Numerical Investigation of the Flow Field and Aerodynamic Load on Impellers Under Radial Inlet Guide Vanes With Trailing Edge Blowing in Centrifugal Compressor

To cite this article: Jianchi Xin et al 2021 IOP Conf. Ser.: Mater. Sci. Eng. 1081 012040

View the article online for updates and enhancements.
NUMERICAL INVESTIGATION OF THE FLOW FIELD AND AERODYNAMIC LOAD ON IMPELLERS UNDER RADIAL INLET GUIDE VANES WITH TRAILING EDGE BLOWING IN CENTRIFUGAL COMPRESSOR

Jianchi Xin1, Haitao Liu2, Zhitao Tian1, Xiayang Liu1, Xiaozhi Kong1, Xiaofang Wang2
(1. Naval Architecture and Ocean Engineering College, Dalian Maritime University;
2. School of energy and power engineering, Dalian University of technology)
E-mail: xinjc@dlmu.edu.cn

Abstract: The radial inlet is a typical upstream component which is widely used in multiple-stage centrifugal compressors or blowers. Adding the guide vane in the radial inlet could improve the flow field along circumference distribution and increase the efficiency of the compressor as well. However, the guide vane in the radial inlet generates the wake which causes the Rotor-Stator-Interaction (RSI) and further raises the issue of impeller fatigue failure. In order to compensate the wake effect caused by the guide vane, the active control of trailing edge blowing (TEB) is adopted. Based on Computational Fluid Dynamics (CFD) simulation, the radial inlets with unevenly distributed guide vanes (UGV) are considered. This study investigates the guide vane wake in a radial inlet with or without TEB, and aims to figure out the flow phenomenon in radial inlet and the aerodynamic load on the downstream impellers. The simulation results show that the trailing edge blowing has effect on the compressor performance and reduces the wake intensity. By conducting unsteady simulations, the aerodynamic load on the impeller’s leading edge is obtained for the radial inlets with or without TEB. The dominant frequency and the pulse amplitude are obtained by Fast Fourier Transforms (FFT). Compared with the UGV model, the TEB shows lower excitation at the harmonics of the impeller passing frequency, and has little influence on the amplitude at the impeller rotation frequency. It means the TEB could improve the safety of the impeller under radial inlet guide vane condition.

Key words: Radial inlet; Trailing edge blowing; aerodynamic load; compressor

1. INTRODUCTION

The radial inlet is a typical upstream component which is widely used in most multiple-stage centrifugal compressors or blowers. It is required to allow for a uniform flow field upstream of the impeller and therefore ensuring an acceptable performance. As the structure of the radial inlet is complex, the flow field in the radial inlet is a complex three-dimensional flow field, the flow can hardly reach a uniform distribution. The distortion flow field in radial inlet has badly influence on performance of impeller [1]. Compared to an axial flow inlet, the radial

1 Please address all correspondences to dlwxf@dlut.edu.cn.
Inlet results in 1 to 4 percent decrease of the efficiency on the performance of the whole centrifugal compressor [2]. Therefore, the study of the radial inlet is meaningful for the compressor.

In order to understand the flow field in a radial inlet, Tan [3] reviewed the radial inlet of the centrifugal compressor, which included the internal flow in radial inlet, the effects of radial inlet on performance, structure parameter design and optimum design method. At presents, most researchers adopted experiments and computation to investigate the flow characteristics. Compared with the radial inlet and axial inlet, Long [4] consider that the uniform at impeller inlet by radial inlet result in 3 percent decrease of the efficiency and 2.4 percent decrease of the total pressure, and impeller rotating has little influence on flow field in radial inlet. Running variable conditions, Dong [5] found that the radial inlet results in the attack angle decreasing close to 1.4°, and the flow field showed stronger uniform distribution along the circumference and radial directions, which is more obviously at off-design conditions. Based on the understanding of the characteristics of inlet volute's structure and interior flow, Wu [6] through analyzed the property of every channel cross-section, provided a parameterized design model, which can simplify radial inlet design process entity's structure and improve design precision and working efficiency. Han [7] measured the detailed flow in radial inlet with five-hole probes. The investigation indicated that there were five main flow separations and vortices, which is the primary cause of the flow loss and the flow distortions. Feng [8] analyzed the flow field of the radial inlet and first stage with CFD, the results indicated that the performance of radial inlet chambers had close relations to geometry shape. In order to improve the bad situation due to the radial inlet, Wang [9] added the splitter vanes in the radial inlet, the results showed that the splitter vanes increased the flow loss, but decreased the distortion of the impeller inlet. By optimizing profile of the vanes, further decreased the separation loss of the flow, which increase the efficiency of the compressor. Using CFD technique, a numerical optimization approach to the radial inlet was developed by Chen [10], the 1.26% improvement of the efficiency was obtained.

