BPF. But as the flow rate increases, large pressure fluctuation is shown and also has a large 3 BPF.

Fig. 10. Pressure fluctuation in monitoring point 2

Fig. 11. Pressure fluctuation in monitoring point 3

4.3 Internal flow field

In figure 12, it can be found that the pressure distribution changes with the different position of the impeller. In all, in the interaction place between the impeller outlet and the diffuser inlet, there is local high pressure. The high pressure is located in the outlet of the suction side of the impeller blade. When the impeller blade is near to the diffuser vane, the high pressure is intensified. In each impeller blade channel, the pressure increases from the suction side to the pressure side. While in small flow rate, the pressure in the middle of the blade channel is low and in the suction side, especially in the outlet, there is obvious low pressure field. This is the typical jet-wake phenomenon. There are flow separation and vortices in the suction side.

Fig. 12. Pressure contours comparison

This could be observed in figure 13(a). In nominal flow rate, the flow field has vortices in each of the diffuser blade channel which is shown in figure 13 (b).

Fig. 13. Stream lines

5 Conclusions

The following conclusions could be derived:

1) In the clearance between impeller and diffuser, it has the largest fluctuation and higher frequencies. The pressure fluctuation transports downstream.

2) In off design flow rate, the pressure fluctuation is larger than in nominal flow conditions. The dominant frequency is 1 BPF in small flow rate and 3 BPF in large flow rate.

3) Jet-wake phenomenon exists in the outlet of the impeller channel. The pressure fluctuation is caused by rotor-stator interaction.

ACKNOWLEDGMENT

This work was supported by the Start-up Research Program under Grant No.2018QD25 and the Anhui Natural Science Foundation of under Grant No. 1908085QE243.

References

1. A. Zobeiri, J.L. Kueny, M. Farhat, F. Avellan. *23rd IAHR Symposium*. **1** (2006).
2. Y.K. Sun, Z.G. Zuo, S.H. Liu, Y.L. Wu, J.T. Liu. *26th IAHR Symposium* (2012).
3. K. Majidi. *Proceedings of ASME Turbo Expo*. GT2004-54099 (2004).
4. N. Arndt, A.J. Acosta, C.E. Brennen, T.K. Caughey. *ASME J. Turbomach.* **111**. 213-221 (1989)
5. N. Arndt, A.J. Acosta, C.E. Brennen, T.K. Caughey. *J of Turbomach.* **112**. 98-108 (1990)
6. T.A. Meakhail, M. Salem, I. Shafie. *International J Energ Eng.* **3(2)**, 65-76 (2014)
7. C.G. Rodriguez, E. Egusquiza, I.F. Santos. *ASME J Fluid Eng.* **11(129)**. 1428-1435 (2007)
8. H. Wang, H. Tsukamoto. *ASME J Fluid Eng.* **12(123)**. 737-747 (2001)
9. R.P. Dring, H.D. Joslyn, L.W. Hardin, J.H. Wagner. *ASME J Eng Power*, **104**. 729-742 (1982)
10. A. Furukawa, H. Takahara, T. Nakagawa, Y. Ono. *Int J Rotating Mach.* **9**. 285-292 (2003)