

Article

Effects of End-Wall Suction on the Flow Field Structure and Aerodynamic Performance of a Compressor Cascade

Wenfeng Xu , Yingxuan Wang, Shilong Zou , Guozhe Ren * and Shengyuan Wang 

School of Aero-Engine, Shenyang Aerospace University, Shenyang 110136, China; xuwf789@163.com (W.X.); 15828042451@163.com (Y.W.); zoushilong251@163.com (S.Z.); wangsy1717@163.com (S.W.)

* Correspondence: sau_2022@163.com

Abstract

With the advancement of aeroengines toward higher pressure and efficiency, the advantages and disadvantages of the compressor working environment significantly affect the whole machine's performance. Boundary layer separation deteriorates the flow field structure. End-wall suction technology is an end-zone control technology that can enhance the aerodynamic performance and flow field structure. This study numerically simulated the influence of end-wall suction at different positions on the vortex evolution and aerodynamic performance under various working conditions. The results suggest that end-wall suction can effectively improve the flow field, reduce the corner separation region, and inhibit low-energy fluid cluster aggregation, thereby improving the cascade diffusion capacity and reducing flow loss. The suction effect gradually diminishes as the suction position moves closer to the trailing edge (TE). The total pressure loss is reduced by 16% when implementing suction at the leading edge (LE), and the loss reduction effect is the best. Furthermore, the suction effect shows a trend of increasing initially before diminishing as the incidence angle increases. When the incidence angle is 0° , the suction effect is the most potent.

Keywords: compressor cascade; flow field structure; end-wall suction; corner separation; aerodynamic performance

1. Introduction

The aviation industry is currently experiencing increases in efficiency and thrust ratio requirements for aeroengines, particularly for axial compressors. The compressor performance directly determines the system efficiency, thrust, and reliability. Today, research focuses on enhancing the aerodynamic performance and reducing the flow loss. In modern high-load designs, the compressor boundary layer is readily separable, and the end-zone flow control technology can significantly reduce the influence range of corner separation. Hence, it is vital to ascertain effective flow control technology in modifying the aerodynamic performance. Exploring flow control technology to enhance performance and stability is highly valuable.

As the crown jewel of industrial technology, aeroengines represent a key focus for national defense and scientific advancement. The compressor, a vital component, directly influences the engine efficiency and stability margin through its performance and reliability. As engines evolve toward higher thrust-to-weight ratios, compressors face increasingly demanding loads. High loads directly induce increased reverse pressure gradients in the



Academic Editor: Jichao Li

Received: 9 February 2026

Revised: 25 March 2026

Accepted: 26 March 2026

Published: 30 March 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

compressor corner regions, thereby exacerbating corner separation. Consequently, investigating flow control techniques to suppress corner separation in high-load compressors holds significant research value for enhancing aerospace engine performance and stability.

In recent years, end-zone flow control technology has gradually emerged as an effective means to enhance compressor performance. Instabilities in end-zone flow, particularly flow separation and reverse flow phenomena near the end walls, are particularly pronounced during compressor operation and often lead to significant degradation in aerodynamic performance. Currently, end-zone flow control primarily encompasses active and passive flow control techniques, where passive flow control techniques primarily encompass vortex generators [1,2], end-wall winglets [3], non-axially symmetric end walls [4], swept blade technology [5], tip vortices [6,7], blade ribs [8], and blade–end-wall integration techniques [9], while active flow control technology achieves flow field regulation by introducing energy forms such as airflow and force into the flow field via auxiliary equipment. These techniques effectively mitigate boundary layer separation by altering end-zone flow characteristics, thereby improving the compressor pressure distribution, flow stability, and overall efficiency. They include end-wall suction [10,11], pulse-jet suction [12], plasma excitation [13,14], pulsed jets [15], and jet technology [16].

The fundamental principle of end-wall suction involves arranging suction slots or passages on the end wall to actively draw in low-energy fluid. This alters the fluid velocity distribution and reduces vortex formation, thereby slowing boundary layer growth and preventing flow separation. Significant progress has been achieved in recent years, where end-wall suction can mitigate corner separation, suppress low-energy fluid agglomeration, and consequently reduce flow losses while enhancing aerodynamic performance. Cao et al. [17] found that arranging suction slots at 30–70% of the axial chord length (c) can eliminate the TE separation well and significantly improve the cascade diffusion capacity. Wang et al. [18] performed a full-span slit treatment on the large-arc compressor cascade and established a combined flow control scheme that incorporates two distinct boundary layer suction slots. They found that the full-span slotting configuration can eradicate the TE separation that is situated at the midpoint of the span, whilst concomitantly attenuating the separation. Cao et al. [19] discussed the coupling of the bending and suction flow mechanism in the compressor cascade. They found that coupling negative bending and end-wall suction can improve the performance and minimize the loss coefficient. Moussavi et al. [20] established that the implementation of end-wall suction can enhance the rotor power coefficient through the creation of a narrow slot in the medial part of the blade, in conjunction with the application of varying suction intensities. The improvement in aerodynamic performance achieved through end-wall suction is closely related to operational parameters such as the positional distribution of suction slots and incidence angles. To this end, researchers have conducted the following studies on the influence patterns of suction schemes: Liu et al. [21] proposed two new suction slit configurations for the boundary layer. They discovered that the extended suction slot on the middle end wall of the single slot performed better in the entire operating incidence angle range under the same suction flow. Methel et al. [22] experimentally investigated the critical relative dimensions of two surface defects found on the suction and aerodynamic surfaces. Zhao et al. [23] proposed critical parameters that elucidate the turbulent properties of convective heat transfer under uniform suction conditions. Zhang et al. [24] examined the repercussions of end-wall suction on flow control and aerodynamic performance, which have different loss ratios under large incident angles and design incident angles. They found that the substantial airflow suction capacity is responsible for separating the low-span area under considerable incident angles, and the non-gap end corner separation region expands upward. You et al. [25] defined several surface element functions, each of which exhibited a distinct effect on the secondary