As the impeller is a key component of the compressor, the flow field by radial inlet not only affects the performance, but also affects the safety of the impeller. Wang [11] reviewed the centrifugal compressor excited by unsteady pneumatic loads, and pointed out that the flow field by radial inlet should be considered in impeller safety analysis. The research by Han [12] indicated that the impeller has little influence on the radial inlet, but the radial inlet results in aerodynamic load excitation on the impeller. Xin [13] set the guide vane in the radial inlet, the results showed that unevenly distributed guide vanes provided a favorable condition for the impeller, and decreased the aerodynamic load at machine rotating frequency, which improved the safety of the impeller.

All the previous work of radial inlet mainly focused on the fluid analysis and optimization of the structure, which allows for a better flow performance. Adding guide vanes is a good way to improve the flow field in radial inlet. However, the guide vane in the radial inlet generate the wake result in the SRI (Stator rotator interaction) [14], and it is another aerodynamic load excitation on the impeller as well. The trail edge blowing (TEB) is an active control method, which is a way to reduce the wake method. It was proposed by Park [15], and it was widely used in rotating machinery to reduce the SRI. Lewis [16] adopted the TEB method on Francis turbine, the resulted in 43% decrease in global torque variation at the runner passing frequency. Based on the numerical simulations, Xin [17] analyzes the flow field and aerodynamic load under different guide vane angle during the centrifugal compressor operation. It pointed out that the aerodynamic load on impellers has been reduced significantly when the momentumless wake appeared.

In our previous work, although setting guide vane in radial inlet could decrease the aerodynamic load on impeller at machine rotating frequency, it also make the excitation at impeller passing frequency. The
aerodynamic load on it is an important issue related to the compressor safety, which ensures a smooth operation of the compressor and therefore a long lifetime. In order to decrease the excitation at impeller passing frequency, based on the best performs UGV model, the TEB method are adopted on the guide vanes. Through the flow field and aerodynamic load analyzed, the effects of TEB method on the performance of centrifugal compressor and aerodynamic load on impeller are studied using the numerical in this paper.

The remainder of this article is organized as follows. Section 2 describes the compressor with its radial inlet model, the computational consideration and the results of the UGV. Section 3 describes the scheme of trailing edge blowing and the momentumless wake criterion. Compared with the UGV model, the flow field and aerodynamic load results in TEB model are shown in Section 4. Finally, section 5 provides some concluding remarks.

2. GEOMETRY DESCRIPTION AND COMPUTATION CONSIDERATIONS

2.1 Radial inlet centrifugal compressor geometry

Fig. 1 gives the model of the centrifugal compressor with radial inlet, which is used to compress the recycle gas in a methanol process. To handle the large volume flow rate at the given working condition, this compressor adopts the double-sided radial inlet with a compact and symmetrical structure. The first stage of the centrifugal compressor is selected in this numerical study is shown in Fig. 1(a). It includes the radial inlet, the first stage impeller, the vaneless diffuser, the curved channel and the return channel. The radial inlet with unevenly distributed guide vanes (UGV) is shown in Fig. 1(b), which includes two splitter vanes in line with the incoming flow, and the number of the guide vans is 18. The stage feature 13 impeller blades, and the return channel has 28 vanes.

