Influence of volute design on flow field distortion and flow stability of turbocharger centrifugal compressors

Zhenzhong Sun¹, Xinqian Zheng¹, Zelin Linghu¹, Tomoki Kawakubo², Hideaki Tamaki² and Baotong Wang²

Abstract
Volute-induced distortion of the flow field remarkably exacerbates the flow stability of turbocharger centrifugal compressors; hence, reducing the flow field distortion by modifying the volute design is of great significance. This study investigated the influence of volute design on the flow field distortion and flow stability of turbocharger centrifugal compressors. Volumes with different throat areas and different area-to-radius ratio ($A_u/R_m$) distributions were designed, and their performances were analyzed via both numerical and experimental methods. The results show that the throat area and the $A_u/R_m$ distribution significantly influence the volute-induced pressure distortion. By modifying the volute design, the amplitude of the pressure distortion can be reduced from 8.82% to 3.22%, and the stable flow range can be extended from 49.8% to 58.7%. Additionally, to reduce flow field distortion, the throat area should be appropriate to guarantee a match between the volute and upstream components, and the $A_u/R_m$ distribution near the volute tongue should have a moderate slope. This study provides design methods for reducing flow field distortion by modifying the volute, and it also verifies the belief that smaller flow field distortion is beneficial to the flow stability of compressors.

Keywords
Turbochargers, centrifugal compressor, volute, distortion, flow stability, stable flow range

Date received: 18 June 2017; accepted: 8 November 2017

Introduction
Energy saving and emission reduction are two important issues in the automotive industry, and turbocharging is a promising way to achieve both of these goals. Turbocharging helps to significantly downsize internal combustion engines, thereby reducing fuel consumption and emission levels.¹ It can also increase the limiting torque curve and torque back-up, and increase the engine rate.² The centrifugal compressor, which is an essential subassembly of a turbocharger, contains three main components: the impeller, the diffuser, and the volute. It has been proven that the flow in a turbocharger centrifugal compressor is intrinsically non-axisymmetric because of the non-axisymmetric geometry of the volute,³, ⁴ especially when the compressor operates under off-design conditions, where the volute would act as either a nozzle (high mass flow rate) or a diffuser (low mass flow rate). As a result, pressure distortion along the circumferential direction is induced, which has been proven to remarkably exacerbate compressor performance and stability.⁵ Considering that the stability of turbocharger centrifugal compressors is a major concern in current internal combustion engines, owing to the current downsizing trend,⁶ volute design has gained increasing attention as it significantly influences turbocharger performance.

Most investigations on volumes focus on geometric parameters. According to Ayder et al.,⁷ five geometric parameters of volumes are closely related to compressor performance; these parameters are the cross-sectional area of the volute passage, the shape of the cross-section, the radial and axial positions of the volute passage, and the shape of the volute tongue. Studies by Bowerman and Acosta⁸ and Stiefel⁹ found that the cross-sectional area of a volute passage had...
noticeable effects on compressor flow stability and efficiency. By contrast, investigations\(^\text{10, 11}\) showed that the shape of the cross-section had little influence on compressor performance. Additionally, studies on the radial position of a volute by Mishina and Gyobo\(^\text{11}\) and Stiefel\(^\text{9}\) indicated that a smaller radial position often led to a higher generation loss. The axial position of a volute is often discussed in terms of the symmetric volute and overhung volute; Qiang et al.\(^\text{12}\) and Chen et al.\(^\text{13}\) compared the performances of these two kinds of volute via numerical methods, and their results showed that a much stronger mixing process occurred with the symmetric volute than with the overhung volute, resulting in higher losses. In addition, the volute tongue is another important geometric parameter for volute design. Usually, a volute tongue with a sharp leading edge has a good volute performance at the design point. However, such tongues lack tolerance to flow condition variation; thus, volute tongues are often designed to be a smooth transition. Xu and Müller\(^\text{14}\) noticed an improvement in the stable flow range and efficiency under off-design conditions after smoothing the leading edge of a volute tongue, and Reunanen\(^\text{15}\) experimentally found that the efficiency of a compressor decreased by 0.1% at the design point, while the efficiency increased by 0.8–0.9% at off-design points, when the leading edge of a volute tongue was smoothed. Apart from the five geometric parameters mentioned, studies focusing on other parameters have also been carried out. The work of Brown and Bradshaw\(^\text{10}\) showed that the roughness of a volute passage wall hardly affected the compressor performance, while Jiao et al.\(^\text{16}\) introduced a double-volute design, which was reported to be able to reduce the negative effects of a non-axisymmetric flow field on upstream components; at the same time, a wider stable flow range and a higher efficiency were obtained.

