Experimental and numerical investigation on the influence of the diffuser blade setting angle on the performance of centrifugal compressor

Junyi Fang¹, Chunjun Ji¹, Chunyang Li¹, Ruiwu Zhang² and Shalyaev Denis²

¹ School of Power Engineering. Dalian University of Technology, Dalian, China,
² Liaoning Fu An gas turbine Co. Anshan, PR China

E-mail: 719783952@qq.com;

Abstract: Setting angle of the front blade row of tandem cascade diffuser was investigated experimentally to understand its influence on the performance of centrifugal compressor. Three diffusers with different setting angles were tested. The change of front blade setting angle mainly alters. The angle of attack (AOA) on the leading edge of front blades, which affects the surge point of the compressor. The throat area of the blade row is also modified, which affect the chock point of the compressor. With the increase of setting angle, performance curves of centrifugal compressor move to large flow rate side, the operation range is increased at the same time. In addition, numerical simulation was used to investigate the flow details on the passage of diffuser blades. Overall, the setting angle of front blade row has a significant influence of the performance of centrifugal compressor.

Nomenclature

| Symbol | Description |
|--------|-------------|
| AOA    | Angle of attack, ° |
| b      | Blade height, mm |
| Cₚ     | static pressure recovery coefficient |
| D      | Setting diameter, mm |
| Gₘ     | Mass flow rate, kg/s |
| Ma     | Relative Mach number |
| k      | Adiabatic exponent |
| P      | Static pressure |
The design of diffuser is an important part of a compressor design, especially for the design of transonic compressors. Transonic compressor has a high rotating speed and large backward curved angle, it means the flow angle at the impeller outlet will be large and the flow speed will be fast. In this condition, the diffuser will bear a large load. A large flow angle will cause more friction loss[1]. Vane-diffuser is widely used in transonic compressor, many researches[2-4] showed that vane-diffuser could generally attain a higher efficiency and pressure ratio than vane less diffuser. The single row vane-diffuser has a good performance but its operation range is narrow. To compromise, low solidity vane-diffuser was put forward[5]. Another choice is the tandem cascade diffuser[6]. Tandem cascade diffuser can get a better performance than single row vane-diffuser, its flow field is more uniform[7].

The current research on cascade diffusers mainly focuses on the aspect ratio[8], blade bending angle[9], number of blades, and on the relative positions of the tandem cascades[10-12]. These studies have accumulated an extensive number of calculations and experimental data. Liu Lei used numerical simulations to reconstruct the diffuser of a centrifugal compressor, and quantified the different performances with the three chord length ratios of 0.7, 1.0, and 2.0[13]. The results indicate that the efficiency of the cascade diffuser is better than that of the wedge diffuser, and the compressor stability margin can be significantly widened by the cascade diffuser. The circumferential offset has considerable influences on the load distribution of the front and rear row blades. When the circumferential offset increases, the load of the front row will also increase. Minor changes of the inlet flow angle of the rear row are documented as a function of the inlet attack angle of the tandem cascade. However, when the front row is associated with a large attack angle, the rear row will have a small attack angle so that the tandem cascade can maintain a small loss within a large range of attack angles[14]. Zhou[15] used an experimental design method to improve the load of tandem cascades.

1. Introduction

- **P**\*: Total pressure
- **R**: Ideal gas constant
- **β**: Setting angle
- **ΔK_y**: Surge margin
- **π**: Total pressure ratio
- **η**: Isentropic efficiency
- **ζ**: Total pressure loss coefficient

**Subscript and superscript**
- **3**: Leading edge of front blade row
- **4**: Trailing edge of front blade row
- **5**: Leading edge of rear blade row
- **6**: Trailing edge of front blade row
- ***: “Total” state
- **design**: Design point
- **surge**: Surge point
and noted that the inlet stagger angle of the rear row has a significant influence on the response value of the total pressure ratio and isentropic efficiency. Studies on the flow angle of diffuser blades are mostly aimed at the single blade diffuser. Tamaki[16] showed the effect of the diffuser throat area on the stable range of a centrifugal compressor and categorized the matching between an impeller and vaned diffuser into four types. The other experiment of Tamaki presented the effect of diffuser vane setting angle on the performance of a centrifugal compressor with volute. The experiment provided the evidence that the choke flow was independent of the van setting angle but determined by the throat area. The surge flow rate tended to decrease as the vane setting angle increase. Being closed to the impeller, vaned diffuser will be greatly affected by the unsteady flow form the impeller outlet, it is called impeller-diffuser interactions. Zhao[17] indicated that the generation and shedding of leading-edge vortex is closely related to the altering inlet flow angle in the circumferential direction. Anish[18] noted that the main factor influencing the diffuser instability is the circumferential variation of the flow angle at the leading edge of the diffuser vane. The aerodynamic loss in the diffuser is induced mainly by the generation and shedding of the vortexes which exist on the diffuser vane leading edge, on the suction side of the diffuser vane near the hub and on the diffuser vane trailing edge.

