Influence of rotor–stator component partition structure on the aerodynamic performance of centrifugal compressors

Chunyang Li a, Chunjun Jia b, Mao Shi c and Qi Sun d

a School of Energy and Power Engineering, Dalian University of Technology, Dalian, People’s Republic of China; b Key Laboratory of Ocean Energy Utilization and Energy Conservation of Ministry of Education, Dalian University of Technology, Dalian, People’s Republic of China; c Dalian Zhongyi Turbin Technology Co., Ltd., Dalian, People’s Republic of China; d Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing, People’s Republic of China

ABSTRACT
The partition structure (PS) between the impeller and stationary blade of a centrifugal compressor considerably influences a compressor’s performance. Based on computational fluid dynamics, this study constructed a coupling calculation model for the main flow, wheel cover, wheel disk side partition clearance, and sealing structure. Furthermore, we examined the influence that different PS shapes (i.e. circular arc, trapezoidal, and rectangular) and radial clearances (2, 4, and 6 mm) at the entrance of the cavity between the rotor and stator components have on the aerodynamic performance and loss mechanism of the centrifugal compressor using a numerical method. The influence that the PS has on the transition zone between the impeller outlet and stator inlet sections was analysed in detail. According to the results, when the PS was in a circular arc form, with a clearance of 2 mm, it achieved the highest efficiency, which increased the efficiency of the original model by 0.99% and the total pressure ratio by 1.23%. Moreover, an examination of the main flow interaction provided an enhanced insight into the microflow phenomenon in centrifugal compressors.

ARTICLE HISTORY
Received 14 May 2019
Accepted 14 September 2019

KEYWORDS
Multistage centrifugal compressor; rotor–stator components; partition structure; aerodynamic performance; loss mechanism

Introduction
Centrifugal single-shaft multistage compressors are frequently used in the oil and gas industry (Rube, Rossbach, Wedeking, Grates, & Jeschke, 2015). Each stage comprises a rotating centrifugal impeller and stationary stator with a vaneless or vaned diffuser, bend, and return channel with a vane (Aalburg et al., 2011). Owing to high load and a small radial gap between the impeller and diffuser, unsteady impeller–diffuser interactions can be significantly large (Liu, Zhang, & Liu, 2014). Previous studies described the impeller–diffuser interaction. For example, Zamiri, Lee, and Chung (2016, 2017) described this interaction in detail, demonstrating that it causes unsteadiness at the interface region and a pulsating behavior within diffuser passages. Sato and He (2000) concluded that efficiency deteriorated if the radial gap between blade rows decreased owing to intensified blade row interaction. It was observed that this interaction could lead to resonance, which could have a destructive effect on efficiency (Vogel, Abhari, & Zemp, 2015). As the flow rate decreases, there is an increase in the impeller–diffuser interaction (Zhao, Wang, Zhao, & Xi, 2018).

Flow channel design between the impeller and diffuser must consider two aspects. The first is radial clearance—if it is too long, the gas flow path is prolonged, thereby increasing friction loss; if it is too short, the rotor and stationary blades interfere with each other such that the wake of the impeller blade affects the uniformity of the inlet air flow at the stationary blade (Ji, Li, Fang, & Sun, 2018). Hosseini, Sun, He, and Zheng (2017) achieved a radial clearance ratio that yielded the best pressure recovery coefficient, loss coefficient, and grade efficiency when $\mu_0 \geq 1$. Optimal radial clearance exists for specific working conditions (Krishnababu, Imregun, Green, & Hoyniak, 2010; Anish & Sitaram, 2009). Robinson, Casey, Hutchinson, and Steed (2012) reported that decreasing the radial clearance ratio substantially increased efficiency. However, when the diffuser was closer to the impeller than the optimum value, the impeller likely experienced mechanical excitation.

