Numerical study of the influence of high-transonic outlet blade angle of the Wedge diffuser with airfoil characteristics on the performance centrifugal compressor

Huan Bai¹, Qingkuo Li²,*, Yi Wang³*

¹School of Aeronautics and Astronautics, Central South University, Changsha, China, 410083
²Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing, China, 100190
³School of Aeronautics and Astronautics, Central South University, Changsha, China, 410083
*1qk0418@163.com
* e-mail: lqk0418@163.com

Abstract—In this paper, a transonic centrifugal compressor is investigated, and a design method of a wedge diffuser with airfoil characteristics is proposed. The effect of different outlet blade angles of the wedge diffuser with airfoil characteristics on the performance of the compressor was studied using a validated numerical calculation method. The results show that the use of a wedge diffuser with airfoil characteristics can more suitable for the transonic flow, which greatly reduces the flow loss of the diffuser pressure surface due to the inconsistent incidence. In addition, without changing the shape of the prototype compressor, the compressor performance is best when the outlet blade angle is 20 ~ 22 °. Compared with the prototype compressor, its peak efficiency increased by 0.93%, the pressure ratio increased by 0.517%, and the stable working margin increased by 4.2%.

1. INTRODUCTION

Due to the advantages of compact structure, high pressure ratio, good reliability and wide operating margin, centrifugal compressors are widely used in turbochargers and gas turbines; In the aviation field, high-pressure-ratio centrifugal compressors have been valued due to the high thrust-to-weight ratio, high-efficiency and compact structure requirements [1]. However, with the increase of the pressure ratio, the stable working margin and efficiency of the centrifugal compressor will greatly decrease [2-4]. Therefore, the research on high-pressure, high-efficiency and marginal centrifugal compressors has always been a hot issue. Researchers have performed aerodynamic analysis and experimental design on centrifugal compressors with pressure ratios of 6: 1 [5] and 8: 1 [6]. Palmer D L[7] proposed a two-stage centrifugal compressor with a pressure ratio of 14.

In recent years, with the improvement of measurement methods and computer performance, researchers have adopted more advanced measurement methods to study the internal flow field of the compressor. Wernet MP[8] uses PIV measurement technology to reveal the time evolution of unsteady flow during surge. With the deepening of the understanding of the internal flow field of the compressor, the blade shape has also developed from two-dimensional design to three-dimensional. At the same time,
the application of tandem blades, splitter blades, and swept-back blades [9-11] enables the impeller efficiency to basically reach more than 90%. However, the stage performance of the centrifugal compressor has not been greatly improved, the main reason is that the airflow velocity at the impeller outlet is high and unsteady. In addition, the inlet of the diffuser and the outlet of the impeller are very close. There is a strong unsteady interaction between them, which deteriorates the matching of the centrifugal impeller and the diffuser. And greatly reduce the efficiency and stable working range of centrifugal compressors, leading to the design of high-performance diffusers is very challenging [12,13].

Currently, many researchers work on diffusers. Chenguang Lai [14] used numerical simulation to study the internal flow of four types of vaned diffusers, and obtained the flow field and pressure distribution of different vanes. The effects of four different diffusers on the performance of the compressor stage are analyzed. The research shows that the performance of the compressor stage of the airfoil blade diffuser is the best. Lei Qiu [15] studied the conformal channel diffuser and found that this diffuser can eliminate the phenomenon of backflow and mixing caused by sharp turns of the airflow, thereby avoiding higher flow losses. Drtina [16] studied the splitter diffuser with different circumferential positions in a low pressure ratio centrifugal compressor through numerical simulation. Research shows that under low flow conditions, when the leading edge of the splitter blade moves toward the suction surface of the main blade, the performance of the diffuser is improved.

Based on previous studies, this paper modified the wedge diffuser blades of a high transonic centrifugal compressor. The tailing edge of the diffuser was optimized, and the influence of different outlet blade angles of the diffuser on the performance of the compressor was explored. The internal flow field and loss mechanism of the diffuser blade are analyzed in detail, and it is expected to provide guidance and reference for the design of the high transonic centrifugal compressor diffuser.

