Numerical prediction of rotating stall in a low-specific speed centrifugal pump

C Wang¹, J J Feng, X Q Luo, J L Lu and G J Zhu

¹Institute of Water Resources and Hydro-electric Engineering, Xi’an University of Technology, Xi’an 710048, China

E-mail: hi1464274105@126.com

Abstract. Rotating stall is a common unstable flow phenomenon in centrifugal pumps, which usually occurs in part-load conditions. Rotating stall will cause performance instability when the pump is running, and even excite the resonance of the whole pump system in some extreme cases. In this paper, the stall phenomena at part-load conditions have been investigated by CFD simulation for a centrifugal pump. The CFD results have been compared with the experiments, including characteristic curves, internal flow structures and corresponding frequency spectrum analyses. Furthermore, a parameter named the blockage coefficient for the pump impeller passage is put forward to quantitatively evaluate the magnitude of stall. The large-scale vortex in the stalled channel is defined as a stall cell. At the operation condition Q=0.35Q₀, five rotating stall cells can be observed in the impeller, which propagate among different impeller channels with a rotating frequency lower than that of the impeller. From the frequency spectrum analysis, the main frequency of the stall cells is about 23.4% of the impeller rotational frequency, resulting in 4.68% for each stall cell. For the operation condition of Q=0.38Q₀, a “stationary stall” phenomenon has been observed in the impeller passages.

1. Introduction

When running at part load conditions, it is easy for back flow to occur on the suction side of the impeller due to the increase of the attack angle. With the expansion of the back-flow zone, on of the channel blocked so that water flow burst into an adjacent channel. This leads to the change of the attack angle of the adjacent channel, accelerate stall cells come into being from the last channel and restrain it from the next channel. This phenomenon usually called “rotating stall” since that stall cells often rotate in a direction reverse to the impeller. Usually, there is “hump” or “positive slope” caused by rotating stall. These greatly affect the performance of a centrifugal pump. Therefore, it is significant to explore the generation mechanism and characteristics of rotating stall to restrain the generation of the stall and enhance the stability about the pump.

The theory of rotating stall was first put forward by Emmons et al.[1], who thought it is because of the boundary layer becomes thick, leading to flow separation in the suction side of the blade, then the separation zone develops from the back of the blade to another. With the rapid development of CFD and flow visualization technique nowadays, there are many researchers who have used CFD technology combined with PIV, LDV and other visual measurements to study the flow in the stall conditions. Hergtet. al. [6-9] observed that there are rotating stall cells exists when he carried out an experiment measurement of the inner flow. Sano et al.[10-12] studied by the method of experimental and numerical simulation. Then catch sight of “positive slope” in the characteristics curves of the pump.
At present, researches on rotating stall mainly concentrates on lower frequency pressure pulsation and the propagation of stall cells, but the mechanism of rotating stall has not yet been explained clearly. This paper examines the stall phenomena by numerical simulation for a centrifugal pump at part-load conditions, in order to improve the understanding of stall phenomena in pumps.

2. Modelling and boundary condition

2.1. Computational model

![Figure 1. Pump impeller](image)

The model chosen is a low specific speed centrifugal pump [14-15]. The model is consisted of a shrouded impeller and a vaneless diffuser. The inlet diameter is 103.25mm and the outlet diameter is 278mm, with 5 backward swept blades for the impellers. In order to avoid the influence of the asymmetry of the geometry, there are 12 pipes with 21.8mm diameter equally spaced in the outlet. The specific speed of the centrifugal pump is 90.9, the rotating speed \( n_0 \) is 600 r/min, and the designed flow rate \( Q_0 \) is 47.5m\(^3\)/h. Figure 1 is the structure of the impeller of the model, the blade angle at the inlet is 19 deg and 23 deg at the outlet. The red box represents the location of the camera when measured by PIV.

2.2. Mesh and boundary condition

The computational grid has a vital influence on the results of numerical simulation, the quality of the mesh determines whether the results of numerical simulation are accurate or not, in the meanwhile, the number of grids has a direct impact on the accuracy of the numerical simulation and the occupied computing resources. Thus, to ensure this simulation more accurate. In this work, a high quality structured grid is used to model the centrifugal pump model by using ICEM CFD software. In order to make rational use of computing resources and save computing time, it is necessary to verify the number of grid independence, so as to ensure the accuracy and efficiency of the entire numerical simulation process. After a grid-independent study, with the efficiency and head as the parameters, the computational grid is selected around 2.77 million for further numerical simulation, comprised of 1.8 million grids of the impeller domain and 0.97 million grids of the diffuser domain. Figure 2 shows the meshes for the impeller and diffuser.
The model pump is simulated by commercial software ANSYS CFX, a total pressure was set to be the inlet boundary condition, and the mass flow rate was regarded as outlet boundary condition then. No slip wall as wall boundary as well, steady simulations were performed with the SST model in turbulent resolving mode. The convection and space are discretized by second order accuracy. The impeller domain and diffuser domain are connected by "Transient Rotor-stator", so that the data transfer between upstream and downstream being realized. Time step in the transient simulation was set to $8.33 \times 10^{-4}$ s, it is said that the impeller rotates 3 degrees at each time step, max residual target was controlled to smaller than $10^{-3}$ as convergence criteria. In order to accelerate the convergence of simulation, the steady results under the same conditions are taken as the initial flow field for transient calculation. Whether there been a periodic change in the curve of variable monitored in the monitor points was deemed to the norm for the judging of the convergence. For the centrifugal pump discussed in this passage, it is needed to get the reliable result after 10 revolutions of simulation.