![Figure 1](image.png)

**Figure 1.** The first stage with the radial inlet including (a) profile of the compressor meridian plane, and (b) structure of the radial inlet

The compressor performance characteristics at the operation condition are summarized in Table 1. The parameters refer to the compressor first stage without the seal leakage loss.
Table 1 Compressor performance at operation condition

| Parameter                      | Symbol | Operation Condition |
|-------------------------------|--------|---------------------|
| Flow rate (kg/s)              | \( Q_d \) | 60.95               |
| Rotational speed (rpm)        | \( n \)   | 7982                |
| Intake total pressure (MPa)   | \( P_{in} \) | 6.53                |
| Intake total temperature (°C) | \( T_{in} \) | 45                  |
| Pressure ratio                | \( \varepsilon \) | 1.045               |
| Efficiency (%)                | \( \eta \)   | 89.3                |

2.2 Computational considerations

In this investigation, all simulations are carried out by ANSYS CFX. In order to improve the CFD computing efficiency, we adopt the radial inlet, the first stage, the vaneless diffuser and the return channel as simulation domain. However, the seal area has been ignored in this simulation. The user-defined recycle gas which adopts the real gas parameters is used as the working medium. Note that the simulation model of the compressor adopts the actual size for processing, ensuring the accuracy of describing the flow region.

In the simulation model, the total energy equation is selected for the heat transfer, the high-resolution advection is used for both steady and unsteady simulations, and the second order backward Euler is used for unsteady simulation. To ensure the convergence, the RMS (Root Mean Square) residual target is set to 1e-5 for pressure and velocity components, and the standard of the convergence should be less than 0.2% for the mass flow relative error between the inlet and outlet. Meanwhile, the efficiency and the pressure ratio have to be kept constant. The steady simulation result is used as the initial condition for the unsteady simulation. During the unsteady simulation, the total computing time is set to 10 circles of the impeller rotation for each radial inlet model in order to get stable and precise aerodynamic data on the impeller blades. The time step size is related to the compressor operation condition, i.e., 180-time steps per impeller rotation cycle are chosen for each case and the data in each time step is saved. The loops of each time step coefficient are set from 3 to 10 to ensure convergence.

As the purpose is to obtain the aerodynamic load on the impeller, the monitoring points are used to save the flow data. Due to the radial inlet in front of the impeller, the monitors are placed at the impeller leading edge from hub to shroud on a single blade equidistantly, in a way that 12 points are on the pressure side of the impeller and the remaining 12 points are on the suction side of the impeller, see Fig. 2. All the monitor points are set on the rotating coordinate system, and the speed of the coordinate system is related to the impeller operation condition. After the unsteady simulation, the aerodynamic load signals of the monitoring points are calculated by means of Fast Fourier Transforms (FFT) to obtain the dominant frequency and pulse amplitude.
In this analysis, the $k - \omega$ based SST (Transport of the Turbulent Shear Stress) turbulence model is used. The SST model is based on the equation of turbulence energy and the equation of diffusion rate [18]. This model is designed to give highly accurate predictions of the onset and the amount of flow separation with adverse pressure gradients, and is suitable for the rotating machinery.

In both the steady and unsteady simulations, the boundary conditions are set as follows. The total pressure and total temperature are defined at the inlet, and as the working medium is a compressible fluid, the mass flow rate at the outlet is used. The no-slip condition is set for the blade surfaces and the wall, the rotating and stationary components are connected by the General Grid Interface (GGI). The frozen rotor interface is used for steady simulation because this type of interface does not produce any mixed state in front of or behind the impeller, and the transient rotor-stator interface, which takes into account all the transient flow characteristics and allows for a smooth rotation between components, is used for unsteady.

In this article, for the blade passage, we use the ANSYS turbo-grid to generate the structured meshes. Then the meshes of a periodic passage are copied to the full wheel model. As it is difficult to generate a structured mesh for the radial inlet, we use the commercial mesh generator ICEM to generate a structured/unstructured mesh in this flow domain, see Fig. 3.
Generally, a mesh independent study should be conducted to investigate the effect of mesh density and to determine which mesh refinement is appropriate for simulation. Fig. 4 shows the simulated results at the operation condition for different mesh numbers.