However, there are few researches on the setting angle of the front vanes of the tandem cascade diffuser. In this study, three different setting angles of the front blades of a tandem cascade diffuser are simulated and tested on a centrifugal compressor. This study proves that the setting angle plays an important role in the tandem cascade, it greatly affect the performance of the centrifugal compressor. What is more, the setting angle will play a same role as the relative positions with the tandem cascades. The results of this study have high reference value for the design of tandem cascade diffuser.

2. Geometric model and test procedures

The diffuser is from a centrifugal compressor of a 4 MW gas turbine. It is the second stage of the gas turbine compressor. At design point, its mass flow rate is 5.2 kg/s, pressure ratio is 2.56 and efficiency is 0.85. There are 13 main blades and 13 split blades on the impeller, and 23 blades in each row of the diffuser. Figure 1 presents the flow path and the geometric model of the diffuser.

![Fig 1: Meridional flow path and geometric model of diffuser.](image_url)
Table 1 lists the geometric parameters of the diffuser. This study provides 3 different setting angles, named as case 1, 2 and 3 respectively. In the three cases only the setting angle of the front blade row is changed, the vane profile remains the same. Figure 2 shows the relative position of the front and rear rows and the test model. It is showed that the front row of blade is detachable, so that all the three cases could be tested on the compressor. When the setting angle changes, the chink between the front and rear blades change greatly the throat area of front blade is also changed.

Table 1. Geometric parameters of the diffuser

|                  | Case 1 | Case 2 | Case 3 |
|------------------|--------|--------|--------|
| Inlet setting angle of first row, $\beta_3 (^\circ)$ | 67.3   | 69.3   | 71.3   |
| Outlet setting angle of first row, $\beta_4 (^\circ)$ | 64.0   | 66.0   | 68.0   |
| Inlet setting angle of second row, $\beta_5 (^\circ)$ | 66.0   | 66.0   | 66.0   |
| Outlet setting angle of second row, $\beta_6 (^\circ)$ | 60.0   | 60.0   | 60.0   |
| Blade Number of first row, $Z_3$                  | 23     | 23     | 23     |
| Blade Number of second row, $Z_4$                  | 23     | 23     | 23     |
| Throat area, mm$^2$                                | 11232  | 10200  | 9167   |

Fig 2. Relative position of the blades and the test model

Figures 3 and 4 show the compressor instrumentation and the diagram of test facility. There are six fixed measuring positions showed in Figure 4, which are used to measure the environmental data and the stage performance parameters of the compressor during the experiment. Figure 3 presents the additional measuring positions at the inlet of impeller, inlet of the diffuser and outlet of the diffuser, each position has probes for $P^*$, $P$ and $T^*$. The systematic error of the probe is 0.025%. During the experiment, the data were collected after 30 minutes of stable operation. All the total pressure and total temperature were obtained as time and area averaged.
3. Experimental result

The experiment for this study is a performance test, and the raw data of the experiment is $P^*$, $P$, and $T^*$ at each measuring position. The total pressure ratio ($\pi^*$) is defined in formula 1, and isentropic efficiency ($\eta$) formula 2. The result is showed on Figure 5. It is obviously that the three cases make a great difference on the performance characteristics. With the increase of the setting angle, the total pressure ratio is slightly increased, and the efficiency is slightly reduced. The operation range is also different, case 3 has the minimum surge margin ($\Delta K_y$) and case 1 has a maximum surge margin. $\Delta K_y$ is defined in formula 3. There is an abnormal point in case 1, on which point the compressor is already surged. Table 2 presents the surge point and $\Delta K_y$ of the experiment.
\[
\pi^* = \frac{P_b^*}{P_1^*} \quad (1)
\]

\[
\eta = \frac{\pi^{(k-1)\times k^{-1}}}{P_b^*} \quad (2)
\]

\[
\Delta K_y = \left(\frac{\pi_{\text{surge}}\times G_{\text{m,design}}}{G_{\text{m,design}}\times \pi_{\text{design},\text{surge}}} - 1\right) \times 100\% \quad (3)
\]