flow. This generated a novel end-wall profile for resolving the corner separation problem. Fan et al. [26] discovered that when the suction position was at 30% c , the end-wall suction had significant effects at differing incidence angles. It is evident that while end-wall suction technology demonstrates significant advantages in optimizing compressor aerodynamic performance, a systematic review and analysis of the existing literature reveals that several critical issues require further in-depth research and systematic resolution. These include the influence patterns of end-wall suction on vortex evolution in the corner region and overall aerodynamic performance under different operating conditions, as well as the methods for optimizing and determining the end-wall suction location.

Therefore, this study investigated the separation mechanism in the suction control angle region near the end wall of high-load axial compressor blades through numerical simulation methods to identify the optimal scheme in terms of the aerodynamic performance and flow field structure under various suction position schemes and incidence angles. This research provides a theoretical basis and technical support for the development of new high-load compressors.

2. Research Object and Numerical Method

2.1. Research Object

A typical NACA65 K48 cascade was selected as the research subject to investigate the effect of end-wall suction on corner separation in this study. Previous studies have revealed significant boundary layer separation and lateral secondary flow in the corner region of this blade section. Figure 1 illustrates the structure and parameters of this cascade. In compressors, corner separation usually takes place at the end-wall intersection of the passages and the suction side (SS). The separation is attributable to the backward pressure gradient of the SS, and the fluid gathers in the corner area, progressively deteriorating the separation. An earlier conclusion [17] indicates that suction can improve the flow field. However, the present study did not examine the impact of varying suction schemes. Instead, this study designed slot schemes with three positions to explore the regulatory effect of end-wall suction convective field corner separation.

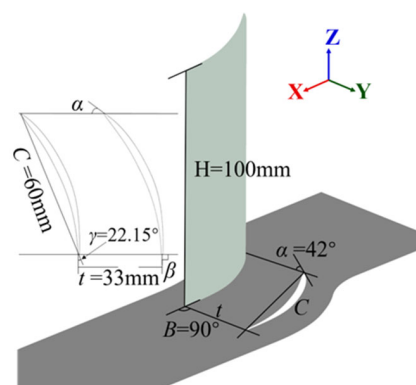


Figure 1. Cascade parameter diagram.

This study set the suction slots at different positions around the cascade: Case 1 was set at 10–20% of the chord on the LE, Case 2 was set at 50–60% c on the LE, and Case 3 was set at 90–100% c on the LE. Figure 2 shows the positions of the suction slots. In addition, to evaluate the flow distribution characteristics, the normalized flow rate was defined as follows:

$$Q_{norm} = \frac{Q_{out,jet}}{Q_{in}} \quad (1)$$

where $Q_{out,jet}$ is the flow rate at the outlet jet, and Q_{in} is the flow rate at the inlet of the passage. The corresponding normalized mass flow rates for Cases 1, 2, and 3 were 0.37%, 0.44%, and 0.48%, respectively.

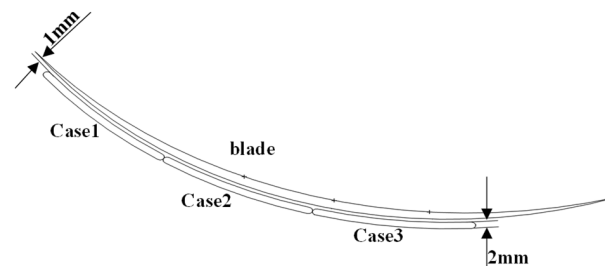


Figure 2. Diagram of the layout of the suction slots.

2.2. Computational Domain Meshing

This study employed ANSYS ICEM (version 2021 R1) for the mesh generation of the computational domain. Figure 3 shows the domain mesh division structure. The first layer of the mesh had a thickness of 0.002 mm, with a mesh growth ratio of 1.12. The y^+ contour plot of the SS, obtained through numerical calculation results, is shown in Figure 4. The value of y^+ was maintained near 1, which meets the requirements of the SST turbulence model for grid accuracy. This study performed mesh refinement around the end wall, deploying O meshes closely adjacent to the blade surface to ensure mesh quality and y^+ values. When performing analog calculations, to ensure that the mesh density matches the suction slot inlet, outlet, and end wall, this study set the suction slots and blade grid nodes according to a certain proportion to avoid a significant difference in the mesh density.