The second design aspect is partition structure (PS) between the impeller and stationary blade. When analysing rotor–stator interactions, most studies (Liu, Zhang, & Liu, 2015; Sun, Wang, & Zhang, 2018) considered only the flow zone and did not account for the cavity.
and seal. This was because several grids were required for calculation of the cavity and seal, which increased calculation time. For optimization designs using numerical methods, the influence of the cavity and seal on compressor performance is typically ignored. During actual operation, a gap exists between the partition and wheel cover and the wheel disk of the impeller, which can easily lead to airflow leaks. Leaking flow and cavity are important when predicting stage efficiency (Younsi, Corneloup, Moiroud, & Baldacci, 2017). These subtle flow phenomena interact with the main flow and can threaten the efficiency and stable operation of centrifugal compressors (Qin, Sun, Wang, Ju, & Zhang, 2016). In earlier studies, the rotor and stationary components were simplified to a ‘rotating stationary disk’ problem to examine the flow structure in detail (Remy, Gauthier, & Buisine, 2005; Remy & Buisine, 2006; Jiao & Fu, 2016). However, in recent years, studies have focused on investigating the actual structure. Previous studies investigated the influence of the centrifugal compressor cavity on its aerodynamic performance (Wang & Xi, 2011; Wang, Xu, & Xi, 2010). The results of these studies indicate that the influence of leakage is significant in the centrifugal impeller at a flow coefficient below 0.02. Further, the results demonstrated that models with a seal had lower efficiencies and pressure ratios than models without a seal, especially at low flow rates. Lower flow coefficients result in cavity flow having a greater influence (Wang, Ju, & Zhang, 2017; Guidotti et al., 2014). Further, mixing between leak and through flow can lead to flow separation at both the inlet and outlet of the centrifugal impeller (Wang et al., 2017). Micio, Facchini, Innocenti, and Simonetti (2011) indicated that clearance has a strong influence on leak loss, heat transfer, and flow field development. The leak flow in the seal on the wheel-cover side disturbed the boundary layer on the cover side of the impeller inlet, which reduced the stable operating range of the compressor (Mischo, Ribi, Seebass-Linggi, & Mauri, 2009). As is evident from the studies cited above, investigations thus far focused on the internal flow of the cavity and seal structure (Liu, Kong, Liu, & Feng, 2017; Kong, Liu, Liu, Lei, & Zheng, 2018). Thus, research on the flow field characteristics during coupling between the impeller outlet PS and main flow, as well as their influence on centrifugal compressor performance is lacking. The PS is a high-loss area in the entire gas channel despite being narrow; therefore, the PS at the impeller outlet significantly influences the gas flow, impeller outlet, and inlet flow field of downstream components, implying that its optimization could potentially increase stage efficiency.

The use of numerical techniques such as computational fluid dynamics (CFD) is prevalent in several engineering applications (Mou, He, Zhao, & Chau, 2017; Akbarian et al., 2018; Chitrakar, Dahlhaug, & Neopane, 2018; Ramezanizadeh, Nazari, Ahmadi, & Chau, 2019; Ghalandari, Koohshahi, Mohamadian, Shannshirband, & Chau, 2019). In this study, based on CFD, we established coupling calculation models of the main flow, partition clearance on the wheel cover and disk side, and seal structure. We first numerically analysed the influence of the rotor–stator PS on the aerodynamic performance of a centrifugal compressor and its loss mechanism. Further, the effect of PS on the area between the outlet section of the impeller and the inlet section of the stationary blade was analysed in detail. Consequently, the PS, its interaction mechanism, and the main flow in the compressor were revealed. Our results provide theoretical guidance for the design of partition cavity structures.

**Research object and methods**

**Geometric model**

To study the influence that the PS, between the impeller and stationary components, has on the gas flow, a solid model comprising the flow-through components, wheel cover, cavity on the wheel disk side, and seal structure was established in this study. The flow field was analysed using fluid dynamics software. The modeling method is consistent with the actual operation state of a centrifugal compressor and provides important guidance for theoretical research and engineering applications.

Figure 1 presents the geometric parameters of the model. The flow zone comprises the inlet passage, impeller, integrated stationary blade, and outlet passage. The wheel cover seal is located on the outside of the impeller wheel cover. The entire smooth labyrinth seal limits gas leaks from the high-pressure impeller outlet to the low-pressure impeller inlet. Labyrinth seals are widely used in industrial multistage centrifugal compressors to reduce internal leaks and prolong compressor running time (Marechale, Ji, & Cave, 2015). The disk seal is located at the connection between the cavity on the disk side and integrated stationary blade outlet, which limits the return gas flow from the stationary blade outlet to the impeller outlet. The wheel disk and cover sealing structures are identical, with six teeth. In this study, we investigated the intermediate stage of a vertical split type multistage centrifugal compressor with a closed impeller and 13 rear curved blades, followed by ten integrated stationary blades (Ji et al., 2018). Table 1 lists the geometric parameters and operating conditions.

Figure 2 shows that the calculation domain of Model A only includes the flow-through components of the compressor, i.e. the middle stage, impeller, and integrated blade, while neglecting the partition cavity and sealing
Figure 1. Geometric model parameters.

Table 1. Model geometric parameters and operating conditions (Ji et al., 2018).

| Parameter | Value |
|-----------|-------|
| Number of impeller blades, $Z_1$ | 13 |
| Speed, N (rpm) | 8,129 |
| Average impeller inlet diameter, $D_1$ (mm) | 326.54 |
| Impeller outlet diameter, $D_2$/outlet width, $b_2$ (mm) | 600/39 |
| Number of integrated stationary blades, $Z$ | 10 |
| Inlet diameter of integrated stationary blades, $D_{in}$/inlet width, $b_{in}$ (mm) | 750/31.2 |
| Inner arc radius of Bend, $R_1$ /Outer arc radius, $R_2$ (mm) | 18/49.12 |
| Outlet diameter of integrated stationary blades, $D_{out}$ /Outlet width $b_{out}$ (mm) | 466/56.53 |
| Medium | Air |
| Design point mass flow, $Q$, (kg/s) | 7.6 |

structure. The calculation domain of Model B comprises the flow-through components, as well as the two-sided partition cavity and sealing structures.

