A Study of Stall Condition of a Centrifugal Compressor based on CFD and Experimental Methods

R. Wang\textsuperscript{a,1}, H. L. Wang\textsuperscript{a}, S. H. Yang\textsuperscript{a,b,*}, J. J. Tan\textsuperscript{a}, S. Jiang\textsuperscript{a}, J.Q. Ren\textsuperscript{a}

\textsuperscript{a}. Shenyang Blower Works Group Corporation, Shenyang, China
\textsuperscript{b}. School of Energy and Power Engineering, Dalian University of Technology, Dalian, China
\textsuperscript{1} These authors contributed equally to this work.

\textsuperscript{*} wangruiauthor@aliyun.com (R. Wang), yangshuhua@shengu.com.cn (S. H. Yang)

Abstract. The stall point of a centrifugal compressor is obtained by CFD method and FFT method, and an experiment in the stall point is carried out. Firstly, the curves of the compressor efficiency and total pressure ratio with flow rate are obtained by the steady CFD analysis. Secondly, according to the curves, the unsteady CFD analysis and frequency signal analysis are carried out at the operation points where stall may occur. Then the stall point of the compressor is obtained. On the basis of the results of numerical analysis, the model machine is used to carry out the test. By comparing the results of the numerical method and the experimental method, the stall is identified at the similar operation point, which proves the effectiveness of the prediction method in this paper.

1. Introduction

In a compressor, stall can be regarded as the cessation of a continued rise in static pressure recovery in a diffuser or cascade. The blade passages may stall partially or fully. If mass flow continues to reduce, it can turn to the surge and causes audible flow fluctuations\cite{1}. Cheshire\cite{2} found surge in centrifugal compressors. Emmons\cite{3} confirmed that rotating stall was a precursor to surge, and made clear the difference between the two phenomena through experiments. Moore and Greitzer\cite{4-6} proposed a speed disturbance model to predict the rotating stall, and used active control to suppress the disturbance before stall. Fringe\cite{7} found two stalls at different frequencies with experiments, which correspond to impeller stalls and diffuser stalls respectively. Based on the dynamic model proposed in \cite{4-6}, Liaw\cite{8} performed the bifurcation analysis and proposed an active control technique in axial flow compression system. Tryfonidis\cite{9} used nine high-speed compressors for research and found that there were traveling waves that developed in the circumferential direction before stall initiation. In literatures\cite{10-11}, researchers performed study about the spike type rotating stall and explain the physical mechanism. The rapid development of computer technology has made CFD become one way for studying stall phenomena. Spentzos\cite{12} used 2D and 3D CFD to investigate the dynamic stall of square wings of NACA 0012 and NACA 0015. Sun\cite{13} presented a stall inception model for transonic fan/compressors under some assumptions. By comparing numerical and experimental results, the model can give a reasonable prediction to the stall inception for both subsonic and transonic fan/compressor. In literatures\cite{14-16}, CFD and experiments were also applied to study stall at different places of flow channel of centrifugal compressors, respectively. The tests performed by
Day\textsuperscript{[17]} on Cambridge 4-stage axial compressor showed that for the same type of compressor, the rotation speed of the stall cell has a constant proportional relationship with the rotation speed of compressor during stall. Zhao\textsuperscript{[18]} performed the tests for the investigation of blade vibration induced by stall and observed the signal modulation phenomenon. Generally, the eccentricity of the rotor is not taken into consideration in CFD simulation process. Therefore, the pressure pulsation obtained by CFD cannot identify the frequency corresponding to the rotation speed ($f_r$) through FFT. However, due to the effects of rotation and fluid-structure interaction, the characteristic frequencies corresponding to the stall can still be obtained in the low frequency region and near the blade passing frequency. These frequencies are the frequency of the stall cell ($f_u$), the rotation frequency of the stall cell ($f_{BPF}$ or $f_{BPF} - f_u$) and the blade passing frequency ($f_{BPF}$). The steady and unsteady CFD simulations of a centrifugal compressor are performed to analysis the operation point where stall may occur in this paper. On the basis of the numerical results, at similar operation point, a series of comparative experiments were carried out using a centrifugal compressor. The results show that both the numerical method and the experimental method obtained the stall phenomena at the similar operation point.

2. CFD Simulation

The research object is a centrifugal compressor with vaned diffuser in this paper. In numerical modeling, the inlet and outlet of the channel were extended appropriately. The calculation domain includes an impeller, a vaned diffuser and a vaned return channel, shown as Fig. 2.1. The diameter of impeller is 500mm. The blade numbers of the impeller, diffuser and return channel are 19, 13 and 24, respectively.