![Graph showing mesh independence verification](image)

**Figure 4. Verification of mesh independence**

It is observed that when simulating with more than 6432000 elements, the simulated results are almost the same and in agreement with the data in Table.1. Thus, considering the simulation accuracy and computing time, the 6432000 meshes with a 23.45° minimum orthogonality is used in subsequent calculations. The number of elements for the individual component are as follows. The radial inlet has 3004210, the first impeller has 752895, the vaneless diffuser has 447525, and the return channel has 2227480. Meanwhile, the values of the Y plus on the blade and the wall of the simulation domain are found to satisfy the requirement of the SST turbulence model.

### 2.3 Results of the UGV model

The steady simulation results offer the overall performance of the compressor and the flow detail in the radial inlet. Fig. 5 gives the contour plots of Mach number, pressure, total pressure and shows the vector plot as well as the streamlines at the outlet of the radial inlet, i.e., the inlet of the first impeller, see section 1-1 in Fig. 1(a).
Figure 5. The distributions of (a) Mach number, (b) pressure, (c) vector, (d) streamline and (e) total pressure at the impeller inlet of the UGV model

After the fluid passes through the splitter vane, the flow direction turns from the radial to the axial. From each contour of the parameters, the flow field of the UGV model shows uniform in the circumference and radial directions. It means that using the guide vane is a good way to reduce the flow losses and improve the flow field distortion in radial inlet. However, the wake of the guide vane leads extra total pressure loss, and makes the impeller generate aerodynamic load at the impeller passing frequency.

Through the unsteady simulation, the frequency domain of the pressure fluctuation of different monitors is calculated by means of the FFT. In this compressor, the main frequencies of the load on impeller leading edge appear at the machine rotating frequency and the impeller passing frequency. For the impeller passing frequency, the 18 guide vanes have to be considered due to the SRI [14]. Table 2 shows the machine rotating frequency and impeller passing frequency of the compressor at operation condition. The parameter \( f_m \) represents the machine rotating frequency and the parameter \( f_i \) represents the impeller passing frequency, being 18 times higher due to the 18 guide vanes.

| Table. 2 Summary of the frequencies of the compressor under the operation condition |
|--------------------------------------|-----------------|---------------|
| Frequency (Hz) | Formula | Value |
| Machine Rotating Frequency | \( f_m = n / 60 \) | 133.03 |
| Impeller Passing Frequency | \( f_i = f_m \times 18 \) | 2394.6 |

Fig. 6 (a) shows the aerodynamic load at the tip of the impeller leading edge in the UGV model. It is observed that there is an obvious excitation at the 18-frequency harmonic, i.e., the impeller passing frequency. It means that the wake of guide vanes results in the aerodynamic load on the impeller at the impeller passing frequency and further results in the impeller fatigue failure. Fig. 6 (b) shows the amplitudes at \( f_m \) and \( f_i \) distribution at the impeller leading edge. There is no distinct rule in amplitudes distribution at \( f_m \) from hub to shroud, and the amplitudes at \( f_i \) increase from hub to shroud, which is because that the streamline distance between the guide vane and the impeller is decrease from hub to shroud, and the wake intensity is stronger in shorter distance. Compared with the amplitudes at \( f_m \), the amplitudes at \( f_i \) are lower, and the amplitudes at \( f_m \) show increasing
trend from hub to the shroud, with the highest value at the shroud region. However, the amplitudes at $f_m$ are important aerodynamic load on the impeller, which cannot be ignored by considering the impeller safety.

