Numerical study on the influence of the outlet width of vaneless diffuser on the performance of centrifugal compressor

Linfeng Du *, Yupeng Pan
Hangzhou Dianzi University, Hangzhou, China

*Corresponding author e-mail: 916428107@qq.com

Abstract. In this paper, the 3D model of the centrifugal compressor is established by Solidworks software at first. Second, the numerical simulation of the centrifugal compressor is carried out by using CFX software. Combined with the simulation results, the conclusion that properly reducing the outlet width of diffuser can effectively restrain the generation of flow separation and reduce the influence of impeller rotation on diffuser is revealed. Finally, the internal flow field structure and machine performance of the diffuser are improved. The research provides a reference for improving the performance and flow field structure of the compressor as well as the design of the vaneless diffuser.

1. Introduction
Centrifugal compressor, as an important turbomachinery, has been widely used in various fields closely related to the national life and economy, and is the key equipment to the production device. Meanwhile, diffuser is an important part of centrifugal compressor, which plays an irreplaceable role. The main principle is to increase the pressure of compression medium and reduce the speed of the medium through the expansion of its flow channel. Vaneless diffuser, as a kind of diffuser with simple structure, has good adaptability and wide working range, therefore it is widely used in compressors [1]. The vaneless diffuser is divided into contraction type, parallel type and dilated type according to the width difference between outlet and inlet. However, the dilated vaneless diffuser is prone to flow separation, which results in flow loss, therefore it is rarely used, while parallel type and shrink type are more used in compressors.

In the research of vaneless diffuser, domestic scholars have made a lot of efforts. In 1990, Huashu DOU [2] proposed the design principle of non-parallel wall of vaneless diffuser. In 2004, Mozi GAO et al. [3] have a method that reduces the width of diffuser by experiment, which can make the tangential velocity distribution of internal flow in diffuser more uniform, as well as shorten the flow path and reduce the friction loss. In 2008, Limin GAO et al. [4] studied the flow field of centrifugal compressor, and investigate the influence of straight-wall diffuser and equal-area diffuser on the flow field and aerodynamic performance of centrifugal compressor. It was found that the effect of equal-area vaneless diffuser on centrifugal compressor is small, while the flow separation of straight-wall diffuser occurs in the diffuser. In addition, Chao MA et al. [5] analyzed the influence of diffuser shrinkage angle on centrifugal compressor, and revealed that the larger shrinkage angle can improve the flow field in the downstream region of impeller, increase the radial velocity, and weaken the flow separation of suction surface, simultaneously, the reflux was effectively inhibited. Lixin MO et al. [6] studied the inlet shape
of diffuser in 2012, and described that the efficiency of diffuser can be effectively improved by modifying the shape of diffuser inlet. In 2017, Songling WANG et al. [7] studied the radius ratio and internal flow of diffuser, and found that the change of radius ratio did not affect the stall mechanism of compressor, but only affected the critical flow rate and critical flow angle when stall occurred.

In recent years, with the continuous improvement of computer performance and the rapid development of computer technology, as well as the progress of computational fluid dynamics (CFD) software and numerical simulation methods, modern CFD technology has become an important way to study centrifugal compressors [8-11].

From the above, it can be seen that some parameters such as diffuser width, diffuser shrinkage angle, diffuser shape, inlet shape and radius ratio have a certain impact on the performance of the compressor. However, the change of the diffuser width the whole parallel movement, and the study of the shrinkage angle are changed by extending the straight wall of the diffuser. At present, there is a little research on the performance and internal flow field of contractile diffuser by modifying the outlet width of diffuser, which needs to be further studied. Therefore, in view of this problem, this paper uses CFX commercial software to carry out numerical simulation, and analyzes the influence of diffuser outlet width on compressor performance and flow field in detail.