High-pressure gas from the impeller outlet can leak into the impeller inlet along the wheel cover and partition cavity, such that leaking gas is compressed twice. Leak control directly affects the efficiency and energy loss of the entire machine. Therefore, reducing the amount of leaking has significant economic advantages. The main factors that influence gas leaking are the pressure difference between the main flow and partition cavity, PS, clearance area, and rotational speed. To reduce gas leaks and consider the collision between the impeller and partition, we reduced the radial clearance between the wheel cover and wheel disk side partition. The PS was then modified to slow the vortex separation during gas turning. Figure 3 shows the changes in the PS, from top to bottom, for the circular arc, trapezoidal, and rectangular partition structures. In this study, based on Model B, the structure of the impeller outlet partition was altered to establish the stage solid model. In Figure 4, Models C, T, and R show the
circular-arc, trapezoidal, and rectangular PSs at the outlet of the impeller, respectively. The radial clearance, \( d \), is represented by numbers, such as \( T_6 \) for the trapezoidal PS, with a radial clearance of 6 mm.

**Methodology**

**The Mesh.** AutoGrid 5, which is a multi-block structured mesh generator, was used for geometric dispersion. For the flow channel in Model A, a structured mesh was generated using the O4H mesh topology. The mesh was encrypted near the wall and we set the first layer of the wall to a thickness of 0.01 mm. Figure 5 shows the verification of a single-channel grid independence. These parameters did not change significantly when there were more than two million mesh elements (Ji et al., 2018). In all directions, the mesh met the calculation requirements for multiple grids, i.e. the orthogonality, aspect ratio, and extension ratio (minimum orthogonal angle > 15°, maximum aspect ratio < 2,000, and maximum extension ratio < 5). To reduce the error induced by geometric dispersion, the single-channel mesh number of Model A was set to three million with an identical mesh topology for all examined cases in this study.

In Models B, C, T, and R, the rotating and static regions in the cavity channel were specified and divided into blocks, where the structured grid was used to discretise them. At the impeller inlet and stationary blade outlet, the grids of the cavity and main flow had a matched connection. At the impeller outlet, the partition cavity and main flow grids had a perfectly unmatched connection. When the grid independence of the cavity channel was verified, the grid number of the mainstream channel remained unchanged, after which the number of grids in the cavity channel was increased by one million. When the number of grids in the cavity channel exceeded four million, the efficiency and pressure ratio of the model did not change significantly, such that the grid was considered independent. Therefore, the number of single-channel grids for Models B, C, T, and R ranged from seven to ten million.

**Numerical method.** FINE/Turbo, which is a flow analysis software, was used to solve the turbulent Navier-Stokes equation in the numerical model. The Spalart–Allmaras turbulence model was adopted in these calculations, which performs well for near wake and complex flows (Spalart & Allmaras, 1994). We compared the computation results, using different turbulence models for a NACA model 404-III centrifugal compressor, to experimental data available in the literature (McKain & Holbrook, 1997). Figure 6 shows the results calculated using the Spalart–Allmaras model, which provided the closest match to the results of practical experiments (Ji et al., 2018).

Figure 7 shows the distribution of \( y^+ \) values on the solid wall. Values of \( y^+ < 10 \) are acceptable for the Spalart–Allmaras model to obtain valid conclusions (Ji et al., 2018). Spatial discretisation was performed using the space-centre differential format. The flow medium was air, gas was considered as viscous and compressible, and the Courant-Friedrich-Levy number was set to three.

Figure 8 shows that the rotor–stator interface is a circumferential, conserved connection surface, with a mixing plane treatment method.

**Boundary conditions.** For single-channel calculations, the inlet boundary conditions are as follows: The given flow direction was the axial direction and the total pressure and temperature were 101,325 Pa and 288 K, respectively. The outlet boundary condition had an imposed mass flow of 7.6 kg/s. An adiabatic wall and no-slip boundary conditions were applied to all wall boundaries. The component that belonged to the impeller rotated...
with the impeller, with a constant rotational speed of 8,129 r/m, while the rest of the wall was static.

Figure 8 shows, for the impeller disk cavity, that two interfaces were present in the main channel, i.e. one in the moving blade domain and one in the stationary domain. Therefore, by taking the R–S Interface 2 as the boundary, the upper half was located in the moving blade domain, which belonged to the rotating domain and the lower part was located in the stationary blade domain, which belonged to the stationary domain. For the cavity and seal structure on the impeller cover side, both interfaces were located in the moving blade domain, such that the structure was entirely in the rotating domain. The rotating domain had the same speed as the impeller. Both the cover and disk seal were included in the CFD calculations. The method used to handle the seal at the cover was identical to that of the disk. Two profile lines (i.e. the ZR coordinates) were used to form the cavity and seal structure. The cavity and seal structure behind the impeller were modeled and meshed together with the mainstream channel, and the numerical method and boundary conditions mentioned above were used to solve the problem.