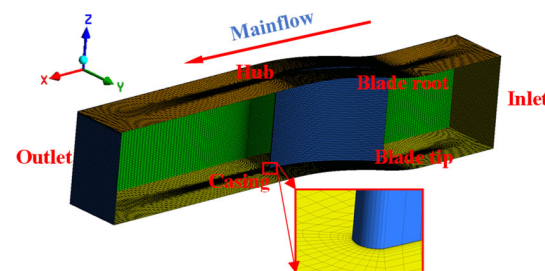


Figure 3. Fluid domain mesh division structure.

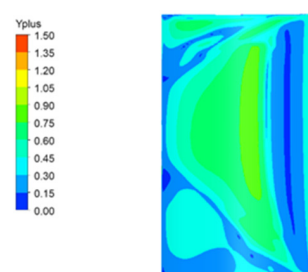


Figure 4. Blade suction surface y^+ contour diagram.

2.3. Boundary Condition Setting

This study numerically simulated the structure shown in Figure 5. Previous studies have found that the CFX derives the numerical solution. To ensure symmetrical computing

results and save resources, this study set the lower boundary to a symmetrical boundary. The intersection interface on both sides was set as a cyclical boundary condition and only calculated the 50% blade height (H) calculation domain. This study designated the inlet as the pressure inlet.

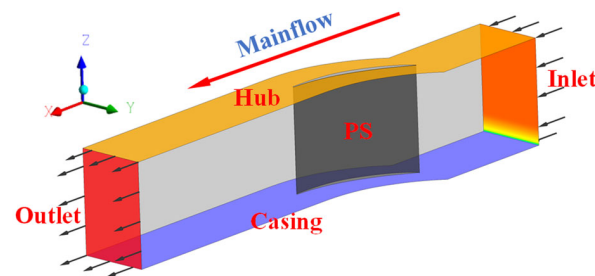


Figure 5. Computational domain boundary.

Figure 6 shows the inlet pressure distribution along the blade height, which was used for the boundary condition setup. The inlet airflow was designated as 0.7 Ma. The temperature was set to 320 K. The outlet was designated as the pressure outlet, with a pressure value of 101,325 Pa. The temperature was preset to 288 K. In this study, it was assumed for simplicity that the end walls were thermally insulated. Moreover, no slippage occurred. The suction outlet was set up as a flow outlet, and 2% of the specified inlet flow rate was set as the suction flow rate.

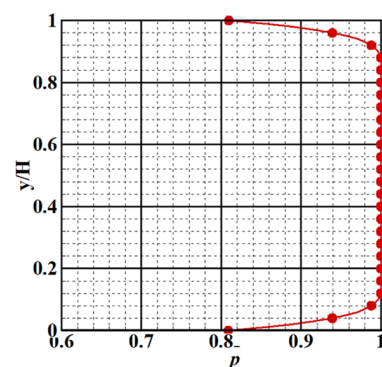


Figure 6. Distribution of the total inlet pressure along the spanwise direction.

2.4. Grid Independence Verification

To ascertain the calculation precision, a mesh independence verification was performed. The C_{pt} is of fundamental importance in the flow loss measurement. The outlet C_{pt} was the evaluation metric and defined as follows:

$$C_{pt} = \frac{p_{t,in} - p_{t,out}}{p_{t,in} - p_{in}} \quad (2)$$

where $p_{t,in}$ and $p_{t,out}$ are the total pressures of the inlet and outlet, respectively, and p_{in} is the average static pressure.

Figure 7 shows the curve of the C_{pt} and Mach number (Ma) at the outlet as the number of grids increases. The C_{pt} and Ma change gradually with increasing grid number. Keeping the amplitude within 5% indicates that the accuracy is up to standard. This study identified that a grid number above 1.4 million ensures accuracy according to the curve shown in Figure 6. This study finally adopted the numerical simulation method of meshing with 1.64 million grids after a comparative analysis that balanced the accuracy and computer performance.

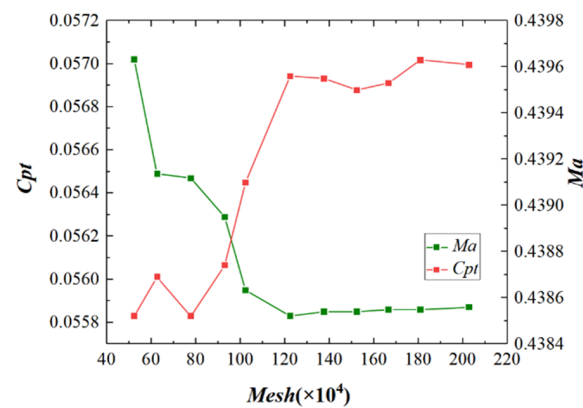


Figure 7. Performance parameters vary with the grid.