![Fig. 2.1. The 3D model of the centrifugal compressor](image)

The IGG module which belongs to the NUMECA software was applied to generate full-channel structured grid. The partial enlarged images of the grids are shown as Fig. 2.2, and the total number of grids is $1.508 \times 10^7$. The thickness of the first layer of grid on the wall is 0.01mm to ensure that most of Yplus values are less than 10. The density of the grids on both sides of all moving-static interfaces are properly increased. This can prevent the grid size difference on both sides of the interfaces from becoming too large and causing unnecessary interpolation errors.

![Fig. 2.2. The mesh of the calculation domain](image)

The FINE/Turbo suite of Numeca software are used to perform the steady and unsteady simulation. The full-channel model are adopted during the CFD simulation. The Spallart-allmaras (SA) turbulence model is employed. The conditions of total pressure and total temperature are set at the inlet, and axial
air intake is set to uniform. The condition of mass flow is set at the outlet. The working fluid is ideal air. The boundary conditions at design point are shown as Table 2.1. The total pressure ratio and the polytropic efficiency in this paper are shown as dimensionless values relative to the design points. The area definition method is applied to set the rotating wall in the impeller. The adiabatic and non-slip conditions are applied for the wall. In the steady calculation, the frozen rotor method is selected between the moving and stationary domains for upstream and downstream data transmission. While for the unsteady case, transient rotor-stator interface option is adopted.

**Table 2.1.** The boundary conditions at design point

| Fluid       | Total Pressure (Pa) | Total Temperature (K) | Rotation Speed (r/min) | Mass Flow (kg/s) |
|-------------|---------------------|-----------------------|------------------------|------------------|
| Ideal Air   | 101000              | 293                   | 5911.2                 | 2.55             |

### 2.1. Steady Simulation

First, the steady CFD method is used to obtain the centrifugal compressor performance curve, including total pressure ratio and polytropic efficiency versus inlet mass flow. The maximum mass flow is 3.57kg/s. Then reduce the mass flow for calculation, constantly. The minimum mass flow is 1.79kg/s. The dimensionless results of total pressure ratio and polytropic efficiency are shown as Fig. 2.3. The horizontal axis is the mass flow in Fig. 2.3. The two curves in Fig. 2.3 are approximately parabolic with an opening downward. The maximum value of the total pressure ratio appears when the mass flow is 1.98kg/s. While the maximum value of the polytropic efficiency is at the mass flow of 2.3kg/s. As the mass flow decreases, the operating point at which total pressure ratio begins to decrease is selected as the suspected stall point. The mass flow at the suspected stall point is 1.98kg/s. Then the unsteady CFD simulations are performed in the section 2.2 with five mass flow values are 1.79, 1.85, 1.91, 1.98 and 2.04kg/s, respectively.

![Fig. 2.3.](image.png)

**Fig. 2.3.** The dimensionless results of total pressure ratio and polytropic efficiency of the steady CFD

### 2.2. Unsteady Simulation

According to the results of the steady method, five operating points are chosen to implement the unsteady simulations. During the simulation processes, 19 time steps are set for each moving blade channel. Meanwhile, set 50 iterations in each time step for better convergence. The streamline diagrams in the Meridian flow channel with five mass flows are shown in Fig. 2.4. It can be seen that when the mass flow is reduced to 1.98kg/s, two blue separation zones will appear at the impeller inlet and in the diffuser. When the mass flow is further reduced, cells will generate in the above two separation areas, and the volumes of the cells will gradually increase. On the basis of the results of the section 2.3, the rotation stall can be observed firstly when mass flow is 1.91kg/s. The results of the unsteady CFD simulation at mass flow is 1.91kg/s will be taken as an example. The partial enlarged views of the streamline when mass flow is 1.91kg/s are shown as Fig. 2.5.
According to the positions of the two cells, the velocity and streamline diagrams are taken at the 95% and 5% blade height sections with mass flow is 1.91kg/s, respectively, as show in Fig. 2.6. The meridian velocity and the spatial streamline are shown as Figs 2.6(a, b), and the relative velocity vectors are shown as Figs 2.6(c, d). It can be seen from Figs. 2.6(b) that there are clearly cells near the shroud of the impeller. At this mass flow, the separation of the flow field has occurred, and one location of the stall cells are generated. The location is near the suction surface of each blade channel which shown as Figs. 2.6(a, b). At the location which is at the inlet and the outlet of each diffuser blade which shown as Fig. 2.6(d) in red circle, the attack angle of the flow is not good for working and low speed locations are obviously.
In general case, when a rotating stall occurs, the stall cells propagate between different impeller channels. The entropy diagrams at the 95% blade height are shown as Fig. 2.7 when the mass flow is 1.91kg/s. There are four times in the Figs 2.7, respectively. The time range is the time for the impeller to sweep through one blade passage. The four times are evenly distributed. In the section 2.3, the unsteady CFD pressure results at five operation conditions are processed by the FFT method, and the compressor rotating stall phenomenon is studied in the frequency domain.