Convergence criteria. When the global residuals and the residuals in each block decreased by more than three orders of magnitude, the relative errors in the inlet and outlet mass flows were less than 0.5%. The flow rate did not change again, such that the calculation was convergent (Ji et al., 2018).

Effect of the PS on the centrifugal compressor performance

Figure 9 presents the aerodynamic performance curves of the centrifugal compressor obtained by solving the model flow field for the design and variable working conditions. The model efficiency and total pressure ratio decreased to a certain extent after considering the partition and sealing structure. At a design mass flow of 7.6 kg/s, the efficiency of Model B was 2.54% lower than that of Model A while the total pressure ratio decreased by 1.55%. During near-surge flow (6.84 kg/s), the efficiency and total pressure ratio decreased by 2.82% and 1.6%, respectively.
The efficiency and total pressure ratio decreased by 1.56\% and 0.94\%, respectively, during near-choked mass flow (9.12 kg/s). The effect that the compressor partition cavity and seal structure had on the stage performance was more significant under small mass flow conditions, which was mainly due to the larger ratio of internal leak loss. The stable operating ranges of Models A and B were identical and unaffected by the partition and sealing structure. By comparing the models with and without the partition and sealing structures, we observed that the influence of microflow components must be considered for the aerodynamic performance analysis and design of a centrifugal compressor.

Comparing the performances of Model B with Models C, T, and R showed that the performance of the partition improved to a certain extent compared to that of the original structure (except for Models C_4 and C_6). The performance at the design point was higher than when under varying working conditions. The stable operating ranges of all models were identical. The efficiency of Model C_2 at each flow rate was better than that of the other calculation models, especially at the design point. Model C_2 achieved the highest stage efficiency of 81.27\%, which was 0.99\% higher than that of Model B, with an increase in the total pressure ratio of 1.23\%.

Figures 10 and 11 show the relationships among the model efficiency, $d$, and $PS$ form at the design point. We observe that, for the same clearance, each model exhibits different efficiencies. When the clearance is small, there is no evident difference among the three structures. With an increase in the clearance, the differences among the three structures are more evident and the grade efficiency decreases to different degrees. Simultaneously, the stage efficiency for a small clearance is generally better than that for a large clearance. There were significant variations in the efficiency of Model C at different clearances.
At various clearances, there was little change in the efficiency of Models T and R. Combined with Figures 10 and 11, this indicates that the radial clearance, \(d\), and PS form were the factors that affected the overall efficiency. The sensitivities that different structural forms have to the clearance were different. Circular-arc structures were the most sensitive, whereas rectangular structures were the least sensitive.

**Influence of the PS**

**PS form**

Based on an analysis of Figure 10, we observe that Model C had the highest efficiency at \(d = 2\) mm while Model R had the highest stage efficiency at \(d = 4\) and \(6\) mm. When the clearance was small, there was no difference among the efficiencies of the three structures. With increases in the clearance, there were clear differences among the three structures. Therefore, we analysed the influence that the different PS forms have on the compressor performance and flow field at \(d = 4\) mm.

Figure 12 presents the velocity distribution along the meridional plane for Models C\(_4\), T\(_4\), and R\(_4\). The diagram indicates that, along the meridional plane, the gas reflux and swirl phenomena exist in the axial clearance of the impeller blade outlet. The number and position of the vortices were determined by the geometric shape of the partition cavity. Vortex motion was due to gas flow at a high speed from the impeller into the left partition cavity, which was influenced by the centrifugal force and static pressure gradient. High-speed gas induces low-speed gas in the cavity into a counterclockwise movement and the formation of a whirlpool. The low-speed gas in the right partition cavity tended to enter the main flow region but was impacted by high-speed air in the main flow area, which caused the whirlpool to move clockwise (see Figure 13). The mass and energy transport of the gas

**Figure 12.** The velocity distribution on the meridional plane of Models C\(_4\), T\(_4\), and R\(_4\). The red streamline in the cavity channel represents the whirlpool in the air flow. (a) Model C\(_4\), (b) Model T\(_4\), and (c) Model R\(_4\).
in these vortices was in constant circulation, such that the leak flow was always coupled with this cyclic process.

In thermodynamics, entropy is used to measure the disorder in a physical system. If the entropy is high, systems are highly disordered, with a high loss. Figures 14 and 15 compare the entropy distributions of the PS at 95% and 5% of the impeller-leaf-height cross-sections at the design point, respectively, while preserving the transition region between the impeller and stationary blade. The entropy distributions of the PS at 95% and 5% indicate the influence that the PS has on the impeller and its rotor–stator transition zone, respectively.