Satisfactory improvements have been achieved in the investigation of the influence of volute geometric parameters on compressor performance. It can be concluded that the cross-sectional area and the volute tongue significantly affect the compressor performance and that an overhung volute can achieve a lower loss coefficient. However, most studies only focus on the loss or efficiency change of the compressor stage; few works illustrate the influence of volute geometric parameters on flow field distortion and flow stability. In addition, few works discuss the design methodology of a volute with the goal of reducing volute-induced distortion and improving the flow stability of turbocharger centrifugal compressors. This study investigates the influence of volute design on the flow field distortion and flow stability of turbocharger centrifugal compressors via numerical and experimental methods, the results of which enrich the knowledge related to volute design and offer insight into the relationship between the geometry, flow field distortion, and stability of a compressor.

### Volute-induced distortion and volute design

The ultimate objective of this study is to improve the flow stability of compressors by modifying the volute geometry; volute-induced distortion plays a significant role in this process. This chapter will first introduce the relationship between volute-induced distortion in the flow field and the flow stability of turbocharger compressors, and then theoretical analysis will be carried out to determine critical geometric parameters that significantly affect the flow inside a volute passage. Later, a volute will be modified according to the theoretical analysis, and then volute design schemes will be presented.

### Relationship between volute-induced distortion and flow stability

As mentioned, the influence of the volute on upstream components and the whole compressor stage, especially the influence on compressor flow stability, is, in part, due to the flow field distortion induced by the volute. It is widely accepted and believed that it is possible to obtain a wider stable flow range for a compressor with smaller flow field distortion in compressors, i.e., the flow field distortion deteriorates the flow stability of compressors; this has been proved directly or indirectly by many studies. Reid\(^\text{17}\) experimentally found that the stable flow range decreased when flow field distortion existed in the incoming flow, particularly at high rotational speeds. Hynes and Greitzer\(^\text{18}\) established a model to examine the influence of the flow field distortion in instability, and found that increasing distortion amplitude affected the surge margin. Numerical and experimental studies conducted by Sorokes et al.\(^\text{19}\) suggested that the volute would cause a non-uniform pressure distribution in the diffuser, which could extend upstream to the impeller, influencing premature stall and the impeller stability. Zheng et al.\(^\text{5}\) compared the stable flow ranges of compressors with and without a volute via a numerical method, and their results indicated that a compressor with a volute has a narrower stable flow range at all investigated rotational speeds because of the volute-induced distortion. Another work focused on the influence of the distortion amplitude, which showed that the higher the amplitude of the volute-induced distortion, the narrower the stable flow range.\(^\text{20}\) The work of Zheng et al.\(^\text{21}\) provided theoretical evidence that proves the relationship between flow field distortion and compressor flow stability; a theoretical model taking the flow field distortion into account was established to explain the flow instability behavior of compressors, and it was noted that the flow field distortion caused flow instability at certain flow passages in advance, hence triggering the whole compressor to stall or surge at a higher mass flow rate, narrowing the stable flow range and exacerbating the compressor flow stability. Additionally, non-axisymmetric flow control
methods, such as the non-axisymmetric casing treatment, were reported to be able to adapt the flow field distortion and extend the stable flow range, which also proves the belief that flow distortion is harmful to flow stability, to some extent.

Because flow field distortion exacerbates the flow stability of compressors, volute geometry is modified in this investigation to reduce volute-induced flow field distortion and improve flow stability. A numerical method is applied to analyze the influence of volute design on flow field distortion, and those volute designs that reduce the distortion are selected for further analysis via experiments; furthermore, the stable flow ranges of compressors with the selected volutes are analyzed and compared in detail to assess the influence of volute design on flow stability.