![Graphs showing pressure and amplitudes](image)

Figure 6. The aerodynamic load at the impeller leading edge in the UGV model

3. THE SCHEME OF TRAILING EDGE BLOWING AND THE MOMENTUMLESS WAKE CRITERION

As the result shows that the guide vane in the radial inlet will produce wake in front of the impeller, we plan to alleviate it. The active method of trailing edge blowing is a way in theory. Considering the guide vane distribution at radial inlet and the flow field distribution, we set the trailing edge blowing method at the trailing edge of the guide vanes, see Fig. 7. Due to the flow separation near the suction side on left and right guide vanes, the blowing location offsets an area to the suction side.

![Diagram showing the setting location of trailing edge blowing](image)

Figure 7. The setting location of the trailing edge blowing at the guide vanes in radial inlet

By using the TEB method, we will compensate the wake deficit, and the goal is to obtain the momentumless wake state. By adjusting the blowing mass flow, we will get four typical wakes. The criterion of momentumless wake is related to the momentum thicknesses $\theta$, which is calculated by the velocity profiles [15], that is

$$\frac{\theta}{d} = \frac{1}{2} \int_{-\infty}^{\infty} \frac{U}{U_\infty} \left(1 - \frac{U}{U_\infty}\right) d\left(\frac{y}{d}\right)$$  \hspace{1cm} (1)
where \( U_\infty \) represents the main flow velocity, \( d \) represents the guide vane trailing edge width, and \( y \) represents the width of the guide vane passage. When \( \theta/d > 0 \), it represents the pure wake or weak wake, that means there is no blowing or blowing is not enough. When \( \theta/d < 0 \) it represents for the jet, and it means the blowing flow is overmuch. When \( \theta/d = 0 \), it represents the momentumless wake state, which means that the blowing flow compensates the wake deficit just in time.

4. THE ANALYSIS OF TRAILING EDGE BLOWING

4.1 The compressor performance analysis

During the simulation of the TEB, the main boundary condition and the numerical method are the same as that in the UGV simulation case. Furthermore, the mass flow boundary is set at the guide vane blowing locations, and the extra mass flow has been considered at the outlet. By adjusting the blowing mass flow, the momentumless wakes of the guide vane are selected as the TEB results for the subsequent comparison.

Through the simulations with various running conditions, Table 3 gives the blowing mass flow distribution under various conditions. From 0.8 \( Q_d \) to 1.2 \( Q_d \), the needed blowing mass as well as relative flow rate is increased. It is because the high mass flow leads to high main flow velocity, high blowing velocity, high blowing mass flow. Due to the blowing mass flow, the compressor corresponding operation condition has changed, especially in the high flow condition. The TEB method results in 3%—7% mass flow increased at 0.8 \( Q_d \) — 1.2 \( Q_d \) operation condition.

| Compressor operation condition | Blowing mass flow (kg/s) | Relative flow rate (%) | Total mass flow corresponding operation condition |
|-------------------------------|--------------------------|------------------------|-----------------------------------------------|
| 0.8 \( Q_d \)                | 2.02                     | 4.13                   | 0.83 \( Q_d \)                                |
| 0.9 \( Q_d \)                | 2.47                     | 4.51                   | 0.94 \( Q_d \)                                |
| 1.0 \( Q_d \)                | 2.87                     | 4.72                   | 1.05 \( Q_d \)                                |
| 1.1 \( Q_d \)                | 3.48                     | 5.18                   | 1.16 \( Q_d \)                                |
| 1.2 \( Q_d \)                | 4.38                     | 6.00                   | 1.27 \( Q_d \)                                |

Fig. 8 gives the compressor characteristic curves of the UGV and TEB models. The pressure ratio and efficiency are defined as:

\[
\text{Pressure ratio} = \frac{P_{out}}{P_{in}} \quad (2)
\]

\[
\text{Efficiency} = \frac{((\gamma - 1) \ln(P_{out}/P_{in}))}{(\gamma \ln(T_{out}/T_{in}))} \quad (3)
\]

where \( \gamma \) represents the specific heat ratio, \( P_{out} \) and \( T_{out} \), respectively, represent the discharge total pressure, and the discharge total temperature.
From a normalized mass flow of 1.2 down to the 0.8, we find that the guide vane trailing edge blowing in the radial inlet changes the compressor best operation point. When the mass flow is less than that at the design point, the efficiency of the TEB models is higher than that at UGV model. When the operation condition mass flow is equal or higher than the design point, the efficiency of TEB model is lower than that at UGV model. It means that the TEB method makes the efficiency lines left moving.