2.5. Numerical Calculation Method Check

To enhance the reliability of the calculation results, this study conducts a comparative analysis by comparing the prototype cascade experimental data (from the reference) with the simulation results. Four turbulence models ($K-\epsilon$, $K-\omega$, SST, SST- $\gamma-\theta$) were used, and their outputs were validated against experiments. As shown in Figure 8, oil flow visualization reveals a separation bubble (red dashed area) near the LE and obvious corner separation. The $K-\epsilon$, $K-\omega$, and SST models capture corner separation but not the SS bubble; $K-\epsilon$ and $K-\omega$ also underestimate the corner separation range along the blade height. The SST- $\gamma-\theta$ model accurately captures the SS separation bubble and boundary layer transition. Figure 8f shows that all four models' exit total pressure loss coefficient (C_{pt}) trends along the blade height agree with experiments. $K-\omega$ and SST models better reflect end-wall flow loss ($y/H < 0.1$) than $K-\epsilon$, with the SST- $\gamma-\theta$ model showing the best agreement and most precise separated flow prediction. Therefore, the SST- $\gamma-\theta$ model is selected as the turbulence model in this study.

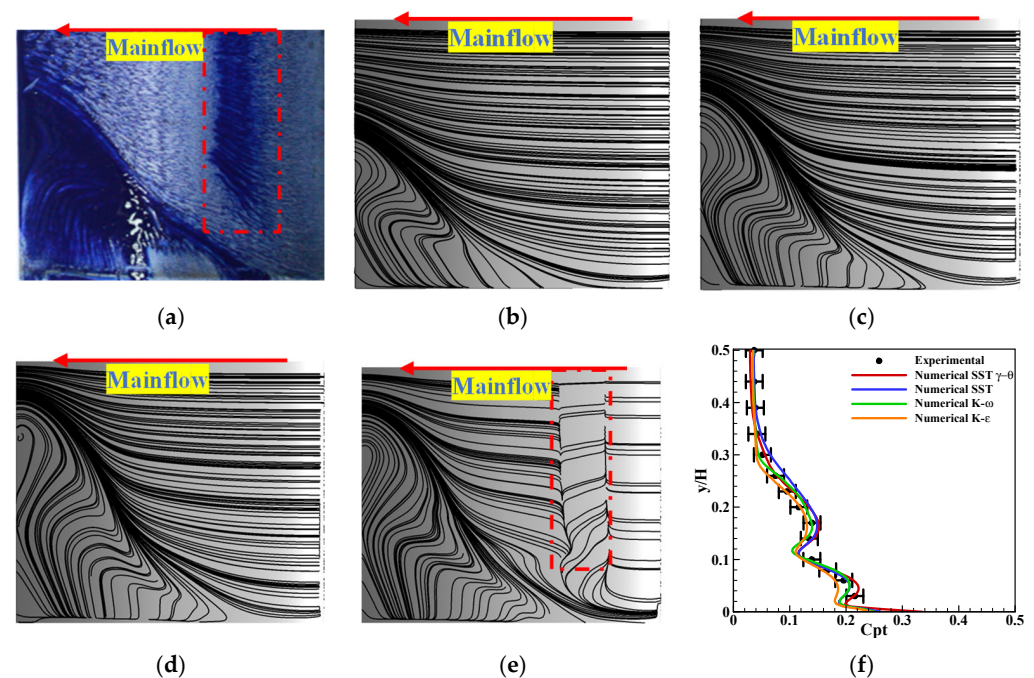


Figure 8. Accuracy verification. (a) Experimental result, (b) numerical result ($K-\epsilon$), (c) numerical result ($K-\omega$), (d) numerical result (SST), (e) numerical result (SST- $\gamma-\theta$), (f) C_{pt} curve.

3. Results and Discussion

3.1. Effect of Different Suction Cases on the Aerodynamic Performance and the Flow Field Structure

3.1.1. Effect of Suction Cases on the Aerodynamic Performance

Figure 9 shows the Cpt curve for each case. Cases 1 and 2 decrease the Cpt compared with the labyrinth seal prototype blade (ORI) case. Case 1 decreases the Cpt by 16%, which was the largest drop. Case 3 increases the Cpt and does not pump well. The suction effect undergoes a progressive increase in intensity as the suction position approaches the LE.

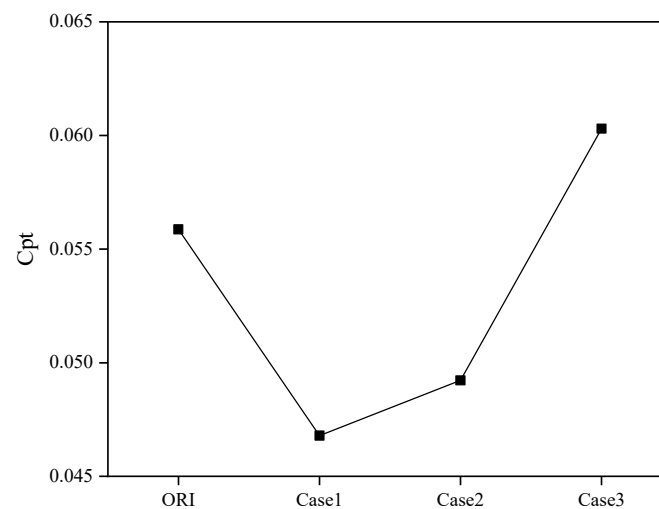


Figure 9. Cpt of different cases.