**Theoretical analysis of flow inside volute passage**

Before the modification of volute geometry, it is necessary to analyze the flow condition inside a volute passage and find the critical geometric parameters. A sketch of the volute model used for theoretical analysis is given in Figure 1. If the wall friction can be neglected, the angular momentum should be conserved; therefore

\[
R \cdot V_t = R_3 \cdot V_{t3} \quad \Rightarrow V_t = \frac{R_3 \cdot V_{t3}}{R}
\]

where \( R \) is the radius, \( V_t \) is the tangential velocity, and the subscript 3 donates the streamwise position of the diffuser outlet (or the volute inlet).

Considering the circumferential position \( \theta \), the mass flow rate passing through the cross-section \( A \) can be expressed as

\[
m = \int \rho V_t \cdot dA_\theta = \int \rho R_3 V_{t3} \cdot \frac{dA_\theta}{R}
\]

From circumferential position \( \theta \) to \( \theta + d\theta \), the increased mass flow rate passing through the cross-sectional area of the volute is

\[
dm = d \left( \frac{\rho R_3 V_{t3} \cdot dA_\theta}{R} \right) = \rho_3 V_{t3} b_3 \cdot dl
\]

Within the range from \( \theta \) to \( \theta + d\theta \), it is appropriate to regard \( V_{t3} \) as constant, since \( d\theta \) is small. Considering that \( dl = R_3 d\theta \), we have

\[
d \left( R_3 V_{t3} \cdot \frac{\rho \cdot dA_\theta}{R} \right) = \rho_3 V_{t3} b_3 R_3 \cdot d\theta
\]

Therefore, the flow inside the volute is controlled by

\[
d \left( V_{t3} \cdot \frac{\rho \cdot dA_\theta}{R} \right) = \rho_3 V_{t3} b_3 \cdot d\theta
\]

It is reasonable to assume that the density of the flow passing across section \( A \) is constant (denoted \( \rho_m \), where "m" denotes the equifinal value). Then, equation (3) can be rewritten as

\[
\frac{d}{d\theta} \left( V_{t3} \cdot \frac{A_\theta}{R_m} \right) = \frac{V_{t3} b_3 \cdot \rho_3}{\rho_m}
\]

where \( A_\theta/R_m = \int R^{-1} \cdot dA_\theta \) is a geometric parameter related to the cross-section of the volute passage.

**Volute design schemes**

Equation (4) expresses the relationship between the flow and the area-to-radius ratio \( A_\theta/R_m \) (cross-sectional area divided by equifinal radius) distribution along the circumferential direction. Considering the ideal case, where the circumferential difference in the flow field can be neglected, a derivative of equation (4) can be calculated, i.e., equation (5). According to equation (5), \( A_\theta/R_m \) should change linearly along the circumferential direction; in this case, the area of the volute throat (shown on the right side of Figure 1) is highly significant, since a larger throat area indicates a larger slope \( k \) of the circumferential distribution of \( A_\theta/R_m \). Because the slope \( k \) is related to the operating conditions of the compressor (i.e., related to \( \rho \) and \( V \)), the slope \( k \) determines the match point, i.e., the throat area determines the match between the volute and the upstream components. If the influence of circumferential distortion cannot be neglected, equation (4) indicates that the circumferential distribution of \( A_\theta/R_m \) is no longer linear and is closely related to the flow field; in this case, the detailed \( A_\theta/R_m \) distribution along the circumferential direction also has significant impact on the volute flow condition

\[
\frac{d}{d\theta} \left( A_\theta/R_m \right) = b_3 \frac{R_3}{\rho_m} \frac{V_{t3}}{V_{t3}} = \text{const} = k
\]

According to theoretical analysis, the throat area and the \( A_\theta/R_m \) distribution are two critical parameters affecting the volute performance. Based on this, two series of volutes are designed. One series (TA series) of volutes is designed to have a linear \( A_\theta/R_m \) distribution, with the throat area of each one being different; their \( A_\theta/R_m \) distributions are illustrated in Figure 2. It should be noted that all these volute designs are based...
on a real volute of a turbocharger product, and that the TA100 design shown in Figure 2 has a throat area equal to that of the basic volute. The throat area of TA70 is 70% of that of the basic volute, etc. The other series (NL series) of volutes has non-linear $A_u/R_m$ distributions, and their throat areas are set equal to that of the TA80 design, as shown in Figure 3. The slopes of the $A_u/R_m$ distributions of NL01 and NL02 are moderate within most of the circumferential region and become steep when close to the volute tongue (circumferential position from 300° to 360°), while the slope of NL03 is large within most of the circumferential region and becomes smaller when close to the volute tongue. The remaining volute designs (NL04 and NL05) have small $A_u/R_m$ distribution slopes within the region near the volute tongue (60° to 120° and 300° to 360°), while the slopes within other circumferential regions are larger.