2. CASE DESCRIPTION

The NASA DDA 404-III high-pressure ratio centrifugal compressor [17] was investigated. The compressor design parameters are given in Table 1. The geometric parameters of the original impeller and diffuser are given by McKain and Holbrook [18], as shown in Figure 1. CASE

| TABLE 1. CENTRIFUGAL COMPRESSOR DESIGN PARAMETERS |
|-----------------------------------------------|
| Parameter          | Value              |
| Design Mass Flow, $m_d$       | 4.54 kg/s          |
| Total Pressure ratio, $\pi$     | 4.0                |
| Adiabatic efficiency, $\eta_a$ | 83.3 %             |
| Rotating speed, $N$            | 21789 r/min        |
| Exit diameter, $R_2$           | 215.72 mm          |
| Impeller, $Z_1$                | 15full+15splitter  |
| Diffuser, $Z_2$                | 24                 |
| Exit blade height, $h_2$       | 17mm               |
| Exit tip speed, $U_2$          | 492 m/s            |

Figure 1. Centrifugal compressor meridian channel and 3D pattern
Table 2 gives the different diffuser parameters. Keep the other geometric parameters of the diffuser blade unchanged, and only change the outlet blade angle of the diffuser.

| Case  | R3/mm | R4/mm | α3A/(°) | α4A/(°) | N  |
|-------|-------|-------|---------|---------|----|
| Case1 | 232.54| 363.22| 12      | 16      | 24 |
| Case2 | 232.54| 363.22| 12      | 18      | 24 |
| Case3 | 232.54| 363.22| 12      | 20      | 24 |
| Case4 | 232.54| 363.22| 12      | 22      | 24 |
| Case5 | 232.54| 363.22| 12      | 24      | 24 |

Since the vaned numbers remains the same, the width of the blade outlet will be changed at the same time as the angle of the outlet blade is changed. To control the outlet width of the diffuser to be consistent, the trailing edge must be appropriately modified. The specific parameterization process is shown in Figure 2. The Bessel curve is used to control the suction and pressure surfaces of the control blades. Given $n+1=6$ control points $D_i$, the $n = 5$th order Bezier curve is formed[19]. The curve expression is defined as follows:

$$D(v) = \sum_{i=0}^{k} D_i B_{i,k}(v) \quad (1)$$

Note that $0\leq v\leq 1$, and $B_{i,k}$ is a Berstein basis function of order $k$, which is defined as follows:

$$B_{i,k}(v) = C_{k}^{i}(1-v)^{k-i}v^{i} = \frac{k!}{i!(k-i)!}(1-v)^{k-i}v^{i} \quad (2)$$

3. NUMERICAL VERIFICATION

3.1 Numerical Methods

This paper uses the three-dimensional steady finite volume method to solve the Reynolds average N-S equation. The modified Jameson central difference format and fourth-order Runge-Kutta format are used to discretize space and time. The turbulence model uses the Spalart-Allamaras model. The governing equation can be defined as:

$$\frac{\partial (\rho \zeta)}{\partial t} + \text{div}(\rho V \zeta) = \text{div}(\Psi \text{grad} \zeta) + \phi \quad (3)$$

Among them, $\rho$ is density, $V$ is volume, $t$ is time, $\zeta$ is universal variable, $\Psi$ is generalized diffusion coefficient, and $\phi$ is generalized source term.
The numerical calculation boundary conditions remain the same as in reference [18], total temperature (288.16K), total pressure (101325Pa), and airflow angle (axial inlet). The outlet is given an average static pressure, and an adiabatic non-slip solid wall surface is used. The impeller speed is 21789r/min, and the diffuser speed is defined as 0. Multi-grid technology and enhanced implicit algorithms are used to accelerate convergence, and the regional inlet and outlet mass flow error is less than 0.5%.

The numerical simulation is from the impeller inlet to the diffuser outlet shown in Figure 1. The grid of the impeller and diffuser is generated by AutoGrid. The "C" grid is used near the impeller blades, and the "H" grid topology is used around the blades. The mesh was refined near the blade surface and near the end walls. In order to better improve the resolution of the boundary layer, ensure that $y^+ \approx 2$. The specific mesh is shown in Figure 3.