2. Model and Numerical Method of Calculation

2.1. Establishment of Model and Division of Mesh
In this paper, the model of the centrifugal compressor is established by Solidworks software, saved as IGES format and imported into Pointwise software to generate mesh. The model structure is shown in Figure 1, and the main structural parameters are shown in Table 1. When the mesh is divided, the model is divided into two parts, namely impeller and rear. The mesh uses the type of mixing of structure and non-structure, and properly encrypts the wall surface. In order to reduce the computational complexity, a single channel meshing type is chosen to mesh the model. As a result, 1.96 million meshes are generated. The mesh division is shown in Figure 2.

| Parameter                          | Numerical Value |
|-----------------------------------|-----------------|
| Inlet Temperature (K)             | 293             |
| Inlet Pressure (kPa)              | 101             |
| Design flow (kg/s)                | 5.6             |
| Rotating speed (rpm)              | 10175           |
| Impeller outlet diameter (mm)     | 450             |
| Impeller outlet width (mm)        | 42              |
| Diffuser outlet diameter (mm)     | 698             |
| Number of impeller blades         | 19              |
| Number of return nozzles          | 18              |

Table 1. Table of Main Structural Parameters of Centrifugal Compressor

Figure 1. Centrifugal Compressor Prototype Structure Diagram
2.2. Numerical Calculation Method
The turbulence model used in this paper is the SST model, which can accurately predict the characteristics of flow separation and has higher accuracy than other models [12]. The specific boundary condition settings are as follows:

1. The impeller part is set to the rotation domain, and the rotation speed is set to 10175 rpm, relatively, the rear part is the static region, and the working medium is the ideal air.
2. Set the total temperature and pressure of the inlet, $T_{in}=293K$, $P_{in}=101325kPa$.
3. Set the mass flow condition of the outlet.
4. The boundary conditions for setting the solid wall surface are non-slip and adiabatic.

2.3. Mesh Independent Verification
In order to reduce the influence of mesh on the numerical simulation, an independent verification of the single flow channel of the centrifugal compressor is carried out, and the mesh numbers of 1.51, 1.96 and 2.3 million are selected for numerical simulation at the design point. The results are shown in Figure 3.

![Figure 3. Grid Independence Verification Result Graph](image)

It can be seen that the performance change of centrifugal compressor can be ignored from the mesh number of 1.51 million. Therefore, the intermediate point 1.96 million is chosen as the mesh number of centrifugal compressor model.

2.4. Numerical Scheme
The outlet of diffuser is reduced to 41mm, 39mm, 37mm, 35mm and prototype 42mm respectively. When the geometric model is established, the outlet of diffuser and impeller are connected by straight line. When the diffuser outlet width is 35mm, the meridian channel diagram is shown in Figure 4.
3. Simulation Results Analysis

3.1. Flow field structure

The radial velocity distribution inside the original wide diffuser at the designed flow rate is shown in Figure 5, in which (a), (b) and (c) represent the radial velocity distribution at different blade heights respectively, 0% blade height is disk side, and 100% blade height is cover side. Meanwhile, the meridian flow chart of the centrifugal compressor is shown in Figure 5(d). It can be seen from Figure 5 (a-c) that there is a small portion of the low-speed fluid region at the diffuser inlet, while the other regions are relatively high-speed. The main reason for this phenomenon is that the internal flow at the inlet of the diffuser is also greatly influenced by the centrifugal impeller, and a small part of the fluid maintains the flow characteristics of the inner fluid of impeller. As a result, there is a little backflow when the fluid just enters the diffuser. After entering the diffuser, the low-speed fluid area disappears, and the radial velocity of fluid in the diffuser decreases gradually. When the fluid gets close to the outlet, the radial velocity is less than 0 at the place where the diffuser blade height is 97%, while the deceleration at 50% and 3% blade height is relatively smooth. It can be seen from Figure 5 (d) that there is an obvious flow separation near the cover at the diffuser outlet, and the flow separation extends to the first half of the curve.