**Analysis of flow field distortion via numerical method**

In this section, the two series of volute designs introduced in the previous section are analyzed via a numerical method to investigate the influence of volute design on flow field distortion. First, the numerical method will be introduced, and then the volute-induced distortions of both series of volute designs under certain operating conditions will be compared. Finally, several volute designs will be selected for further investigation.

**Simulation model and numerical setup**

During the numerical analysis process, a turbocharger centrifugal compressor product whose volute is replaced with new volute designs is simulated via a numerical method. The turbocharger is used in heavy-duty diesel engines, and detailed geometric specifications of the compressor are listed in Table 1.

As specified in Table 1, the simulated compressor has seven main blades and seven splitter blades, and a vaneless diffuser is used downstream of the impeller. Additionally, the maximum rotational speed of the compressor is 115,000 rpm (denoted N115%), at which the stage pressure ratio is over 4.0.

The simulation model used in this paper contains an impeller, a vaneless diffuser, and a volute, and the finished mesh of different components is shown in Figure 4. The fluid domains of the impeller and the diffuser are spatially discretized by a structured mesh using ANSYS TurboGrid commercial software. Nevertheless, because of the complex geometry, the volute is spatially discretized by an unstructured mesh with prism layers near the solid wall, with the unstructured mesh being generated using ANSYS ICEM.
commercial software. The finished mesh of the entire 360° annulus contains nearly five million mesh elements.

The three-dimensional Reynolds-averaged Navier–Stokes equations are solved using ANSYS CFX commercial software, which requires much less computational resources. Regarding the boundary conditions, the total temperature, the total pressure, and the flow direction are imposed as the inlet boundary condition, while the static pressure or the mass flow rate is imposed as the outlet boundary condition when operating under near-choke or near-surge conditions, respectively. In addition, all the solid walls are set as non-slip and adiabatic. The two-equation \( k-\epsilon \) turbulence model is selected for turbulence closure. Another important aspect of the simulation setup is the interface setting between the rotating impeller and the stationary diffuser; in this paper, the frozen-rotor method is selected for the steady simulation, while the transient rotor stator is imposed at the interface for the unsteady simulation.

To guarantee the accuracy of the numerical method, the mesh should meet the requirement of the application of the selected turbulence model. Representing the non-dimensional wall distance along the wall, “\( y^+ \)” is crucial for accurately capturing the phenomena that occur at the boundary layer and has a significant influence on the accuracy of the numerical simulation. The suggested range of \( y^+ \) for the \( k-\epsilon \) turbulence model is from 30 to 300, and it is accepted for \( y^+ \) to be within the range 10 to 300. The \( y^+ \) distribution shown in Figure 5 indicates that the \( y^+ \) values at all solid walls are within this range (the maximum \( y^+ \) value is about 180; this is not shown in Figure 5), indicating that the mesh used in this paper is quite suitable for the \( k-\epsilon \) turbulence model.

**Static pressure distortion of different volute designs**

Under off-design conditions, the volute often induces pressure distortion at the volute inlet, which significantly affects the compressor performance, especially the flow stability. To determine those volute designs that suppress distortion, the most used rotational speed (N80%) is analyzed using a steady simulation method.

Although the *frozen-rotor* method for the rotor–stator interface is believed to be inadequate for capturing the behavior of circumferential distortions, it is used in this study to estimate the pressure distortion at the volute inlet because the distortion at the volute inlet is mainly determined by the volute itself and the location is far from the interface (i.e., the vaneless diffuser is between the interface and the volute inlet); a comparison of the distortions by the steady and unsteady simulation results supports the use of the *frozen-rotor* method.

Figure 6 shows the pressure distortions at the volute inlet when the compressor is operating at a low mass flow rate (near-surge); the blue solid line represents the prediction of the steady simulation with the *frozen-rotor* method, the black dotted line represents a phase-locked result of the unsteady simulation, and the red dash-dotted line represents the time-averaged result of the unsteady simulation. It can be noted that the tendency of the three distortions is the same; in particular, the *frozen-rotor* result and *time-averaged* result are almost overlapping. Therefore, the steady simulation with the *frozen-rotor* is capable of accurately capturing the pressure distortion at the volute inlet, including the phase, the distribution and the amplitude, indicating that it is appropriate to compare the pressure distortions at the volute inlet by performing the steady simulation with the *frozen-rotor* method.