3.2 Grid Independence Verification and Turbulence Model Selection

Before the numerical study, the grid independence is verified first. In this paper, four different numbers of grids are studied, and the calculation results under design mass flow are shown in Table 3. Figure 3 shows the trend of calculation results for all grids. When the number of grid nodes exceeds 2 million, CFD simulation will produce consistent results. Therefore, the numerical method with 2 million grids has reached the requirements of grid independence.

| Impeller | Diffuser | Min. Skewness | $\pi_{1-4}$ | $\eta_{1-4}$ |
|----------|----------|---------------|-------------|-------------|
| 1,036,548| 377,522  | 16.8          | 4.204       | 0.8283      |
| 1,303,248| 409,431  | 17.2          | 4.213       | 0.8298      |
| 1,543,188| 475,665  | 17.5          | 4.215       | 0.8307      |
| 1,887,424| 590,919  | 17.4          | 4.214       | 0.8307      |
In addition to studying the S-A turbulence model, this paper also considers the SST and K-E models and compares them with the experimental results. The stage performance of the compressor is shown in Figure 5. It can be seen from Figure 5 that the K-E model and the S-A model are more accurate in predicting the compressor surge margin. The SST and S-A models are more accurate in predicting stage performance. Therefore, considering the time cost and accuracy, this paper chooses the S-A model for all simulation calculations.

3.3 Calculation Results

Figure 6 shows the comparison of the total pressure ratio and the adiabatic efficiency of the numerical calculation and experimental results of the high pressure ratio centrifugal compressor at 80%, 90%, and 100% speed. Numerical calculations show that the compression ratio is slightly higher and the peak efficiency is slightly lower. And the numerically
Figure 6. Comparison of calculation results with example results calculated choke mass flow at different speeds is about 1.7%. In general, the total pressure ratio and adiabatic efficiency of the compressor are basically consistent with the experimental results, and the calculation accuracy can satisfy the analysis and prediction of the performance and internal flow of the centrifugal compressor.

4. DISCUSSION AND ANALYSIS

4.1 Effect of Different Outlet Blade Angles on Compressor Performance

In this paper, five kinds of wedge-shaped diffusers with airfoil characteristics are studied, and the stage performance of the compressor are obtained by changing the outlet blade angle. Figures 7 and 8 compare the effects of different blade angles on compressor performance.
As shown in Figs. 7 and 8, when the outlet blade angle is modified, as the diffuser outlet blade angle increases, its stable working margin increases first and then decreases. And the efficiency and pressure ratio are all higher than the Baseline wedge diffuser. In order to quantitatively compare the trend of performance changes, the maximum efficiency points and maximum pressure ratios of different blade angle diffusers were taken out, and the comparison graph of the peak efficiency and maximum pressure ratio of different diffusers is shown in Figure 9.
As shown in Figure 9, with the increase of the outlet blade angle, the compressor stage efficiency gradually increases, and the maximum value is 83.88% when the blade angle is 18 °, and then the peak efficiency gradually decreases as the outlet blade angle increases. Similarly, the pressure ratio characteristic also increases first and then decreases as the outlet blade angle increases. When the outlet blade angle is 22 °, the maximum value of the pressure ratio is 4.303. But when the outlet blade angle exceeds 24 °, the pressure ratio will decrease rapidly. Taken together, when the outlet blade angle is 20 °, the compressor stage performance is optimal. Compared with Baseline, it can be seen that the peak efficiency of the diffuser with airfoil characteristics is increased by 0.93%, the pressure ratio is increased by 0.517%, and the stable working margin is increased by 4.2%.

In centrifugal compressors, the function of a diffuser is mainly to convert kinetic energy into static pressure energy. For the determination of the performance of a diffuser, the static pressure recovery coefficient $C_p$ [20] is generally used, which is defined as:

$$C_p = \frac{P_{Rs} - P_{3s}}{P_{3t} - P_{3s}} \quad (4)$$

Among them, $P_{Rs}$ is the average static pressure value of the diffuser along the R section, $P_{3s}$ is the average static pressure value of the diffuser inlet section, and $P_{3t}$ is the average total pressure value of the diffuser inlet section.

![Figure 10. Static pressure recovery coefficient of diffuser](image)

Figure 10 shows the static pressure recovery coefficients of the diffuser with different outlet blade angles near the design point. When other geometric conditions of the diffuser are unchanged, different outlet pressures of the diffuser will cause the static pressure recovery coefficient to be slightly different. When the outlet blade angle is 20-22 degrees, the static pressure recovery coefficient is the highest. When the outlet blade angle is 16 degrees, the static pressure recovery coefficient is the smallest. In general, as the outlet blade angle increases, the static pressure recovery coefficient also increases first and then decreases, which is mutually verified with the results obtained in figure 9.