As shown in Figures 14 and 15, due to the existence of the partition and seal structure, there were entropy-increasing regions at the top and root of the blade, i.e. there was a high-loss area. The difference between the top and root of the blade was that the high-entropy region of the leaf top was concentrated at one-third of the distance of the blade inlet to the blade outlet, where the high-entropy region at the leaf root always continues to the rotor–stator transition region. In other words, the PSs on both sides of the impeller had different impacts on the compressor flow zone. The PS on the wheel cover side mainly affected the gas flow in the impeller, whereas the PS on the wheel disk side had a significant influence on the rotor–stator transition section, i.e. it affects the gas flow field in the downstream sections. Combining the data in Figure 3 to analyse the cause, we observed that high-speed gas flowed from the impeller outlet into the cavity through the wheel cover partition clearance. The gas flowed from the impeller inlet and mixed with the main flow. This was a cyclic process and directly influenced the flow field at the top of the impeller blade. The flow route of the leaking gas in the disk side partition was initially from the stationary blade outlet to the impeller blade outlet, which then joined the main flow via the high-speed gas from the impeller. Therefore, the PS on the disk side directly affected the flow field in the downstream components. Figure 3 shows the flow routes of two leaking gases.

In Figure 14, a high-entropy region occurs at the impeller outlet of Model C4, where the corresponding flow loss increased by approximately 110 J/(Kg·K). The figure shows that the mechanical energy of the gas was converted into thermal energy to a significant extent, which indicates the existence of friction loss. The entropy values of the other two models in the wake region of the impeller were 50–60 J/(Kg·K). According to the above analysis, Model C4 had a larger loss area than the other two models in the rotor–stator transition region, which is shown in Figure 15. The area denoted by the red circle in Figure 15(a) is the high-entropy area of the rotor–stator transition zone, i.e. the high-loss area. Based on the analysis of the geometric structures of the three models, only the leaking gas in Model R can directly enter the partition cavity while the other two models possess a local bending point before the cavity (red circles in Figure 12). Here, the gas was chaotic, resulting in local loss, which was also the main reason why Model R had the highest stage efficiency at $d = 4$ mm. We demonstrated that loss in the local flow loss was the main factor that affected model efficiency when radial clearance was identical. However, the efficiency of Model R was not the highest at $d = 2$ mm, which also indicated that radial clearance, $d$, is important for compressor performance, as well as the PS.

**Radial clearance ($d$)**

Figure 11 shows that Model C2 exhibits the highest efficiency. The efficiency of the circular-arc structure is the most sensitive to $d$ variations; the rectangular structure is the least sensitive. Therefore, taking Model C as the analysis object, the influence of $d$ on the performance and flow field of the centrifugal compressor was analysed.
Figures 16 and 17 show the velocity and entropy distribution, respectively, at the inlets of the wheel cover partitions in Models C2, C4, and C6.

Figure 16 shows that the mass flow of leaking gas, as well as its velocity and kinetic energy, increase with $d$ while there is a gradual stabilization of the vortex system. When $d = 2$ mm, the channel at the inlet of the partition on the wheel cover side was narrow and long. Furthermore, the centrifugal force and a static pressure gradient affected the leaking gas, which caused cyclotron motion. However, due to less leaking gas and an insufficient kinetic energy, the vortex that formed was smaller and affected by the flow channel. Leaking gas moved along the gap of the partition, which damaged the unstable vortex. When $d = 6$ mm, the leaking gas had a sufficient kinetic energy and space in the channel, which allowed Model C6 to form a large stable vortex in the middle of the channel. Combined with Figure 17, the entropy of Model C6 was significantly higher than that of Models C2 and C4, which is related to the formation of stable vortices in the model. Furthermore, the incurred leak loss was substantial. Model analysis at various clearances using the circular-arc structure indicates that efficiency differences are not evident for small clearances due to the kinetic energy of the leak flow and flow channel shape.

At the gap between the wheel cover and partition on the wheel disk side, the main factor that affected the amount of leaking gas was the pressure difference between the main flow and partition channel, PS, and clearance area. The clearance area was proportional to $d$. Thus, $d$ had a direct effect on the leak amount and loss. In the design of a closed impeller, for convenient installation and maintenance, the radial clearance, $d$, at the partition is generally larger than that of a semi-open impeller, i.e. designers must pay attention to this aspect.

The study has demonstrated that the PS and radial clearance, $d$, cause an efficiency loss via different mechanisms in the model. The former affects local gas flow loss, whereas the latter has a direct effect on gas leak loss.
Figure 15. The entropy distribution on the 5% leaf-height cross section: (a) Model $C_4$, (b) Model $T_4$, and (c) Model $R_4$.

Figure 16. The velocity distribution on the meridional plane of Model $C$: (a) Model $C_2$, (b) Model $C_4$, and (c) Model $C_6$.