![Graphs showing pressure and efficiency curves](image)

(a) total pressure (b) efficiency

**Fig 5. Performance curves of the compressor**

| Case | \(G_{\text{m, surge}}\) kg/s | \(\pi^*_{\text{surge}}\) | \(G_{\text{m, design}}\) Kg/s | \(\pi^*_{\text{design}}\) | \(\Delta K_y\) % |
|------|------------------------------|----------------|-----------------|----------------|---------|
| Case 1 | 4.011                       | 2.65           | 5.30            | 2.59           | 35.53   |
| Case 2 | 3.84                        | 2.64           | 4.94            | 2.58           | 31.48   |
| Case 3 | 3.46                        | 2.68           | 4.39            | 2.61           | 30.14   |

On the flow rate of 5.2 kg/s, which is the target design point, case 1 has the best performance. While under the condition of small flow rates, case 2 is better than case 1. Case 3 fails to provide sufficient mass flow to meeting the requirement of design.

The total pressure loss coefficient (\(\zeta\)) is widely used to evaluate the flow loss of diffuser. Figure 6 present the change of \(\zeta\) with the flow rate of the three cases. Each case has a minimum \(\zeta\) point, on the right side of this point, the loss increases quickly with the flow rate. Overall, \(\zeta\) of case 1 is lower than the other cases. It is consistent with figure 5, on the condition of large flow rates, case 1 shows higher pressure ratios.

\[
\zeta = \frac{P_1^* - P_b^*}{P_b^* - P_3} \quad (4)
\]
The Fourth Chinese International Turbomachinery Conference (CITC 2020)  
IOP Publishing  
IOP Conf. Series: Materials Science and Engineering  1081  (2021) 012031  
doi:10.1088/1757-899X/1081/1/012031

Fig 6. Total pressure loss coefficient (ζ)

The experiment could only provide an overall performance. To investigate the mechanism of the experimental result, a numerical simulation was made for this study.

4. Numerical method and analyses

In this study the software NUMECA/Fine turbo was use to perform the numerical computations. It contains a solver to 3-D compressible Reynolds-averaged Navier-Stokes equations, which employs an explicit fourth-order Runge–Kutta scheme for the temporal discretization and a second-order central-differenced scheme for the spatial discretization. The shear-stress transport (SST) model is chosen as the turbulence model, because it is more accurate for gap flows\(^{19}\). Another advantage for SST-extend wall function is that the \(Y^+\) allowed is 20–50, it means that under the limit of grid aspect ratio, mesh for this turbulence model will allow a bigger wall grid size. In this way, the mesh number can be less than other models that without wall extend function.

The mesh is created using “Auto Grid 5”, the mesh number for the compressor is 2.2 million, mesh quality totally meets the requirement of the software. Grid independence verification was carried out to ensure the mesh number is enough for the accuracy in computation. Figure 7 shows the mesh of the compressor, there is a fillet at the root of the diffuser blade, so that the model is more consistent with the real diffuser. Periodic conditions are adopted in the grid. Periodicity of impeller is 13, the diffuser is 23. There are three rotor-stator (R/S) surface in the mesh. R/S surface between the impeller and diffuser is set as conservative coupling by pitch wise row, which means a strict conservation of mass, momentum and energy. R/S surface between the two blade rows of diffuser is set as full non matching frozen rotor, which acquire the periodicity of upstream and downstream must be equal and ignore the rotating speed in calculation.
Figure 7. Mesh of the compressor and leading edge grid of the diffuser blade.

Figure 8 presents the comparison between the calculation and the experiment. With the speed of 87% and 100% to the design speed, the comparison shows little differences. The 105% speed curves show a slight larger error, that is because under the speed of 105%, the power of electric motor cannot provide enough power, so an inlet throttling have to be used to complete the performance curve near surge (the brown curve labelled 105% TEST RUN). In this study only the performance of 100% speed is analyzed, but it must be noted that the conclusion of this study is applicable to other speeds.