Figure 10 displays the impacts of different cases on the outlet Cpt. The main loss area is concentrated from the root to 30% H, which provides an essential basis for formulating loss reduction strategies. In this area, the Cpts of Cases 1 and 2 are 0.05 lower than for ORI. Case 3 increases the Cpt. Cases 1 and 2 significantly improve the aerodynamic performance, while Case 3 exacerbates the Cpt.

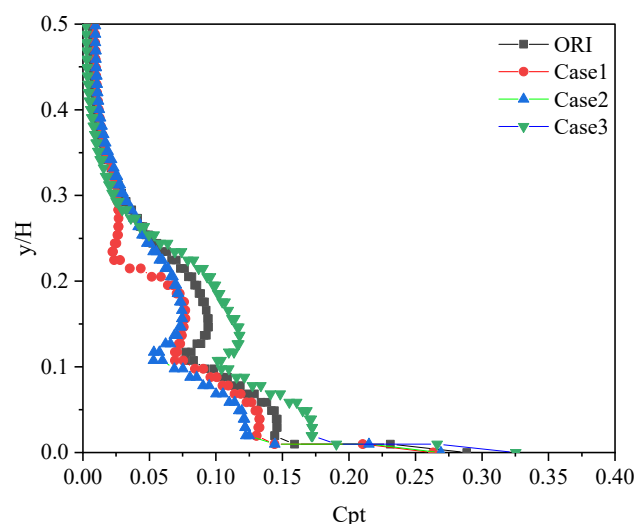


Figure 10. Spanwise distribution curves of outlet Cpts.

Figure 11 shows that the effect of end-wall suction on the blade loading capacity gradually weakens as H increases, and the effect is most potent at 5% H. Blade loading is manifested as the pressure difference between the SS and the pressure side, which can usually be reflected by the static pressure coefficient (Cps). The Cps from 40% c to the TE

increases too slowly, indicating a small diffusion capacity. In Cases 1 and 2, the suction at the LE and middle of the SS considerably reduces the C_p s envelope area. The flow field structure and flow capacity are improved, increasing the blade cascade load and diffusion capacity. The outlet static pressure is also increased. Low-energy fluid accumulates at the TE. Case 3 reduces the effective area of the passage, causing the outlet C_p s to decrease. This weakens the blade's resistance to pressure, diminishing its compressive performance.

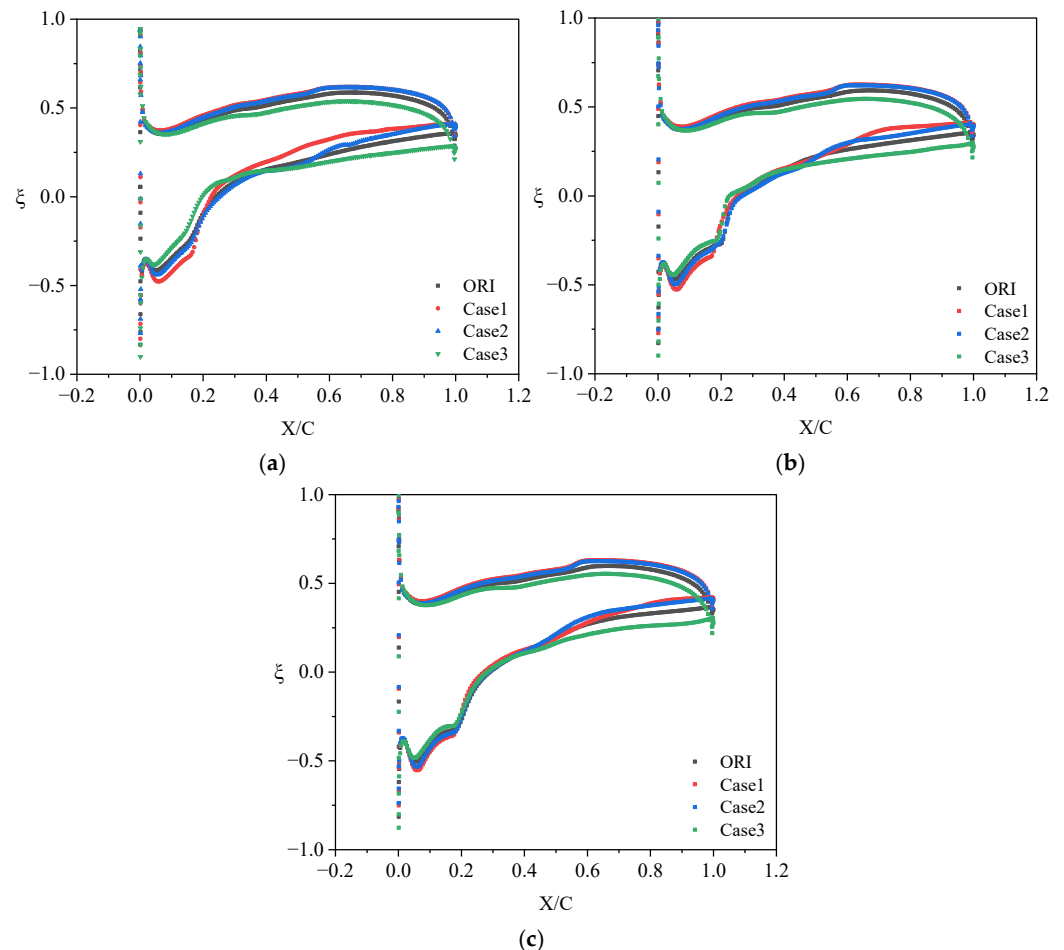


Figure 11. Distribution of C_p s curves for different cases. (a) 5% H, (b) 10% H, (c) 50% H.