3. Results
Figure 3 shows the comparison of head curves between CFD and experiment. It can be observed that the difference between simulation results and the measuring values is less than 3%, showing a good agreement.

Figures 4 and 5 show the flow field comparison between numerical simulation and experimental measurement used TR-PIV at 0.5Q0, including stalled channel and non-stalled channel. In non-stalled channel, the streamline is smoother, but only a slight phenomenon of flow shedding occurs at the blade surface due to some random disturbances. The flow inside the passage is relatively smooth here, flow near the suction side of the blade relatively dense, the relative velocity is slightly higher than the
pressure side, and decreases gradually from the suction side to the pressure side, this is because the flow of liquid in the impeller channel is the result of the addition of the axial vortex flow and the uniform flow.

In stalled channel, there is an “8 shaped” stall cell with a large scale near the suction side of the blade, which is consist of two vortices in the same direction of flow. The presence of the stall cell results in a large area of obstruction in the channel, the back flow in the low velocity region occupies most of the whole flow passage, resulting in the flow capacity of the channel greatly weakened. Throughout the rotating stall process, along with the passage of time, the separation vortex will appear, separate and fall off periodically, which may cause many adverse effects on the performance of centrifugal pumps.

![Figure 4](image-url)

**Figure 4.** Non-stalled channel at 0.5Q₀

![Figure 5](image-url)

**Figure 5.** Stalled channel at 0.5Q₀

![Figure 6](image-url)

**Figure 6.** Frequency spectra
The time signals of the velocity fluctuation measured in monitor points are converted into frequency signals transformed by FFT. From the frequency diagram shown in Figure 6, we can see that peaks appear at low frequencies, from P1 to P5, the peaks at each point appears at $f_s=2.34\text{Hz}$ but a slightly difference in amplitude, this is due to the dissipation of energy associated with the propagation of the stall cells. Compared with the field getting in the simulation, it can be determined that the dominant frequency is analysed in frequency spectrum reaches the pulsation caused by the relative motion between stall cells and the impeller, so it is concluded that the $f_s$ here is the characteristic frequency of the stall cells. The frequency of stall cells measured by PIV experiment is $2.25\text{Hz}$, which is very close to the numerical results. The rotation frequency of the model pump is $10\text{Hz}$, about $4.27$ times of the frequency of the stall cell, this is to say that while the impeller rotates about $4.27$ times, there’s a stall cycling completed in a blade passage.

Stall cells distribution and development in the impeller at $0.35Q_0$ are shown in Figure 7. From the figure, it is apparent that there are 5 cells in the impeller, and each of them presents vary. The stall is propagated between the blade passages at a rotational speed lower than the impeller. The stall frequency is $2.34\text{Hz}$, which is $23.4\%$ of the impeller. Therefore, the rotating angular velocity of the stall is about $4.68\%$ of the impeller speed. As a result, the motion of the cells in the channel is reversed relative to the impeller. It can be observed in Figure 7(a) that there’s a stall cell consists of two small-scale vortices in channel 2. When $t=t_0+T$, two vortices fuse gradually, merge to a large area of silt eventually, occupying the whole passage at the time of $t_0+2T$. After another rotation period, as shown in Figure 7(d), the stall cell developed into channel 1, and there’s a slight flow separation on suction side of channel 2, next stall period starts. It can be concluded from above results that the stall cells generate in the suction side of the blade, growing and approaching the pressure side gradually, thereby block up the whole channel entirely and then transform into the other channel. A large area of low velocity appears in the passage due to the excessive flow rate in this process, the downstream flow will be driven by the rotation of the upstream flow, so that the outlet of the flow path will produce a vortex with the opposite direction of rotation, which is called "exit vortex"[16]. The exit vortex is very close to the pressure side of the blade, which often results in the direct impact of the water on the surface of the blade, resulting in the reduction of hydraulic efficiency and the aggravation of vibration.