Figure 17. The entropy distribution on the meridional plane of Model $C$: (a) Model $C_2$, (b) Model $C_4$, and (c) Model $C_6$. 
Structural improvements

Investigations into the PS and radial clearance indicate that both cannot be ignored when analysing compressor performance. Further, we must consider the microflow components to design compressors. According to the performance curves of the various calculation models shown in Figure 9, the efficiency and pressure ratio of Model C2 at the design point were higher than those of the original model by 0.99% and 1.23%, respectively. The efficiency increased by 1.49% under near-surge conditions and 2.35% under near-choked conditions. Therefore, Model C2, which had the highest efficiency at the design point, was selected as a reference for Model B to analyse the influence that PS variations have on the original flow field.

Figure 18 shows the velocity distribution in the meridional plane of the two models. Furthermore, Figure 18(b) shows that the high-speed gas exiting the impeller enters the cavity through the inlet of the cover-side partition. Due to differences in the centrifugal force on the wall, a velocity difference occurs, i.e. the swirling motion of the gas in the square box region shown in Figure 18(b), which formed a vortex. The shape of the channel damages this cyclotron motion and is driven by high-speed airflow, passing downstream along the channel. Near the right corner of the wheel cover (shown by the arrow in Figure 18(b)), the gas is deflected, which eliminates the entire vortex and its effects.

Similarly, there was a whirlpool at the entrance of the cover-side partition in Model B. However, the effect of the vortex was much larger than that of Model C2, where the reverse flow continued to maintain one-third the length of the entire cavity channel. The resulting flow loss was significantly higher than that of Model C2.

Figures 19 and 20 show the static pressure distributions at 98% and 2% of the leaf-height cross-sections in the two models. The diagram shows that the static pressure distribution for the two PS types was primarily...
identical. However, the static pressure of Model C₂ increased at the moving blade outlet and stationary blade inlet, as well as an increase in the red high-pressure region.

To further analyse the influence that the PS has on the transition region, we investigated the static pressure distributions at the impeller outlet section, intermediate section, and inlet section of the stationary blade, as shown in Figure 21.

The figure shows that the static pressure of Model C₂ was significantly higher than that of Model B in all three sections, particularly the impeller outlet and intermediate sections. The figures also show that there was an enlargement of the range in the high-static-pressure region. Improving the PS design has a certain effect on the rotor–stator flow field and compressor performance. Simultaneously, the influence that the PS has on the main flow zone gradually weakened with gas flow.
Furthermore, in the two models, the static pressure distribution of the gas at the outlet of the impeller and the pressure distribution between the middle and inlet sections of the stationary blade tended to be uniform. The static pressure isoline was smooth, which indicated that, with air flow, there was a gradual weakening of the PS’s ability to affect the inhomogeneity of the flow field in the flow zone.

Conclusions

In this study, we established a coupling calculation model for the flow-through components, wheel cover, partition on the wheel disk side, and seal structure in a centrifugal compressor. This allowed us to numerically evaluate the influence that the rotor–stator PS has on the aerodynamic performance of the centrifugal compressor and its loss mechanism. This study can provide a theoretical foundation for the in-depth understanding of the microflow components in centrifugal compressors. The main results can be summarized as follows:

- By establishing the geometric model (with and without the wheel cover, partition on the wheel disk side, and sealing structure) and solving the flow field, the efficiency and total pressure ratio of the model stage decreased to a certain extent after considering the partition and sealing structure. For a small flow rate, the effect that the partition cavity and seal structure had on stage performance was more evident. This result also indicates that it is necessary to consider the influence of microflow components in an aerodynamic performance analysis and the design of centrifugal compressors.
- The PS form and radial clearance, \( d \), are the factors that affect the overall efficiency. The sensitivity that different PS structure forms have to the clearance vary. The circular-arc-shaped PS is the most sensitive while the rectangular PS is the least sensitive. When the clearance is small, there is no evident difference in the stage efficiency among the three PS types. When the clearance increases, the difference among the three structures is more evident, such that the grade efficiency of each model decreases at varying degrees. Both have different effects on losses in the model efficiency. The PS affects the local gas flow loss while radial clearance, \( d \), has a direct effect on losses in gas leaks, to which designers must pay attention.
- The PS on both sides of the impeller has a different effect on the flow zone of the compressor. The PS on the wheel cover side effects the gas flow in the impeller while that on the wheel disk side has a significant influence on the rotor–stator transition region, i.e. it affects the gas flow field in downstream regions.
- The performance evaluation of different models showed that Model C2 exhibits the highest stage efficiency and pressure ratio. Compared with the original model, the entire stage efficiency and total pressure ratio increased by 0.99% and 1.23%, respectively, by redesigning the PS without modifying the impeller and stationary components (i.e. the diffuser, bend, and return channel). Comparing the flow field parameters of two models indicated that the range of influence for the gas reflux and vortex in the cavity decreased after modifying the PS. Static pressure exhibits an evident improvement, particularly at the impeller outlet and intermediate sections while the high-static-pressure area expanded. The improved PS design has a certain effect on the rotor–stator flow field and compressor performance. However, the influence that the PS has on the flow zone gradually weakened with the gas flow.