3.1.2. Effects of Suction Cases on the Flow Field Structures

Figure 12 shows that the low-energy fluid cluster accumulates in the end region, decreasing the outlet C_p s in each case. Following suction, the peak value in the high-loss area is lowered. Cases 1 and 2 significantly reduce the C_{pt} and suppress the separation while expanding the scope of the main flow. Case 3 involves the high-loss region degrading to varying extents along the pitch-wise and spanwise directions. Case 3 expands the corner high-loss area, and the peak increases compared with other cases. Therefore, the suction at the TE of the SS (Case 1) has the best effect, which corresponds to Figure 10.

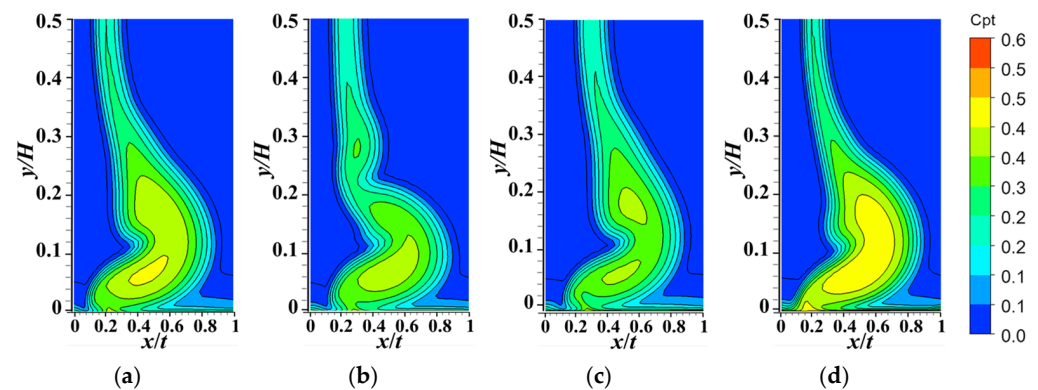


Figure 12. Cpt contours of each case's outlet. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

Figure 13 depicts the Cps and limit streamlines of the SS before and after modification, showing that the separation bubble is generated in a large low-pressure area near the LE. After the suction, the locations of the corner separations in Cases 1 and 2 are slightly delayed compared with ORI. Case 3 forms a spiral passage vortex (PV) in advance, significantly advancing the corner separation position. Cases 1 and 2 result in varying degrees of delayed separation in the direction of airflow and considerably shrink the outlet's high-loss area. Cases 2 and 3 expand the corner separation. Hence, the suction effect of Case 1 is better.

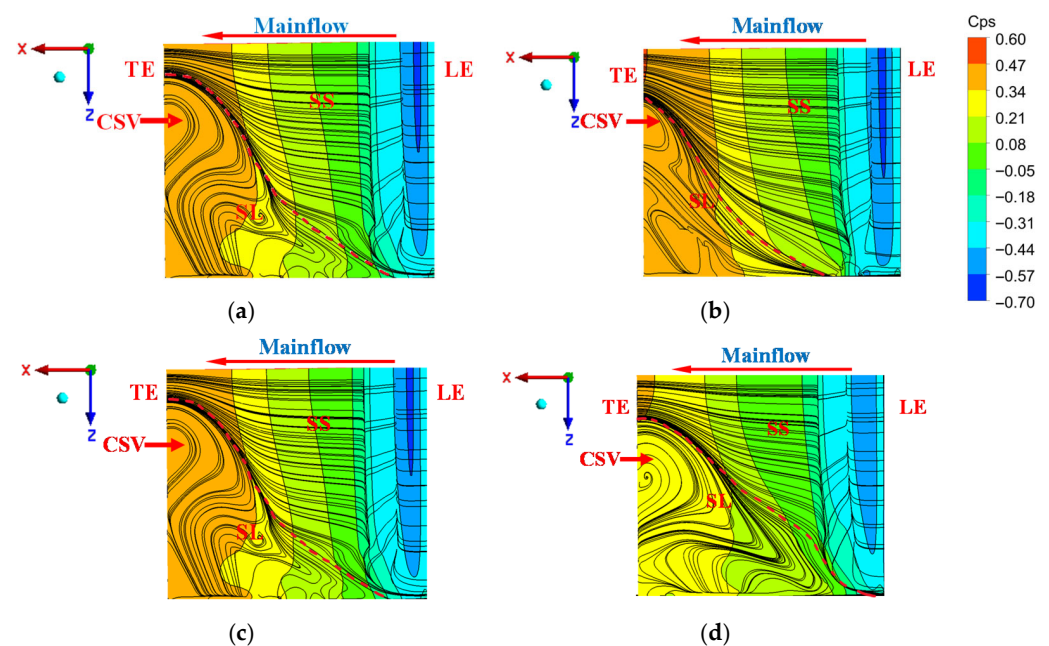


Figure 13. Cps distributions of different schemes. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