2.3. The Analysis of the Unsteady CFD Results in Frequent Domain

Seven monitoring sections (S1 to S7) were set along the flow direction in the calculation domain, respectively. The locations of these sections are shown as Table 2.2 and Fig.2.8.

| Section | Location                  |
|---------|---------------------------|
| S1      | Inlet (in rotating domain)|
| S2      | Diffuser inlet (Impeller outlet) |
| S3      | Diffuser blade            |

Table 2.2. The locations of monitoring sections
In the calculation process, the convergence or not is determined by monitoring the efficiency, total pressure ratio, inlet and outlet flow, and pressure pulsation of each monitoring point. The pressure pulsation with mass flow is 1.98 kg/s of the S2 section and S7 section in time domain are shown in Fig. 2.9. The pressure pulsation is periodic.

The FFT method is applied to process the pressure pulsation results of the monitoring points. The results of the last 2048 time steps are chosen as samples for the FFT method in each operation condition. The results of the monitoring points with the five mass flows in frequency domain are shown in Figs. 2.10-2.14, respectively. The abscissa is frequency and the ordinate is pressure in these figures. In general case, the eccentricity of rotor is not taken into account during the CFD simulation. Thus, the frequency of rotation speed and the modulation frequency near it cannot be observed in the results of CFD simulation. The following conclusions can be obtained from Figs. 2.10-2.14:
1) The blade passing frequency \( f_{\text{BPF}} = 1870.3 \text{Hz} \) and its second order frequency \( 2 \times f_{\text{BPF}} = 3470.7 \text{Hz} \) can be observed in all five operation conditions at the Section 2 to Section 7;
2) There are \( f_{\text{BPF}} \) and modulation frequencies \( f_{\text{mod}} = 1818.4 \text{Hz} \) or \( f_{\text{BPF}} + f_{\text{c}} = 1922.3 \text{Hz} \) in the Figs. 2.12-2.14 when the mass flow is less than or equal to 1.91 kg/s;
3) There are same peaks \( f_{\text{c}} = 52 \text{Hz} \).

According to the results of the CFD method and the literatures, the rotational stall can be identified when mass flow is 1.91 kg/s.
Fig. 2.10. The results of FFT at the seven sections with mass flow is 2.04 kg/s
Fig. 2.11. The results of FFT at the seven sections with mass flow is 1.98kg/s
Fig. 2.12. The results of FFT at the seven sections with mass flow is 1.91kg/s
Fig. 2.13. The results of FFT at the seven sections with mass flow is 1.85kg/s
Fig. 2.14. The results of FFT at the seven sections with mass flow is 1.79kg/s
3. Experimental Investigation
On the basis of the results of the CFD simulation, a centrifugal compressor was used to carry out the experiments with the range of the mass flow is [1.638, 3.661] kg/s. Experiment details are introduced as the following part.

3.1. Experiment Equipment
The model machine used in the experiments is a one stage centrifugal compressor. The experiment device layout and the model machine are shown as Fig. 3.1, and the arrows in the figures point to the direction of the air flow. At the outlet of the impeller, that is, the inlet of the diffuser, a pressure sensor is installed. The sensor model is 116B pressure sensor of the PCB Piezotronics, Inc. The sampling frequency was set to 51.2kHz. In each operation point, 40s data is collected each time, twice in total. Meanwhile the experiment parameters, including inlet pressure, outlet pressure, inlet temperature, outlet temperature, and mass flow, are monitored. The parameters which have been used in experiments are list in the Table 3.1.