The TA series of volute designs are analyzed by performing a steady simulation to investigate the influence of throat area on the volute-induced distortion. The static pressure distortions at the volute inlet under both near-surge and peak-efficiency conditions are illustrated in Figures 7 and 8, respectively. For the volute designs from TA70 to TA120, the slope of the \( A_0/R_m \) distribution increases and the amplitude of the pressure distortion at the volute inlet increases under the near-surge condition; a similar tendency can be found when the compressor operates under the peak-efficiency condition. The only difference between these two conditions is that the distortion amplitude of the TA70 volute design is smaller than that of TA80 under the

![Figure 5. y+ distribution at all solid walls.](Image)

![Figure 6. Comparison of distortions at volute inlet predicted by steady and unsteady simulations.](Image)
near-surge condition, while it is larger under the peak-efficiency condition. Another phenomenon that should be noted is that, under both near-surge and peak-efficiency conditions, changing the throat area of the volute hardly affects the phase of the volute-induced distortion at the volute inlet. Therefore, it can be concluded that, within an appropriate range, changing the throat area of the volute makes it possible to suppress the pressure distortion at the volute inlet without changing the phase of the circumferential distortion, but the throat area should be appropriate to ensure the match between the volute and upstream components.

Similarly, for volute designs of the NL series, the $A_u/R_m$ of which are distributed non-linearly along the circumferential direction, the comparison of static pressure distortions at the volute inlet is presented in Figure 9 (near-surge condition) and Figure 10 (peak-efficiency condition). For volute designs with $A_u/R_m$ distribution slopes that are small in regions near the volute tongue (30° to 90°), such as NL02, NL04, and NL05, the local pressure distortion is reduced significantly under both near-surge and peak-efficiency conditions, it is found that TA80 is the best design among the TA series, while NL04 is the best design of the NL series, so these two volute designs, together with TA100, are selected for further investigations.
volute tongue (as shown in Figures 6 to 10). This method may be combined with a flow control technique put forward by Zheng et al.\textsuperscript{22} and Tomaki et al.\textsuperscript{23} who successfully increased the stable flow range of centrifugal compressors via a non-axisymmetric casing treatment.

**Distortion comparison at different rotational speeds**

Three volute designs, \textit{TA100}, \textit{TA80}, and \textit{NL04}, are selected according to static pressure distortion at the volute inlet at N80\% rotational speed for further study. In this section, a distortion comparison between those three designs is carried out at three streamwise locations and different rotational speeds. The three locations, i.e., the volute inlet, the diffuser inlet, and the impeller inlet, are illustrated in Figure 11. The three rotational speeds are a low rotational speed (N50\%), a medium rotational speed (N80\%), that was discussed earlier, and a high rotational speed (N115\%). For each rotational speed, the point of comparison is the near-surge point, with the same mass flow rate, and the result is obtained via unsteady simulations.

Figure 12 shows the time-averaged pressure distortion obtained by conducting an unsteady simulation at three comparison locations. At the volute inlet, it can be seen that, for all three rotational speeds, the tendency and the phase of the pressure distortion are similar among the three volute designs, while the amplitude of the pressure distortion is different. At all three rotational speeds, the amplitude of \textit{TA100} is the largest, while the amplitude of \textit{NL04} is the smallest, and the amplitude of \textit{TA80} is slightly larger than that of \textit{NL04}. The distortion distributions along the circumferential direction at the diffuser inlet and at the impeller inlet have the same tendency as that at the volute inlet; i.e., at both the impeller inlet and the diffuser inlet, the

![Figure 11. Locations of comparison for the pressure distortion.](image)

![Figure 12. Pressure distortion comparison between three selected volute designs at three measurement positions.](image)
The TA100 volute design has the largest distortion amplitude, while the NL04 volute design has the smallest amplitude. According to the definition of the distortion amplitude given in equation (6), when focusing on the most used N80% rotational speed, it can be seen that the amplitude of the distortion can be reduced from 8.82% (TA100) to 3.22% (NL04) at the volute inlet; from 12.5% (TA100) to 6.12% (NL04) at the diffuser inlet and from 1.24% (TA100) to 0.67% (NL04) at the impeller inlet

\[ \text{Amp} = \frac{P_{\text{max}} - P_{\text{min}}}{P_{\text{ave}}} \times 100\% \] (6)

From this analysis, it can be concluded that changing the \( A_u/R_m \) distribution and its slope can potentially reduce the volute-induced pressure distortion and make the flow field more uniform. Additionally, suppressing the distortion in the flow field is beneficial to the flow stability. Thus, different volute designs will affect the stable flow ranges of compressors; this is investigated by conducting experiments, which are discussed next.