The static pressure coefficient \( C_p \) of the diffusers, which is defined in expression (5):

\[
C_p = \frac{P_s - P_3}{P_s' - P_3} \tag{5}
\]

Figure 9 presents the \( C_p \) at design point of the three cases. In case 1, there is a stagger between the front and read blade rows, meaning that the outlet diameter of the front blade row is larger than the inlet diameter of the rear blade row. In case 2, the two diameters are equal, and in case 3 the outlet diameter of the front blades is smaller than the inlet diameter of the rear blades. As can be seen in the figure, in case 1, the front blades carry more load than the other cases, while in case 3 the opposite is true. Both case 1 and case 2 show good static pressure distribution at the front row, but case 3 shows a large negative loading near the leading edge of the front blades. It means that the leading edge of the
diffuser has a positive attack angle at design point of case 3. In this condition, case 3 will get a smaller surge margin than others. Case 1 still keep a negative incidence angle at the leading edge of the front blades at the design mass flow. The load distribution of the rear blade is also affected, in case 3 there is a big pressure difference between the pressure surface and suction surface of the rear blade. Too much load on the rear blades will have a negative effect on compressor efficiency.

![Fig 9. Cp distribution on 50% span of the diffuser blades at design point.](image)

Figure 10 and 11 present the velocity vector and flow angle at design point. It is obviously that at the leading edge of front blade row, case 1 has a flow angle that about 2~3 degree smaller than case 3, while the setting angle of case 1 is 4 degree smaller than case 3. It can be seen from the streamline that case 1 has a very small negative angle of attack (AOA) on the leading edge of front blade. Flow angle of case 2 is slightly higher than case 1 on the leading edge of front blade, while it is obviously higher than case 1 on the leading edge of the rear blades. In all sections the case 3 has a highest flow angle. Compared with the setting angle, case 3 has a small positive AOA at design point. This result is consistent with that in figure 9. Tandem cascades are effective in suppressing the separate flow on the diffuser blades. Figure 10 shows the start point of separation is near the trailing edge of SS. In Case 1, affected by the narrow gap between the front and rear blade rows, the separation on front blades is weaker than other cases. Another obvious difference is that in case 1 the wake of front blades is very weak in the passage of the rear blades. But in case 3, the influence area of wake occupies almost two-thirds of the passage. The influence of wake results in poor uniformity in the passage, and then the pressure difference between PS and SS of the rear blades is rising, the load is rising at the same time. It should be noted that in case 1, the rear blades are set as splitter blade of the front blade row in the mesh, because of the stagger. In this condition, there is no R/S surface between the two blade rows, and no break in the streamline.
Fig 10. Velocity vector, (a) leading edge of front blade, (b) rear blade.

Fig 11. Flow angle at the leading edge of blade, (a) front blade; (b) rear blade.

The relative Mach number (Ma) distribution on the 5%, 50% and 95% spans are showed in Figure 12. As a supplement to figure 10, Ma can reflect the velocity variation in the diffuser passage more directly. Relatively, Case 1 and case 2 have a higher Ma on the outlet of diffuser,
that is because they have a higher mass flow rate at design point. Case 3 has the most change of Ma form inlet to outlet, which means a larger static pressure rise and a higher $\zeta$. Ma distribution of case 1 is more uniform, so the load changes more evenly. The narrow gap between the front and rear blades of case 1 results in a jet flow, which effectively depress the wake of the front blade. The boundary later separation is a main reason for the unstable operation. A jet flow will also blew away the boundary layer on the pressure surface of the rear blade. In this way, case 1 presents a lower $\zeta$.

![Ma distribution at 5%, 50% and 95% spans at design point.](image)

**Fig 12. Ma distribution at 5%, 50% and 95% spans at design point.**

Figure 13 is the entropy distribution. The entropy distribution has little difference in three cases. In case 1 and case 2, the low entropy flow avoid the rear blades. The fluid with relatively high entropy passes through the rear blade, in this condition, the entropy will not increased too much by the rear blades. In case 3 the fluid with lowest entropy just passes the rear blades, that means the entropy increase is larger. That is , case 3 has more energy loss, and lower efficiency at design point. The highest entropy fluid is mainly on the wake of blades, while in case 1, the wake of front blade is obviously depressed by the gap between the front and rear blades. With the increase of setting angle, the uniformity of entropy distribution is getting worse, it provide that the loss is increased. The entropy distribution display a consistent conclusion with figure 5 and 6, case 1 has the lowest energy loss and highest efficiency.
5. Conclusions
In this study, the setting angle of front blade of tandem cascade diffuser influence on the performance of a centrifugal compressor is investigated experimentally. In addition, the numerical simulation is used to analysis the flow details. The main conclusions are obtained as following:

1) The setting angle of front blade of tandem cascade diffuser will greatly influence the performance of centrifugal compressor. A 2° change on the setting angle will change the surge margin by 1.5%~4.2%, the design point will also move to small flow rate side with the increase of setting angle. Moreover, lower setting angle will perform a higher efficiency.