In this study, the characteristics of various structures were analysed through steady calculations. In future studies, we intend to use the unsteady method to investigate the formation and development of vortices in an impeller disk and the wheel cover cavity.

Disclosure statement

No potential conflict of interest was reported by the authors.

ORCID

Chunyang Li © http://orcid.org/0000-0001-9488-8183

References

Aalborg, C., Sezal, I., Haighmoser, C., Simpson, A., Michelassi, V., & Sassanelli, G. (2011, June). Annular cascade for radial compressor development. In ASME turbo expo 2011: Turbine technical conference and exposition (pp. 2231–2242). Vancouver, Canada: American Society of Mechanical Engineers Digital Collection.

Akkarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K.-W. (2018). Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled diesel engine. Engineering Applications of Computational Fluid Mechanics, 12(1), 517–534.

Anish, S., & Sitaram, N. (2009). Computational investigation of impeller-diffuser interaction in a centrifugal compressor with different types of diffusers. Proceedings of the Institution of Mechanical Engineers Part A-Journal of Power and Energy, 223, 167–178.

Chitrakar, S., Dahlhaug, O. G., & Neopane, H. P. (2018). Numerical investigation of the effect of leakage flow through erosion-induced clearance gaps of guide vanes on the performance of Francis turbines. Engineering Applications of Computational Fluid Mechanics, 12(1), 662–678.
Ghalandari, M., Koohshahi, E. M., Mohamadian, F., Shanshirband, S., & Chau, K. W. (2019). Numerical simulation of nanofluid flow inside a root canal. Engineering Applications of Computational Fluid Mechanics, 13(1), 254–264.

Guidotti, E., Toni, L., Rubino, D. T., Tapinassi, L., Naldi, G., Koyyalamudi, V. N. K. S., & Prasad, S. (2014, June). Influence of cavity flows modeling on centrifugal compressor stages performance prediction across different flow coefficient impellers. In ASME turbo expo 2014: Turbine technical conference and exposition. Dusseldorf, Germany: American Society of Mechanical Engineers Digital Collection.

Hosseini, M., Sun, Z. H., He, X., & Zheng, X. Q. (2017). Effects of radial gap ratio between impeller and vaned diffuser on performance of centrifugal compressors. Applied Sciences-Based, 7(7), 728. doi:10.3390/app7070728.

Ji, C., Li, C., Fang, J., & Sun, Q. (2018). Loss mechanism of static interstage components of multistage centrifugal compressors for integrated blade design. Mathematical Problems in Engineering, 2018. doi:10.1155/2018/9025650.

Jiao, Z., & Fu, S. (2016). Numerical investigation of flows between rotating disks. Chinese Journal of Computational Mechanics, 33, 588–593.

Kong, X. Z., Liu, G. W., Liu, Y. X., Lei, Z., & Zheng, L. X. (2018). Performance evaluation of the inter-stage labyrinth seal for different tooth positions in an axial compressor. Proceedings of the Institution of Mechanical Engineers Part A-Journal of Power and Energy, 232(6), 579–592.

Krishnababu, S. K., Imregun, M., Green, J. S., & Hoyniak, D. (2010, October). Aerodynamics and aerelasticity of impeller vane interactions in a high pressure ratio centrifugal compressor. In ASME turbo expo 2010: Power for Land, Sea, and Air (pp. 1201–1208). Glasgow, Scotland: American Society of Mechanical Engineers Digital Collection.

Liu, G., Kong, X., Liu, Y., & Feng, Q. (2017). Effects of rotational speed on the leakage behavior, temperature increase, and swirl development of labyrinth seal in a compressor stator well. Journal of Aerospace Engineering, 231, 2362–2374.

Liu, B., Zhang, B., & Liu, Y. (2014). Numerical investigations of impeller-diffuser interactions in a transonic centrifugal compressor stage using nonlinear harmonic method. Journal of Power and Energy, 228, 862–877.

Liu, B., Zhang, B., & Liu, Y. (2015). Investigation of model development for deterministic correlations associated with impeller-diffuser interactions in centrifugal compressors. Science China Technological Sciences, 58, 499–509.

Marechale, R., Ji, M., & Cave, M. (2015, June). Experimental and numerical investigation of labyrinth seal clearance impact on centrifugal impeller performance. In ASME turbo expo 2015: Turbine technical conference and exposition. Montreal, Canada: American Society of Mechanical Engineers Digital Collection.

McKain, T. F., & Holbrook, G. J. (1997, July). Coordinates for a high performance 4:1 pressure ratio centrifugal compressor. Indianapolis, IN: NASA.

Micio, M., Facchini, B., Innocenti, L., & Simonetti, F. (2011, January). Experimental investigation on leakage loss and heat transfer in a straight through labyrinth seal. In ASME 2011 turbo expo: Turbine technical conference and exposition (pp. 967–979). Vancouver: American Society of Mechanical Engineers Digital Collection.