Analysis of flow stability via experimental method

In this chapter, the experimental test rig will be introduced first, and then the test results of the three selected volutes will be presented and analyzed to show the flow stability improvements obtained as a result of modifying the volute.

Experimental setups

The test rig facility is illustrated in Figure 13. The compression system is composed of a centrifugal compressor and inlet and outlet pipes together with an outlet value. During the experiment, pressurized air is heated in a combustion chamber in front of the turbine, and the turbine extracts energy from the high-pressure and high-temperature gas and drives the compressor at the required rotational speed. The flow parameters are measured at both the inlet and outlet sides of the compressor. Pressure and temperature probes are installed at section S1 and section S2 to measure the total and static pressure and temperature, and the outlet mass flow rate is measured by probes located at section S2. Additionally, the rotational speed, ambient pressure, and temperature are also determined. All data acquired are analyzed to identify compressor operating conditions and to obtain the compressor performance.

Three compressors with different volutes (TA100, TA80, and NL04) were tested. The test environment is identical for the three compressors, and the impeller and the diffuser, as well as the inlet and the outlet pipes, are also identical. For each compressor, seven rotational speeds, from N45% (45,000 rpm) to N115% (115,000 rpm), were tested, and six operating points, including the choke point and the surge point, were tested at each rotational speed. The choke point is defined as the operating point at which the efficiency is 59.5% ± 0.3%, while the surge point is fixed at the point where the standard deviation of the inlet pressure is larger than 8% (the scientificity and reliability of this method for judging the surge point were proved by Liu and Zheng24).

Figure 13. Test rig facility and related geometric parameters.

Figure 14. Pressure ratio comparison of the three tested compressors.

Experimental performance comparison of three selected volute designs

Owing to the inability of the numerical method in accurately capturing the surge point, one of the primary purposes of the experiment is to determine the surge point at each tested rotational speed, and then the improvement of the stable flow range obtained by modifying the volute can be confirmed. The experimental performances of the three compressors with selected volute designs are shown in Figures 14 and 15 in terms of the total pressure ratio and isentropic efficiency, respectively; the definitions of these two parameters are given in equations (7) and (8), respectively. Moreover,
the mass flow rate and the rotational speed are corrected for standard atmosphere and temperature ($p_{0sa} = 101,300$ kPa and $T_{0sa} = 298$ K), as shown in equations (9) and (10), respectively. Except for the highest rotational speed (N115%), the pressure ratio performance shown in Figure 14 indicates that there is little influence on the total pressure ratio at low and middle rotational speeds. At the highest rotational speed, the total pressure ratios of $TA80$ and $NL04$ are higher than that of $TA100$; this is because the volute throat areas of $TA80$ and $NL04$ are smaller, which is good for achieving a better match at high rotational speeds.

\[
\text{Total pressure ratio : } \pi = \frac{p_{0\text{out}}}{p_{0\text{in}}} \tag{7}
\]

\[
\text{Isentropic efficiency : } \eta = \frac{T_{0\text{in}} \cdot (\frac{\pi^{1.4}}{\pi_{1.4}^{1.4}} - 1)}{T_{0\text{out}} - T_{0\text{in}}} \times 100\% \tag{8}
\]

\[
\text{Corrected mass flow rate : } m_{\text{cor}} = m \cdot \sqrt{\frac{T_{0\text{in}}}{p_{0\text{in}}}} \tag{9}
\]

\[
\text{Corrected rotational speed : } N_{\text{cor}} = N \cdot \sqrt{\frac{T_{0\text{in}}}{T_{0\text{in}}}} \tag{10}
\]

The efficiencies of the three compressors are different. As shown in Figure 15, at all tested rotational speeds, the efficiency of $NL04$ is higher than that of the other two, with $TA100$ having the lowest efficiency, and the difference in peak efficiency can be as high as 1.5%. Therefore, the new volute design is beneficial to the stage efficiency; this benefit is achieved by changing the $A_{v}/R_{m}$ distribution and its slope. At the highest rotational speed (N115%), the difference in peak efficiency between $NL04$ and $TA100$ can be as high as 2.5%, and this efficiency improvement is the main reason for the pressure ratio change at high speeds, as shown in Figure 14.