(a) 3% blade height   (b) 50% blade height
Figure 5. Velocity distribution of original wide diffuser

At the designed flow rate, the radial velocity diagram of 97% blade height at different outlet widths is shown in Figure 6. And the corresponding meridian flow chart is shown in Figure 7. Figure 7(e-h) are enlarged views of the diffuser outlet. It can be seen from Figure 6 (a) and Figure 7 (a) that after reducing the width of diffuser outlet, the flow separation near the side outlet of diffuser cover can be well suppressed, which is specifically reflected in the decrease of radial reflux velocity at 97% blade height. Compared with Figure 5 (c), it is found that the initial backflow position is significantly backward. It can be seen from Figure 7 (a) and Figure 5 (d) that the flow separation is significantly reduced. It can be observed from Figure 6 (b) and Figure 7 (b) that the radial velocity is greater than 0 at the height of 97% blades, but there is still a small circumference in the meridional flow channel, and most of the circumference is located in the bend, which has little effect on the diffuser. It can be seen from (c) (d) in Figure 6 and Figure 7 that when the outlet width of the diffuser is reduced to 37mm or less, the flow separation in the diffuser has disappeared, and the radial velocity at the outlet of the diffuser has significantly increased, but the radial velocity is relatively uniform in the latter half of the diffuser. It can be found from Figure 6 that when the outlet width of diffuser is reduced, the flow effect brought by the impeller at the inlet of the diffuser is also reduced, and its speed is also relatively increased.

Figure 6. the radial velocity diagram of 97% blade height at different outlet widths
From the radial velocity diagram and the meridional flow diagram of the outlet widths of different diffusers, it can be found that reducing the outlet width of diffuser can effectively restrain the flow separation, which is helpful to improve the radial velocity in the diffuser, and reduce the flow impact brought by the centrifugal impeller. Therefore, the flow field in the diffuser can be improved and the flow performance can be optimized by appropriately reducing the outlet width of diffuser in the design.

3.2. Aerodynamic Performance
The performance curve before and after the diffuser outlet changes is shown in Figure 8, it can be seen that the total pressure ratio of compressor and the efficiency change-trend are consistent under different diffuser outlet widths. Compared with the prototype, it can be clearly seen that after reducing the outlet width of the diffuser, both total pressure ratio and efficiency are significantly improved in the stable
operating conditions. However, with a further reduction in the diffuser outlet width, the change in total pressure ratio and variable efficiency of centrifugal compressor is not very sharp, which indicates that the flow separation in diffuser is no longer the main factor affecting the performance of centrifugal compressor.

Compared with the curves before and after the diffuser outlet width changes, it can be found that when the mass flow rate is 5.6kg/s, the variable efficiency of centrifugal compressor increased by about 3.5%, and the total pressure ratio increased by nearly 0.03. In addition, the variable efficiency and total pressure ratio of centrifugal compressor at the near-surge point change, which are about 0.3% and 0.005 respectively. However, the variable efficiency and total pressure ratio of the centrifugal compressor at the near-plugging point are greatly improved, the variable efficiency of centrifugal compressor is increased by 9 percentage points, and the total pressure ratio is increased by 0.04, which is greater than the mass flow rate of 5.6kg/s. In the mass flow rate of 5.6kg/s to 8kg/s, the total pressure ratio and the efficiency of centrifugal compressor also have a considerable improvement, but the amplitude is not as large as that of the near-plugging point. Therefore, it can be concluded that the aerodynamic performance of centrifugal compressor can be effectively improved by reducing the outlet width of diffuser, especially near the plugging point, and the efficiency and total pressure ratio can be greatly improved.

It is not difficult to see from Figure 8 (b) that the maximum efficiency of the centrifugal compressor is no longer generated when the mass flow rate is 5.6kg/s, but at a mass flow rate of 6.3kg/s. Therefore, the change in diffuser outlet width has a certain impact on the design point. Compared with the performance curves of different diffuser outlet widths, it is found that when the diffuser outlet width is 41mm, the highest value of variable efficiency appears at a mass flow rate of 6.3kg/s, which is 87.32%. The result is 1.18% higher than that of 5.6kg/s, and the error is less than 1% compared with other outlet widths. However, the maximum total pressure ratio is still at a mass flow rate 5.6kg/s, and the maximum value is 1.45 when the outlet width of diffuser is 35mm. It can be seen that the change in the outlet width of diffuser has little influence on the flow rate at the highest point of total pressure ratio.

In general, it can be concluded that reducing the width of diffuser outlet can make the maximum efficiency point of centrifugal compressor move to the direction of large flow rate. Within the stable working range, the aerodynamic performance can be effectively improved by reducing the width of diffuser outlet, so as to optimize the centrifugal compressor.