With the increasing of the mass flow, the pressure ratio shows a decreasing trend. Due to the blowing flow, more mass flow leads to the lower pressure ratio. It is found that the pressure ratio line of the TEB models is lower than that in UGV models from $0.8 \bar{Q}_d$ to $1.2 \bar{Q}_d$. It means that the TEB method makes the pressure ratio lines down moving.

![Figure 8. Comparison of the compressor characteristic performance of the UGV and TEB models](image1)

**4.2 The flow field analysis in radial inlet**

All the simulation results of the TEB refer to the $1.0 \bar{Q}_d$ operating point. In order to obtain the momentumless wake style, the ring lines are set respectively at 0.1, 0.5 and 0.9 span behind the guide vane, see Fig.9. Calculating the velocity profiles on the ring lines by Eq. (2), we could estimate the wake style and adjust the blowing mass flow distribution.

![Figure 9. The locations of the ring lines](image2)
Fig. 10(a) gives the velocity distribution behind the guide vanes at different spans in the UGV model. According to the Eq. (2), the wake style is significantly influenced by the main flow velocity, and the velocity distributions are different in both circumferential and spanwise directions. The velocity is the highest at 180° because it is the nearest place to the inlet of the compressor. Since the flow channel turns direction from the radial to the axial, the velocity is the highest near the hub. The wake intensity is stronger near 90° and 270° at the 0.5 and 0.9 span, near 180° at the 0.1 span. The wake velocity distribution in radial inlet is significantly different in comparison to the axial inlet.

![Velocity Distribution](image1)

Figure 10. The velocity distribution behind the guide vane at different spans of (a) the UGV model and (b) the TEB model

According to the velocity profile, if we want to obtain the momentumless wake style, the needed blowing mass flow at each guide vane is different, and the single guide vane that needed the blowing is different from hub to shroud as well. By adjusting the blowing mass flow, Fig. 10(b) gives the velocity distribution behind the guide vane at different spans in TEB model. By using Eq. (2), it is found that the value of the $\theta/d$ for each wake approaches zero, which means the momentumless wake of the guide vanes is obtained. Compared to the UGV model, the main flow velocity is higher in the TEB model.

Fig. 11 gives several contour plots at section 1-1 for the TEB model. In comparison to the UGV results in Fig. 5, by adding the trailing edge blowing on the guide vanes, the flow field upstream of the impeller has changed a little. Due to the blowing flow, the velocity has increased and the pressure has decreased, the low total pressure region has reduced, especially at the shroud region, which means the wake intensity has decreased.

![Contour Plots](image2)
Figure 1. The distributions of (a) Mach number, (b) pressure, (c) vector, (d) streamline and (e) total pressure at the impeller inlet of the TEB model

Because of the aerodynamic load on the impeller is determined by the total pressure and the flow angle at the outlet of the radial inlet, Fig. 12 gives the flow angle (0° represents the axial direction) and the total pressure ratio (defined as the total pressure divided by the inlet total pressure) distributions of the UGV and the TEB models. The absolute flow angles at the leading edge are close to zero, and total pressure ratios are close to 1, which means the radial inlet performance well. The flow angle and the total pressure ratio show obvious fluctuations at different spans, and the number of fluctuations is equal to the number of guide vanes, which means that the fluctuation is caused by the guide vane wake. There is obviously value change at 0°, which is due to the flow mixing from the left and right sides at top of the radial inlet. Compared with the UGV model, the TEB model shows lower fluctuation intensity due to the trailing edge blowing, especially at the impeller shroud region, which is the region significantly affected by the guide vane wake.