Figure 14 shows the distribution of Cpt and low-energy fluid in the passage before and after modification to further investigate the loss evolution and low-energy fluid accumulation within the flow passage. The accumulation of low-energy fluid within the passage can be visualized by iso-surfaces (Iso-surf) of zero axial velocity density. Figure 14 shows that the significant loss areas in Cases 1 and 2 are lower than those of ORI, with Case 1 having the most obvious effect. Meanwhile, the high-loss regions in both schemes are significantly reduced compared with the prototype. In Case 3, the high-loss regions extend to varying degrees along the spanwise and pitch-wise directions. At the LE, Cases 1 and 2 reduce the loss to different extents. However, the high-loss area in Case 2 increases slightly along the spanwise direction.

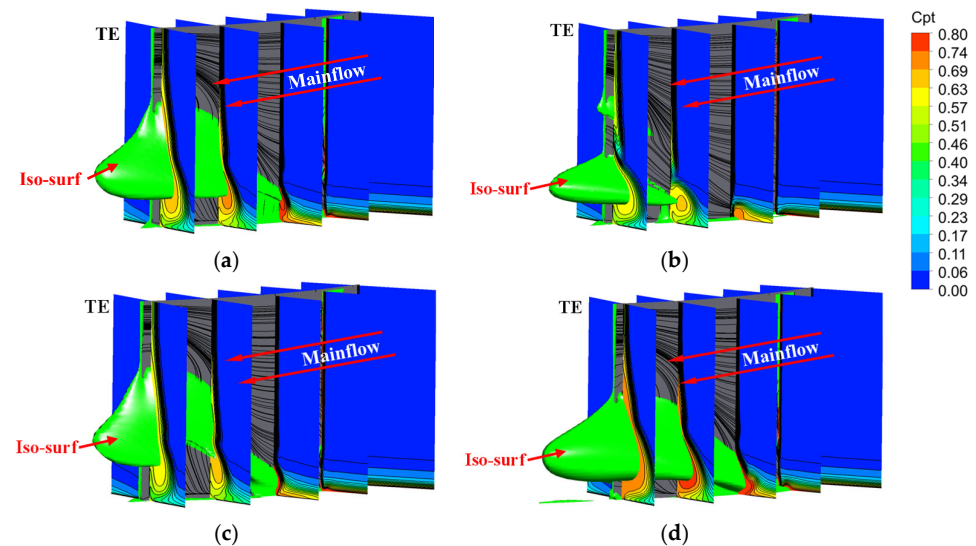


Figure 14. Distribution of low-energy fluid mass and C_{pt} in each case. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

The vortex structure is identified in the flow field using an iso-surface with a Q -criterion value of $9.06 \times 10^6 \text{ s}^{-2}$ (Figure 15). The color of the iso-surface represents the local velocity. The separations in Cases 1 and 2 are delayed, allowing for a smoother flow path and thus aiding the fluid flow. Cases 2 and 3 produce wall vortices and worsen the flow field structure. The separation in Case 3 is advanced, which causes the passage blockage to intensify and deteriorate the structure. Therefore, Case 1 presents the ideal flow field structure.

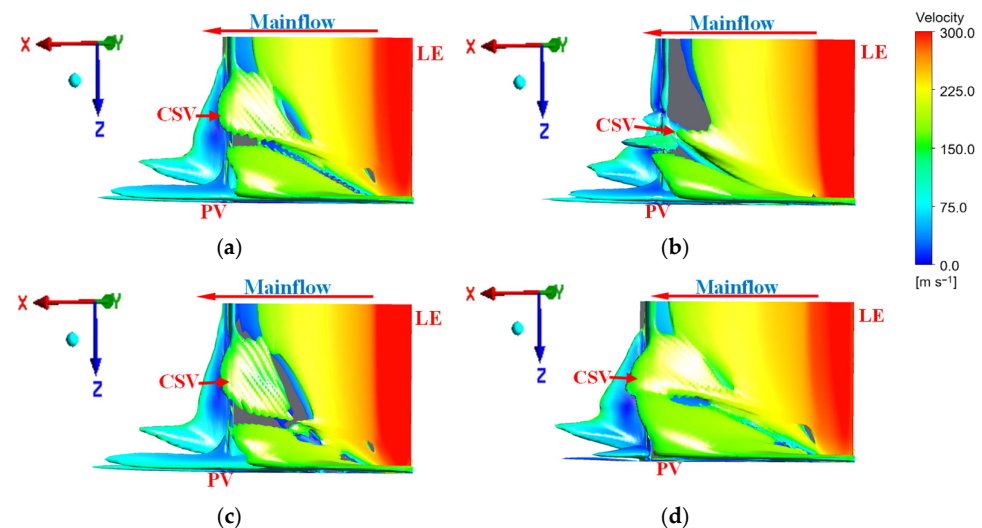


Figure 15. Three-dimensional vortex structures for different cases. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