![Fig. 3.1. Experiment device](image)

| No. | Inlet Pressure (Pa) | Outlet Pressure (Pa) | Inlet Temp. (K) | Outlet Temp. (K) | Mass Flow (kg/s) |
|-----|---------------------|----------------------|----------------|----------------|-----------------|
| 1   | 98688               | 105750               | 295.8          | 310.3          | 3.661           |
| 2   | 99316               | 113904               | 296.3          | 312.5          | 3.001           |
| 3   | 99740               | 116817               | 296.5          | 313.6          | 2.509           |
| 4   | 99981               | 118219               | 296.7          | 314.7          | 2.159           |
| 5   | 100048              | 118573               | 296.8          | 315.2          | 2.040           |
| 6   | 100118              | 118779               | 297.0          | 315.8          | 1.916           |
| 7   | 100198              | 118867               | 297.1          | 316.4          | 1.798           |
| 8   | 100298              | 118643               | 297.1          | 317.2          | 1.638           |

The dimensionless results of total pressure ratio and polytropic efficiency based on the steady CFD method and experimental method are shown in Fig. 3.2, respectively. It can be seen from Fig. 3.2 that the steady CFD results are in good agreement with the experimental results. It can be proved that the CFD simulation is credible.
Fig. 3.2. The dimensionless results of the CFD method and the experimental method

3.2. Experiment Results Analysis in Frequent Domain

The results in frequent domain by the FFT method of the eight operation points are shown as Fig. 3.3, respectively.

(a) The results of FFT with mass flow is 3.661kg/s
(b) The results of FFT with mass flow is 3.001kg/s
(c) The results of FFT with mass flow is 2.509kg/s
(d) The results of FFT with mass flow is 2.159kg/s
The results of FFT with mass flow is 2.04kg/s

The results of FFT with mass flow is 1.916kg/s

The results of FFT with mass flow is 1.798kg/s

The results of FFT with mass flow is 1.638kg/s

The following conclusions can be obtained from Fig. 3.3:
1) The blade passing frequency ($f_{BPF}=1871.19$Hz) and its second order frequency ($2\times f_{BPF}=3742.48$Hz) can be observed in all eight operation conditions;
2) The rotational frequency ($f_r=98.44$) and its higher order frequencies can be observed in each operation condition. Especially when the mass flow is 1.916, 1.798 and 1.638kg/s, the amplitudes of the first three-order rotational frequency are more significant than that of the higher-order frequencies. It can be inferred that the compressor rotor has poor alignment.

In addition to the above conclusions, the part of Fig. 3.3(g) is enlarged, as shown in Fig. 3.4.

Although there are many peaks that are difficult to distinguish in the Fig. 3.4, several characteristic peaks can still be identified.
1) There are $f_{BPF}$ and a modulation frequency ($f_{BPF}-f_s=1834.38$Hz) in the Fig. 3.4(b);
2) There are $f_r$ and a modulation frequency ($f_r-f_s=52.25$Hz) in the Fig. 3.4(a).
3) There is a peak of $f_s=46$Hz. According to the literatures, the rotational stall can be identified when mass flow is 1.798kg/s. Moreover, the part of Fig. 3.3(h) is enlarged, as shown in Fig. 3.5.

![Fig. 3.5. The results of FFT with mass flow is 1.638kg/s](image)

A series of frequencies can be gained. They are 10.35, 20.8 and 31.15Hz, respectively. The 10.35Hz is about $0.1 \times f_r$. The phenomena of the compressor surge has been observed with mass flow is 1.638kg/s. According to the results of the experiments, the compressor surge phenomenon was observed in the frequency domain. The results of the CFD method in this paper cannot observe the surge phenomenon. On the one hand, if continue to reduce the mass flow, which is less than 1.79kg/s, during the unsteady CFD simulation process in this paper, the periodicity of the pressure pulsation cannot be observed. Therefore, it cannot be judged whether it has converged. On the other hand, the compressor surge caused by the flow state of the complete compressor system. The model for the simulation of the CFD method in this paper is focused on the core parts, including impeller, diffuser and return channel. The exhaust volute and downstream piping system have not been taken into account.

4. Conclusions

The stall condition of a centrifugal compressor based on CFD and experimental methods has been investigated. The steady and unsteady CFD simulations has been performed. A series of experiments has been carried out. The following conclusions can be drawn:

1) By comparing the results of the total pressure ratio and polytropic efficiency of the CFD and the experimental methods, it can be proved that the CFD model used in this paper can be performed for the simulation of the model machine.