4.2 Flow Field Analysis of Diffuser with Different Outlet Blade Angles

Because the streamline can intuitively reflect the flow of steady numerical simulation, the Mach number contour can reflect the Mach number distribution. Therefore, the flow field distribution of the outlet blade angles of 16 °, 20 °, 24 ° and the Baseline wedge diffuser was investigated. Then analyze the flow field distribution at 50% span at the choke mass flow rate of 4.75 kg/s and near design mass flow rate of 4.5 kg/s.
When the mass flow rate is 4.75 kg/s, the diffuser is in a choke state at this time, which is specifically manifested as a supersonic at the throat, as shown in figure 11. And there are trailing edge vortices of different sizes at the diffuser outlet. As the outlet blade angle increases, the trailing edge vortex increases. This leads to increased losses, which is why the intermediate efficiency in Figure 9 continues to decline. And it can be found that when the diffuser incidence is negative at the choke mass flow, the pressure surface flow separation is easy to occur at this time. Therefore, when the diffuser outlet blade angle is small, the curvature of the suction surface will increase. The flow separation of the pressure surface of the diffuser blade is suppressed.

As shown in figure 11, when $\alpha_{4A} = 16^\circ$, there is less flow separation on the pressure surface, but it is quickly suppressed. However, with the increase of the outlet blade angle, the fluid is restrained by the restraining force, and the flow separation of the pressure surface will gradually increase. Finally mixed with the diffuser trailing edge vortex. For the Baseline diffuser, the flow separation on the pressure surface is larger, and the flow separation occurs on the leading edge of the blade, so the loss is greatest here, resulting in a decrease in efficiency.
Figure 12. Mach number and streamline at 50% span when $m=4.5\,\text{kg/s}$

When the mass flow rate is $4.5\,\text{kg/s}$, the incidence $i=0\,\text{°}$ of the leading edge of the diffuser causes a transonic flow at the inlet of the diffuser. As shown in figure 12, the internal flow of the diffuser is better at this time, and there is basically no obvious flow separation. However, similarly, as the outlet blade angle increases, the degree of curvature of the suction surface of the blade decreases. With the increase of the passage area, the tailing edge of the diffuser will have backflow. The wedge diffuser has the most serious backflow area of the trailing edge, while the diffuser with the airfoil trailing edge has a smaller backflow area. This is the advantage of having an airfoil trailing edge diffuser.

In order to further represent the flow loss inside the diffuser, the entropy distribution near the design point is given below.
As can be seen from Figure 13, as the diffuser outlet blade angle increases, the diffuser entropy increases along the pressure surface. Because when the outlet blade angle is small, its diffuser blade profile has a curvature from the inlet to the outlet, which is more suitable for transonic flow. As the blade angle of the outlet continuously increases, its blade profile gradually becomes a wedge diffuser. As a result, the loss of the leading edge of the diffuser due to the inconsistent incidence increases, so the performance is poor. In general, diffusers with airfoil characteristics are more suitable for transonic flow and improve the static pressure recovery coefficient of the diffuser. In addition, the compressor stage performance and stable working range have been improved.

5. CONCLUSION
In this paper, a numerical method is used to simulate a centrifugal compressor, and a modeling method of a diffuser with airfoil characteristics is proposed. Then the influence of the diffuser with airfoil characteristics on the performance of transonic compressors is analyzed. The main conclusions are as follows:

1. A modeling method of diffuser with airfoil characteristics is proposed. First, the Bezier curve control point is found by parameterizing the wedge diffuser blade. Then by adjusting the control points to adjust the blade thickness and blade angle, a diffuser trailing edge with airfoil characteristics is obtained.

2. Compared with the wedged diffuser, the diffuser with airfoil characteristics is more suitable for transonic flow, which greatly reduces the flow loss of the diffuser due to the inconsistent incidence. In addition, the characteristics of the airfoil trailing edge can effectively reduce the flow loss caused by trailing edge vortex mixing.

3. For transonic centrifugal compressors, when using diffuser blades with airfoil characteristics, it is recommended to set the outlet blade angle to 20 ~ 22 °, and the performance of the diffuser is better. Compared with the prototype compressor, its peak efficiency increased by 0.93%, the pressure ratio increased by 0.517%, and the stable working margin increased by 4.2%.