![Figure 7. Stall cell distribution in impeller at 0.35Q₀](image)
The phenomena of “stationary stall” is observed at 0.38Q0 shown in Figure 8. There are 4 stall cells which are distributed in channel 1, channel 2, channel 3 and channel 5. During the pump rotation, the relative position between the stall cells and the impeller remains nearly unchanged.

![Stall cell distribution in impeller at 0.38Q0](image)

**Figure 8.** Stall cell distribution in impeller at 0.38Q0

4. Conclusions

In this paper, the stall phenomenon in a centrifugal pump impeller is numerically simulated by CFD, and the results are compared with the experimental measurements. The main conclusions are summarized as follows:

There are 3 stall cells in 0.5Q0 and 4 stall cells in 0.38Q0 in the impeller. The stall cells are relatively stationary with the impeller, which do not propagate between different passages, a “stationary stall” phenomena aroused. Compared with rotating stall, stationary stall is a much more stable phenomenon.

Under condition of 0.35Q0, the stall cells propagate between different channels. The dominant frequency of the stall cells is 2.34Hz, about 23.4% of the impeller frequency. The rotation speed of the stall cells is about 4.68% because there are 5 stall cells here in the impeller. Stall cells generate from the suction side of the impeller, developing and propagating in the impeller passage with time.

Acknowledgements

This research was supported by the National Natural Science Foundation of China (Grant nos. 51679195, 51339005 and 51479167).

References

[1] Emmons H W, Pearson C E and Grant H P 1955 Compressor surge and stall propagation *Transactions of ASME* volume 77 issue 3 pp 455-469.
[2] Ullum U, Wright J and Dayi O 2006 Prediction of rotating stall within an impeller of a centrifugal based on spectral analysis of pressure and velocity data *Institute of Physics Publishing, Journal of Physics: Conference series* volume52 issue1 pp36-45.

[3] Liu T, Zhang Y N, Du X Z 2015 A review of rotating stall of pump turbines *Journal of Hydroelectric Engineering* volume34 issue11 pp16-24.

[4] Yang J, Yuan S Q and P J 2015 Overview of rotating stall in centrifugal pumps with vaned diffuser *Journal of Drainage and Irrigation Machinery Engineering* volume33 issue5 pp369-373.

[5] Zhang N, Yang M G and Gao B 2015 Unsteady rotating stall characteristics in a centrifugal pump with slope volute at low flow rates *Journal of Vibration and Shock* volume34 issue6 pp189-194.

[6] Hergt P, Starke J 1985 Flow patterns causing instabilities in the performance curves of centrifugal pumps with vaned diffusers *The second Int. Pump Symp* Texas USA pp67-75.

[7] Yoshida Y, Murakami Y, and Tsurusaki T 1991 Rotating stalls in centrifugal impeller/vaned diffuser systems *First ASME/JSME Joint Fluids Engineering Conference* Portland USA pp125-130.

[8] Saha S L, Kurokawa J and Matsui J 2000 Suppression of performance curve instability of a mixed flow pump by use of J-groove *ASME Journal of Fluids Engineering* volume122 pp592-597.

[9] Sinha M, Pinarbashi A, Katz J 2001 The flow structure during onset and developed states of rotating stall within a vaned diffuser of a centrifugal pump *ASME Journal of Fluids Engineering* chapter123 pp490-499.

[10] Sano T, Yoshida Y and Tsujimoto Y 2002 Numerical study of rotating stall in a pump vaned diffuser *Journal of Fluids Engineering* volume124 pp363-370.

[11] Liu H L, Yang D S and Tan M G 2012 3D-PIV Measurements of stall in Double-Blade centrifugal pump *Journal of Shanghai Jiaotong University* volume46 issue05 pp734-739.

[12] Johnson D A, Pedersen N and Jacobsen C B 2005 Measurements of Rotating Stall Inside a Centrifugal Pump Impeller *ASME Fluids Engineering Division Summer Meeting and Exhibition* Houston USA.

[13] Pan Z Y, Li J J and Li H 2011 Overview for research on rotating stall of pump *Fluid Machinery* volume39 issue2 pp35-39.

[14] Krause N, Zhhringer K, Pap E 2005 Time-resolved particle imaging velocimetry for the investigation of rotating stall in a radial pump *Experiments in Fluids* volume39 issue2 pp192-201.

[15] Lucius A, Brenner G 2010 Unsteady CFD simulations of a pump in part load conditions using a scale-adaptive simulation *International Journal of Heat and Fluid Flow* volume31 issue6 pp1113-1118.

[16] Zhou P J, Wang F J and Yao Z F 2015 Investigation of Pressure Fluctuation in Centrifugal Pump Impeller under Rotating Stall Conditions *Transaction of the Chinese Society for Agricultural Machinery* volume46 issue10 pp56-61.