2) Both the CFD method and the experimental method observe the rotating stall phenomenon in frequency domain. For CFD, the stall appears at flow rate 1.91kg/s, the corresponding $f_s$ is 52Hz. For experiments, the stall appears at flow rate 1.798kg/s, the corresponding $f_s$ is 46Hz. There are some reasons for this deviation. For instance, the exhaust volute, clearances, and surface roughness were not taken into consideration during the process of the numerical simulations.

5. Acknowledgment

The authors are grateful for support from the project of new energy-saving and system intelligent control technology for fluid machinery, the National Key R&D Program of China (Grant No. 2018YFB0606105).

References

[1] R. C. Pampreen (1993). *Compressor Surge and Stall*. Concepts ETI, Inc. Norwich, Vermont, USA.

[2] L. J. Cheshire, M. S. Tech, A. M. I. Mech (1945). *The Design and Development of Centrifugal Compressors for Aircraft Gas Turbines*. Proceedings of the Institution of Mechanical Engineers,
153, pp 426-440.

[3] H. W. Emmons, C. E. Pearson, and H. P. Grant (1955). Compressor surge and stall propagation. Transactions of the ASME, 79, pp 455–469.

[4] E. M. Greitzer (1976). Surge and Rotating Stall in Axial Flow Compressors-Part II: Experimental Results and Comparison with Theory. Journal of Engineering for Power, 98(2), pp 199-211.

[5] F. K. Moore, E. M. Greitzer (1986). A Theory of Post-Stall Transients in Axial Compression Systems: Part I-Development of Equations. Journal of Engineering for Gas Turbines and Power, 108(1), pp 68-76.

[6] F. K. Moore, E. M. Greitzer (1986). A Theory of Post-Stall Transients in Axial Compression Systems: Part II-Application. Journal of Engineering for Gas Turbines and Power, 108(2), pp 231-239.

[7] P. Frigne, R. V. D. Braembussche (1984). Distinction between Different Types of Impeller and Diffuser Rotating Stall in a Centrifugal Compressor with Vaneless Diffuser. Journal of Engineering for Gas Turbines and Power, 106(2), pp 468-473.

[8] D-C. Liaw, E. H. Abed (1996). Active control of compressor stall inception: a bifurcation-theoretic approach. Automatica, 32(1), pp 109-115.

[9] M. Tryfonidis, O. Etchevers, J. D. Paduano, A. H. Epstein, G. J. Hendricks (1995). Prestall Behavior of Several High-Speed Compressors. Journal of Turbomachinery, 117(1), pp 62-80.

[10] T. R. Camp, I. J. Day (1998). A study of spike and modal stall phenomena in a low-speed axial compressor. Journal of Turbomachinery, 120(3), pp 393-401.

[11] G. Pullan, A. M. Young, I. J. Day, E. M. Greitzer, Z. S. Spakovszky (2015). Origins and Structure of Spike-Type Rotating Stall. Journal of Turbomachinery, 137(5), pp 2567-2579.

[12] A. Spentzos, G. Barakos, K. Badcock (2004). CFD investigation of 2d and 3d dynamic stall. The AHS 4th Decennial Specialist’s Conference on Aeromechanics, San Fransisco, California, January, pp 21-23.

[13] Sun X. F., Sun D. F., Yu. W. W (2011). A Model to Predict Stall Inception of Transonic Axial Flow Fan/Compressors. Chinese Journal of Aeronautics, 24(6), pp 687-700.

[14] J.N. Everitt, Z.S. Spakovszky (2013). An Investigation of Stall Inception in Centrifugal Compressor Vaned Diffuser. Journal of Turbomachinery, 135(1), 011025.

[15] A. Broatch, J. Galindo, R. Navarro, J. García-Tiscar (2016). Numerical and experimental analysis of automotive turbocharger compressor aeroacoustics at different operating conditions. International Journal of Heat and Fluid Flow, 61(B), pp 245-255.

[16] N. Fujisawa, T. Inui, Y. Ohta (2019). Evolution Process of Diffuser Stall in a Centrifugal Compressor With Vaned Diffuser. Journal of Turbomachinery, 141(4), 041009.

[17] I. J. Day (1992). Stall and Surge in Axial Flow Compressor. Axial Flow Compressor Lecture Series. Von Karman Institute for Fluid Dynamics.

[18] Zhao X. W., Zhou Q., Yang S. H., Li H. K (2109). Rotating Stall Induced Non-Synchronous Blade Vibration Analysis for an Unshrouded Industrial Centrifugal Compressor. Sensor, 19(22), 4995.