2) Under the condition of constant vane profile, the change of setting angle will change the relative position of the front blades and rear blades. The narrow gap between the front and rear blade will inhibit the wake of the front blade, which is beneficial to reduce the load of the rear blades and energy loss.

3) Surge and chock all occur in the passage of front blade. Change of setting angle changes the AOA, which affect the surge point. The change of throat mainly affects the chock point.

Reference

[1] Zangeneh M, Schleer M and Ploger F, et al (2004). Investigation of an inversely designed centrifugal compressor stage part 1 : Design and numerical verification. Journal of Turbomachinery, 126: 73-81.

[2] Moustapha H (2003). Small Gas Turbine Technology: Evolution and Challenge. AIAA :2559

[3] Dehkharqani A, Boroomand M and Eshraghi, H (2014). A numerical investigation of loss coefficient variation in various incidence angles in tandem blades cascade. Volume I: Advances in aerospace technology, doi:10.111539881

[4] Tan Gaoming, He xianghui and Liu Houlin, et al (2016), Design and analysis of a radial diffuser in a single-stage centrifugal pump. Engineering Applications of Computational Fluid Mechanics, 10:1,500-511

[5] Senoo Y (1981). Low solidity circular cascade for flow range blower. Proceedings of advanced concepts in Turbomachinery. Hanover, New Hampshire.

[6] Lars M, Dragan K and Detlev W, et al (2011). High Turning Compressor Tandem Cascade for High Subsonic Flows Part 2: Numerical and Experimental Investigations, AIAA, 5602

[7] Obukhov O, Smirnov and Gysak O (2015). Numerical and experimental investigation of efficiency of waned diffuser of centrifugal compressor. International communication in heat and mass transfer available online,649-658

[8] Sanger L (1971). Analytical study of the effects of geometric changes on the flow characters of tandem blade compressor states. NASA TN, d-6264.

[9] Hautr C (1975). Experimental study of tandem blades for rotor blade usage in a single stage axial flow compressor. Knoxville, US: University of Tennessee.

[10] Tan Gaoming, He xianghui and Liu Houlin, et al, Design and analysis of a radial diffuser in a single-stage centrifugal pump. Engineering Applications of Computational Fluid Mechanics, 2016,10:1,500-511
[11] Oh J, Buckley C and Agraw L, et al (2012). Numerical investigation of low solidity vaned diffuser performance in a high-pressure centrifugal compressor. Part III: Tandem vanes. Journal of Turbomachinery. 134 : 061025-1.

[12] Xie R, Yang Y and Hai Y, et al (2010). Research on centrifugal compressor staggered arrangement cascade diffusers. Gas Turbine Technology 2: 48-52.

[13] Liu lei, Wu Shixun and Zhao Honglei (2019). Numerical Study of the Tandem Cascade Chord Length Ratio on the Performance of Centrifugal compressor. Gas Turbine Technology. 3-32

[14] Shan Shujun, Hou Anping and Li Jianxiong (2018). Effects of Percent Pitch on Performance of High Load Tandem Cascade. Journal of Aerospace Power. 33(5).

[15] Zhou Li, Wang Zhanxue and Liu Zengwen (2014). Investigation on influence of design parameters for tandem cascades diffuser using DOE method. Engineering Applications of Computational Fluid Mechanics, 8(2), 240-251

[16] Tamaki H, Nakao H and Saito M (1999). The Experimental Study of Matching Between Centrifugal Compressor Impeller and Diffuser, ASME J. Turbomachinery. 121(1), pp. 113-118.

[17] Zhao Pengfei, Liu Yan and Li Hongkun, ed al (2016). The effect of impeller–diffuser interactions on diffuser performance in a centrifugal compressor. Engineering Applications of Computational Fluid Mechanics. 10:1, 565-577

[18] Anish S, Sitaram N and Kim H D. (2014). A numerical study of the unsteady interaction effects on diffuser performance in a centrifugal compressor. Journal of Turbomachinery, 136, 011012.

[19] Guo Ran, Jia Liping and Fan Xiaoli, et al. NUMECA tutorials. China Machine press. 2013

Acknowledgement

Firstly, I would like to appreciate the careful guidance form my tutor, professor Chunjun Ji. Then, I pleased to acknowledge Liaoning Fu-An gas turbine Co, who funded this study. At last many thanks for all the staff of the laboratory.