Mischo, B., Ribi, B., Seebass-Linggi, C., & Mauri, S. (2009, January). Influence of labyrinth seal leakage on centrifugal compressor performance. In ASME turbo expo 2009: Power for land, sea, and air (pp. 1283–1293). Orlando, FL: American Society of Mechanical Engineers Digital Collection.

Mou, B., He, B.-H., Zhao, D.-X., & Chau, J.-W. (2017). Numerical simulation of the effects of building dimensional variation on wind pressure distribution. Engineering Applications of Computational Fluid Mechanics, 11(1), 293–309.

Qi, R., Sun, Y., Wang, S., Ju, Y., & Zhang, C. (2016). Flow structure in impeller backside cavity and its effect on aerodynamic performance of a centrifugal compressor. Journal of Engineering Thermophysics, 37, 2349–2354.

Ramezanizadeh, M., Nazari, M. A., Ahmadi, M. H., & Chau, K.-W. (2019). Experimental and numerical analysis of a nanofluidic thermosyphon heat exchanger. Engineering Applications of Computational Fluid Mechanics, 13(1), 40–47.

Remy, D., & Buisine, D. (2006). Experimental and numerical study of a spiral structure at the periphery of an aspirated rotor–stator cavity. Experiments in Fluids, 41, 393–399.

Remy, D., Gauthier, G., & Buisine, D. (2005). Instabilities between rotating and stationary parallel disks with suction. Physics of Fluids, 17, 018102–018102.

Robinson, C., Casey, M., Hutchinson, B., & Steed, R. (2012, June). Impeller–diffuser interaction in centrifugal compressors. In Proceedings of the ASME turbo expo 2012: Turbine technical conference and exposition (pp. 767–777). Copenhagen, Denmark: American Society of Mechanical Engineers Digital Collection.

Rube, C., Rossbach, T., Wedeking, M., Grates, D. R., & Jeschke, P. (2015, June). Experimental and numerical investigation of the flow inside the return channel of a centrifugal process compressor. ASME turbo expo: Turbine technical conference and exposition, Montreal, Canada.

Sato, K., & He, L. (2000, May). A numerical study on performances of centrifugal compressor stages with different radial gaps. In Proceedings of the ASME turbo expo 2000: Power for land, sea, and air (p. V001T03A036). Munich, Germany: American Society of Mechanical Engineers Digital Collection.

Splaart, P. R., & Allmaras, S. R. (1994). A one-equation turbulence model for aerodynamic flows. Recherche Aerospatiale, 1, 5–21.

Sun, P., Wang, Y., & Zhang, X. (2018, July). Effects of airflow deflection angle in diffuser on forced response caused by impeller-diffuser interaction in centrifugal compressors. MATEC web of conferences (Vol. 179, p. 01009).

Vogel, K., Abbhari, R. S., & Zemp, A. (2015). Experimental and numerical investigation of the unsteady flow field in a vaned diffuser of a high-speed centrifugal compressor. Journal of Turbomachinery-Transactions of the ASME, 137, 1259–1268.

Wang, S., Ju, Y., & Zhang, C. (2017). Coupling analysis between through flow and hub-shroud side leakage flow of a centrifugal compressor stage. Journal of Xi’an Jiaotong University, 51, 73–77.

Wang, Z., & Xi, G. (2011). Influences of cavity leakage on the design of low flow coefficient centrifugal impeller. Science China-Technological Sciences, 54, 311–317.

Wang, Z., Xu, L., & Xi, G. (2010, October). Numerical investigation on the labyrinth seal design for a low flow coefficient centrifugal compressor. In ASME turbo expo 2010: Power for land, sea, and air (pp. 2031–2041). Glasgow, Scotland: American Society of Mechanical Engineers Digital Collection.
Younsi, M., Corneloup, C., Moïroud, F., & Baldacci, A. (2017, June). Unsteady flow in a centrifugal compressor stage equipped with a vaned diffuser and cavities. In ASME turbo expo: Turbine technical conference and exposition. Charlotte, NC: American Society of Mechanical Engineers Digital Collection.

Zamiri, A., Lee, B. J., & Chung, J. T. (2016, June). Numerical evaluation of the unsteady flow in a centrifugal compressor with vaned diffuser via URANS approach. ASME turbo expo 2016: Turbomachinery technical conference and exposition, Seoul, South Korea. American Society of Mechanical Engineers Digital Collection.

Zamiri, A., Lee, B. J., & Chung, J. T. (2017). Numerical evaluation of transient flow characteristics in a transonic centrifugal compressor with vaned diffuser. Aerospace Science and Technology, 70, 244–256.

Zhao, J., Wang, Z., Zhao, Y., & Xi, G. (2018). Investigation of transient flow characteristics inside a centrifugal compressor for design and off-design conditions. Journal of Power and Energy, 232, 364–385.