Figure 12. Comparison of the (a) flow angle and (b) total pressure ratio distribution at the impeller leading edge of the UGV and TEB models

4.3 The aerodynamic load analysis

In order to get accurate and stable monitoring data, all the data are chosen after five revolutions in each unsteady simulation case. Fig. 13 presents the comparison of the time history oscillations (the last two cycles) at the tip monitor 11 on two sides of the impeller of the UGV and TEB models.
The pressure data has an obvious periodicity and stability for two models, and each wavelength corresponds to the time of one impeller revolution. Compared to the suction side, the mean pressure on the pressure side is very high, as this side is the working surface. The trend of the pressure data is almost the same of the two models. Each revolution has 18 fluctuations on both sides on the impeller, which is due to the wake of the guide vanes. The fluctuations in the TEB model are obviously weaker than that in the UGV model, especially on the pressure side, which means that the trailing edge blowing has positive effect on wake.

In order to analyze the aerodynamic data, the frequency domain of the pressure fluctuation of different monitors is calculated by means of the FFT. Fig. 14 gives the frequency domain of the aerodynamic load at monitor 11 in two models. Compared to the UGV model, the amplitudes at $f_m$ are a little higher in the TEB model, which is because the extra flow result in the velocity of the main flow has increased, and a little strengthens the non-uniformity along the circumference direction. However, the amplitudes at $f_r$ are significantly lower in the TEB model, and reduction of the value is about 90 percent, which means that the trailing edge blowing has positive effect on the wake, and reduces the aerodynamic load at $f_r$ on the impeller. Meanwhile, the amplitudes at harmonic from 16 to 20 have decreased as well.
Fig. 15 gives the aerodynamic load distribution at $f_m$ and $f_i$ from hub to shroud in two models. The amplitude values at $f_m$ have little difference of the two models; however, the amplitude values at $f_i$ in the TEB model are obviously lower than that in the UGV model, especially at the mid and tip region of the impeller. It means that the trailing edge blowing is an effective way to decrease the wake deficit and reduce the aerodynamic load at $f_i$ on the impeller, and further improves the safety of the impeller.

![Comparison of the amplitude values at (a) $f_m$ and (b) $f_i$ at the impeller leading edge from hub to shroud of the UGV and TEB models](image)

5. CONCLUSION

The radial inlet is one of the most crucial parts of a centrifugal compressor. Combined with the trailing edge blowing method, the UGV model of the compressor inlet is analyzed numerically in this paper. Based on the compressor working condition, the characteristic performance and the flow field of two models (UGV and TEB) are analyzed. Unsteady simulations are conducted to obtain the aerodynamic load on the impeller leading edge. The consideration of the aerodynamic load due to the trailing edge blowing method could help the designer to analysis similar questions. According to the simulation results, the conclusions are listed below:

1. Setting guide vanes in the radial inlet is a good way to improve the flow field, decrease the flow losses and the distortion upstream of the impeller. However, the guide vanes increase the total pressure loss due to the wake, and the wake results in the aerodynamic load excitation at $f_i$ on the impeller, which affects the safety of the impeller.
2. The trailing edge blowing is an active control method, the wake style is related to the velocity profile behind the guide vane. In the UGV model, the main flow velocity behind the guide vanes is different along the spanwise and circumference directions. Therefore, by adjusting the blowing mass flow in both directions, the momentumless wakes are obtained.
3. Compared to the UGV model, the characteristic line has changed due to the extra blowing mass flow in the TEB model. It makes the efficiency lines left moving and the pressure ratio lines down moving. The blowing flow leads the main flow increased as well.
4. The flow field at impeller inlet has improved in the TEB model, the total pressure and the absolute angle fluctuations are lower due to the trailing edge blowing. By the FFT, the TEB model shows a little influence on the aerodynamic load at $f_m$, but shows obviously lower at $f_i$, especially at the mid and the tip region of the impeller. It means that the trailing edge blowing is an effective way to improve the safety of the impeller.
6. ACKNOWLEDGMENTS