REFERENCES
[1] Krain H. Review of centrifugal compressor’s application and development[J]. Journal of turbomachinery, 2005, 127(1): 25-34.
[2] Gravdahl J T, Egeland O. Centrifugal compressor surge and speed control[J]. IEEE Transactions on control systems technology, 1999, 7(5): 567-579.
[3] Boyce M P. Principles Of Operation And Performance Estimation Of Centrifugal Compressors[C]//Proceedings of the 22nd Turbomachinery Symposium. Texas A&M University. Turbomachinery Laboratories, 1993.
[4] Gui F, Reinarts T R, Scaringe R P, et al. Design and Experimental Study of High-Speed Low-Flow-Rate Centrifugal Compressors[R]. MAINSTREAM ENGINEERING CORP ROCKLEDGE FL, 1995.

[5] Klassen H A, Wood J R, Schumann L F. Experimental performance of a 16.10-centimeter-tip-diameter sweptback centrifugal compressor designed for a 6:1 pressure ratio[J]. 1977.

[6] Osborne C, Runstadler PW, Stacy WD. Aerodynamic and mechanical design of an 8:1 pressure ratio centrifugal compressor[R]. Washington, D. C : NASA, 1974.

[7] Palmer M P, Bright M M, Skoch G J. An investigation of surge in a high-speed centrifugal compressor using digital PIV[J]. Journal of turbomachinery, 2001, 123(2): 418-428.

[8] Pedersen N, Larsen P S, Jacobsen C B. Flow in a centrifugal pump impeller at design and off-design conditions—part I: particle image velocimetry (PIV) and laser Doppler velocimetry (LDV) measurements[J]. Journal of fluids engineering, 2003, 125(1): 61-72.

[10] Josuhn-Kadner B. Flow Field and Performance of a Centrifugal Compressor Rotor With Tandem Blades of Adjustable Geometry[C]ASME 1994 International Gas Turbine and Aeroengine Congress and Exposition. 1994:V001T01A004.

[11] Zhu S. Computation and Analysis of Three Dimensional Flow Field for Centrifugal Compressor Impeller with Backward Swept Blades[J]. Transactions of Csice, 1983.

[12] Baldssarre L, Cellai A, Ferrara G, et al. Experimental investigation and characterization of the rotating stall in a high pressure centrifugal compressor Part III: influence of diffuser geometry on stall inception and performance[R].ASME Paper 2003-GT-3890,2003.

[13] H Krain, B Hoffmann. Flow Study of a Redesigned High-Pressure-Ratio Centrifugal Compressor[J].Journal of Propulsion & Power, 2007 (2008-10 ) :7.

[14] Chenguang Lai, Hongqiang Zeng, Yan Zhuang, et al. Numerical simulation of the effect of different blade type diffuser on the performance of centrifugal compressor [J]. Journal of Chongqing University of Technology: Natural Science Edition, 2016, 30 (12): 42-47.

[15] Lei Qiu, Yan Yin, Liu Jiao, et al. Development of Conformal Channel Diffuser for Centrifugal Compressor of Missile Turbojet Engine [J]. Journal of Ningbo University of Technology, 2016, 28 (1): 1-5.

[16] Drtina P, Dalbert P, Rütti K, et al. Optimization of a Diffuser With Splitter by Numerical Simulation[C]//ASME 1993 International Gas Turbine and Aeroengine Congress and Exposition. American Society of Mechanical Engineers, 1993: V001T03A051-V001T03A051.

[17] Skoch G J, Prahst P S, Wernet M W, et al. Laser anemometer measurements of the flow field in a 4:1 pressure ratio centrifugal impeller[R].Technical Report ARL-TR-1448,1997.

[18] McKain T F, Holbrook G J. Coordinates for a high performance 4:1 pressure ratio centrifugal compressor[R].NASA contractor report 204134,1997.

[19] Demeulenaere, Alain, Alban Ligout, and Charles Hirsch. "Application of multipoint optimization to the design of turbomachinery blades." ASME turbo expo 2004: power for land, sea, and air. American Society of Mechanical Engineers Digital Collection, 2004.

[20] Filipenca V G, Deniz S, Johnston J M, et al. Effects of inlet flow field conditions on the performance of centrifugal compressor diffusers: Part 1—discrete-passage diffuser[J]. J. Turbomach., 1998, 122(1): 1-10.