As expected, modifying the volute has a significant influence on the flow stability of a compressor, as indicated by the surge boundary lines shown in Figure 14. Compared with $TA100$, the surge boundary lines of $TA80$ and $NL04$ move to the left, especially at middle and high rotational speeds. At the same time, the change in the choke line of the three compressors is quite small; thus, the stable flow ranges of the compressor are very different. Using the definition of stable flow range given in equation (11), the biggest difference in stable flow range appears at the N80% rotational speed, where the stable flow range of $TA100$ is approximately 49.8% and the stable flow range of $NL04$ is approximately 58.7%, a relative improvement of up to 17.74% is achieved. It should be noted that the N80% rotational speed is the most used rotational speed for this turbocharger compressor (the pressure ratio is over 2.0); thus, modifying the volute is of great significance for a turbocharger centrifugal compressor because of the potential flow range extension.

\[
\text{Stable flow } = \left( \frac{m_{\text{choke}} - m_{\text{surge}}}{m_{\text{choke}}} \right) \times 100\% \quad \text{at } N = \text{ const} \tag{11}
\]

**Conclusions and discussion**

The objective of this study is to investigate the influence of volute design on the flow field distortion and flow stability of turbocharger centrifugal compressors. Both numerical and experimental investigations were carried out to analyze the performances of different volute designs with different throat areas and different $A_{v}/R_{m}$ distributions. The results indicate that the throat area and the $A_{v}/R_{m}$ distribution, which significantly influence the volute-induced pressure distortion under both near-surge and peak-efficiency conditions, are two critical geometric parameters for volute design. Numerical analysis shows that the amplitude of the pressure distortion at the volute inlet can be reduced from 8.82% to 3.22% by modifying these two critical parameters, while the phase of the pressure distortion is hardly changed. Additionally, the improvement in the stable flow range is verified by experimental investigations, which proves that the stable flow range is extended from 49.8% to 58.7% at 80% rotational speed. Experimental results also show that obvious improvements in stable flow range are obtained at middle and high rotational speeds, although the surge line
at low speeds remains the same, or moves slightly toward the right side.

However, modifying the volute design alone seems inadequate in completely suppressing the flow field distortion; it is challenging to avoid an abrupt change in the static pressure distribution in regions near the volute tongue. At the same time, non-axisymmetric flow control methods that can adapt for a distorted flow field, such as a non-axisymmetric casing treatment, may help in further reducing the flow field distortion and improving the flow stability of compressors. Therefore, a combination of volute design modification and the non-axisymmetric flow control method is recommended to obtain a more satisfactory flow field and stability improvement of turbocharger centrifugal compressors.

Declaration of conflicting interest
The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding
The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was supported by the National Natural Science Foundation of China (grant number 51176087) and the IHI Corporation, Yokohama, Japan.

References


Appendix

Notation

- $A$: cross-sectional area of the volute passage
- $A_{up}$: amplitude of the static pressure distortion
- $b$: width of vaneless diffuser outlet
- $k$: slope of the area-radius ratio $A_b/R_m$ distribution
- $l$: circumferential arc length
- $m$: mass flow rate
- $N$: compressor rotational speed
- $p$: static pressure
- $p_0$: total pressure
- $R$: radius
- $T_0$: total temperature
- $V$: absolute velocity
- $\gamma$: ratio of specific heat capacities
- $\eta$: isentropic efficiency
- $\theta$: circumferential angle or position
- $\pi$: total pressure ratio
- $\rho$: density

Subscripts

- $3$: volute inlet (or vaneless diffuser outlet)
- ave: average value
- choke: choke condition
- in: inlet of the compressor stage
- m: equifinal value
- max: maximum value
- min: minimum value
- out: outlet of the compressor stage
- r: radial component
- sa: standard atmosphere
- surge: surge condition
- t: tangential component