![Total pressure ratio diagram](image1)
![Efficiency Diagram](image2)

**Figure 8. Performance Curve Diagram of Different Diffuser Outlet Width**

4. Conclusion
In view of the above-described study of the diffuser outlet width, the following conclusions can be drawn:

(1) The effect of centrifugal impeller on diffuser can be reduced and the inlet flow can be improved by reducing the width of diffuser outlet.

(2) The flow separation inside the diffuser can be effectively restrained by reducing the width of diffuser outlet, so that the radial velocity at the diffuser outlet is more uniform, and the flow field structure inside the diffuser is improved to some extent.
(3) When the outlet width of diffuser is appropriately reduced, the overall performance of compressor is obviously increased, which is mainly reflected by the fact that when the mass flow rate is 5.6kg/s, the total pressure ratio increases by 0.03% and the variable efficiency increases by 3.5%. Simultaneously, the total pressure ratio increased by 0.04% at the near-plugging point, and the variable efficiency increases by 9%.

(4) There is an optimal outlet width of diffuser, which makes the efficiency of centrifugal compressor reach the highest, but not the smaller the better. Combined with the internal flow field of diffuser, it is very important to choose an appropriate outlet width of diffuser. In this paper, the best outlet width of diffuser is 37mm.

Acknowledgements

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: The Public Welfare Technology Foundation of Zhejiang Province in China (Grant No. 2017C31018), the Open Project of State Key Laboratory of Clean Energy Utilization, Zhejiang University (Grant No. ZJUCEU2011002), The Foundation of Key Research and Development Plan in Zhejiang Province in China (Grant No. 2018C04002) and the Public Welfare Technology Foundation of Zhejiang Province in China (Grant No. LY18E090005).

References

[1] ZHONG Hao. Introduction and Proficiency of Centrifugal Compressor [M]. Beijing: Machinery Industry Press, 2015.
[2] DOU Huashu. Design Principles of Radial Vaneless Diffuser with Non-parallel Wall [J]. Fluid Machinery, 1990 (3): 15-19.
[3] MO Zigao, SHI Jianjun, GUAN Xu, et al. Experimental verification of the effect of vaneless diffuser width on compressor performance [J]. Fluid Machinery, 2004, 32 (1): 1-3.
[4] Gao Limin, Liu Bo, Wang Huan. Numerical Study of the Influence of Bladeless Diffuser on Flow Field and Performance of Centrifugal Compressor[J]. China Mechanical Engineering, 2008, 19(17).
[5] Ma Chao, Liu Yulong, Zhu Zhiwu, et al. Numerical study on the influence of the contraction Angle of vaneless diffuser on the performance of automotive centrifugal compressor [J]. Automotive engines, 2010(4):11-15.
[6] ME Lixin, KUANG Zhonghua, LIU Yang, et al. Study on meridian shape of vaneless diffuser import [J]. Fan Technology, 2012 (6): 16-21.
[7] WANG Songling, KONG Yu, ZHANG Lei. Study on the influence of radius ratio of vaneless diffuser on internal flow [J]. Electric Power Science and Engineering, 2017 (10): 45-49.
[8] Dorney D J , Davis R L , Mclaughlin D K . Numerical simulations of flows in centrifugal turbomachinery[J]. Journal of Propulsion & Power, 1971, 11(5):899-907.
[9] Strelets M. Detached eddy simulation of massively separated flows[C]// AIAA Fluid Dynamics Conference and Exhibit. 2001.
[10] KANG Shun, LIU Qiang, QI Mingxu. C FD Analysis and Results Confirmation of a High Pressure Ratio Centrifugal Impeller Part I: Results Confirmation [C]// Academic Conference on Thermo-engine Aerodynamics of the Chinese Society of Engineering Thermophysics. 2004.
[11] LIU Ruitao, Xu Zhong. Research progress of internal flow in centrifugal turbomachinery [J]. Mechanics Exhibition, 2003 (04): 518-532.
[12] ZHAO Huijing, WANG Ziheng, SUN Ye Chen, et al. The numerical study of the effect of turbulence model on the aerodynamic performance and flow field structure of high pressure ratio centrifugal compressor [J]. Fan Technology.