The authors appreciate the supported by Fundamental Research Funds for the Central Universities (3132020122), National Natural Science Foundation of China (No. 52006021) and Natural Science Foundation of Liaoning Province (No. 2020-BS-069)

REFERENCES

[1] Engeda A, Kim Y, Aungier R. 2003 The Inlet Flow Structure of a Centrifugal Compressor Stage and Its Influence on the Compressor Performance. ASME Journal of Fluids Engineering, 125: 779-785.
[2] Tan J, Wang X, Qi D, 2011 et al. The effects of radial inlet with splitters on the performance of variable inlet guide vanes in a centrifugal compressor stage[J]. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 225(9):2089-2105
[3] Tan J, Wang X, Liu J, 2014 et al. Review of the Study on the Radial Inlet of Centrifugal Compressor[J]. Compressor, Blower & Fan technology, (4):63-70
[4] Long F, Tan J. 2010 The Influence of Radial Inlet Volute on the Stage Performance of Centrifugal Compressor[J]. Compressor, Blower & Fan technology, (2):7-11
[5] Dong F, Yu Y, Zhu X. 2012 Influcences of the Radial Inlet Chamber on the Characteristics of a Centrifugal Compressor [J]. Fluid Machinery, 40(11):15-20
[6] Wu D, Wang L, Chen J, 2010 et al. The Design Method of Radial Inlet Volute in Centrifugal Compressor[J]. Compressor, Blower & Fan technology, (1):23-28
[7] Han, F., Qi, D., Tan, J., Wang, L., 2012 et al. Experimental and Numerical Investigation of the Flow Field in the Radial Inlet of a Centrifugal Compressor[C]. Proc. ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, American Society of Mechanical Engineers, 831-845.
[8] Feng Z, Gao Y, Song Q. 2016 Numerical Simulation Research of the Impact of Radial Inlet Chambers on the Performance of Impellers[J]. Compressor, Blower & Fan technology, (3):37-41
[9] Wang R, Qi D, Wang X, 2009 et al. Flow loss for the Radial Suction Chamber of Centrifugal Compressor improved Design[J]. Chinese Journal of Applied Mechanics, 26(3):437-443
[10] Chen Z, Gu C, Shu X, 2010 et al. Shape Optimum Design for Centrifugal Compressor Radial Inlet Based on CFD Technique[J]. Journal of Mechanical Engineering, 46(14): 124-129
[11] Wang Y, Liu Y, Guo T. 2016 Research Progress in Unsteady Flow Characteristics and Dynamic Response of Centrifugal Compressor Impellers[J]. Compressor, Blower & Fan technology, (5):81-87
[12] Han F, Mao Y, Tan J. 2017 Effects of Flow Loss and Outlet Distortions Caused by Radial Inlet[J]. Journal of Engineering Thermophysics, 10(1):2125-2129
[13] Xin J, Wang X, Zhou L, 2016 et al. Numerical Investigation of the Flow Field and Aerodynamic Load on Impellers in Centrifugal Compressor with Different Radial Inlets[C] Asme Turbo Expo: Turbine Technical Conference & Exposition. GT2016-57180
[14] Dring, R., Joslyn, H., Hardin, L., Wagner, J. 1982 Turbine rotor-stator interaction[J]. Journal of Engineering for Gas Turbines and Power, (4): 729-742.
[15] Park W, Cimbala J. 1991 The effect of jet injection geometry on two-dimensional momentumless wakes [J]. Journal of Fluid Mechanics, 224(2):29-47
[16] Lewis B J, Cimbala J M, Wouden A M. 2012: Investigation of distributor vane jets to decrease the unsteady load on hydro turbine runner blades. [C]. Proceedings of the Proc 26th IAHR Symposium on Hydraulic Machinery and Systems, Beijing China. 2006.

[17] Xin J, Wang X, Chen X, 2019 et al. Numerical investigation of variable inlet guide vane with trailing-edge blowing in centrifugal compressor[J]. Journal of Harbin University, 2018, 39(6):1006-1011

[18] Ansys, A. F., "19.0 Theory Guide," ANSYS inc.