As illustrated in Figure 16, the figure shows the limiting streamlines and the temperature of the suction and end-wall surface before and after the modification. Along the direction of the airflow, the temperature of the end wall and suction face within the compressor passage gradually increase. A higher-temperature region exists in the corner near the TE of the end-wall and SS. By comparing the vortex structure with the limiting streamlines on the SS, it is evident that the surface temperature is higher in the corner region where the flow accumulates at the TE. The suction near the LE and mid-span of the SS significantly reduces the corner separation range, decreases the low-energy fluid volume

near the TE of the root, and consequently diminishes the extent of the high-temperature region. In contrast, Case 3 applies suction only near the TE end wall, which does not significantly affect the corner separation. However, low-energy fluid within this region is drawn out due to the suction effect, enhancing heat transfer between the low-energy fluid and the main flow. Consequently, the temperature in the high-temperature region on the end wall and SS is reduced.

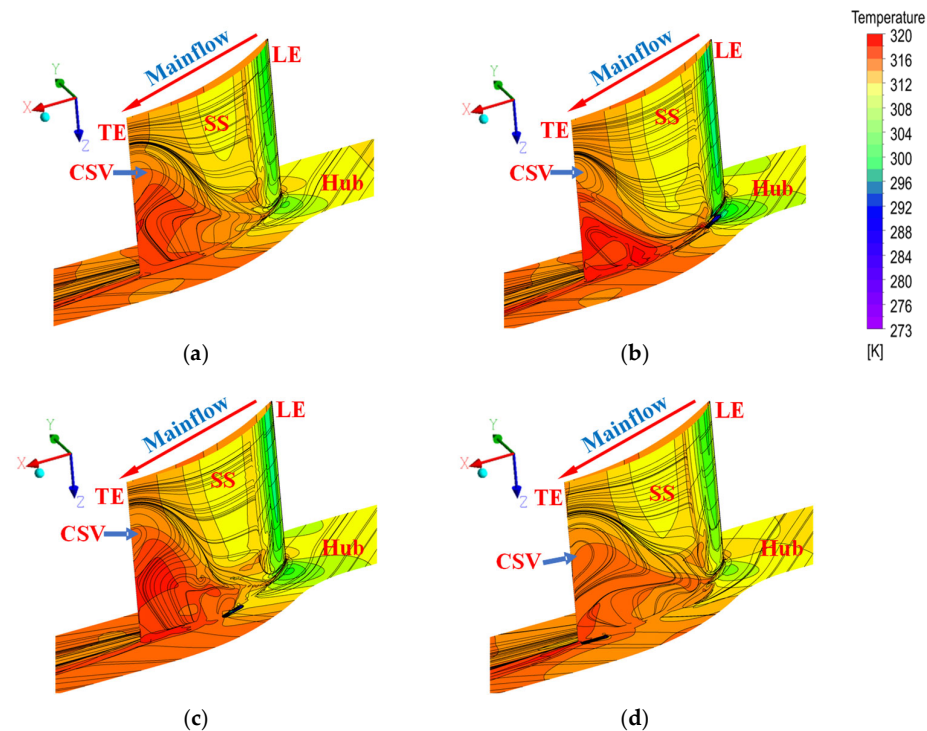


Figure 16. Limiting streamlines and temperature distribution. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

Figure 17 shows the temperature contours of outlet sections for the prototype blade and the three modified schemes. It can be observed that the ORI exhibits a higher-temperature region in the corner area, while the temperature in the main flow region is lower. All three modification schemes reduce the extent of the high-temperature region of the outlet to varying degrees compared with the ORI, as well as the magnitude of the high temperatures. Among them, Case 1 is most effective in suppressing the high-temperature region, significantly reducing its extent both along the spanwise and pitch-wise direction. In contrast, Case 2 only reduces the extent along the pitch-wise direction and cannot significantly reduce its extent along the spanwise direction.

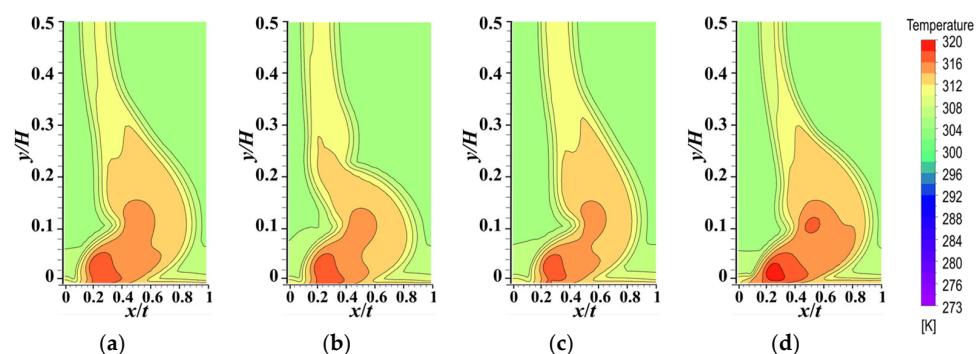


Figure 17. Temperature contours of each case outlet. (a) ORI, (b) Case 1, (c) Case 2, (d) Case 3.

3.2. Effect of End-Wall Suction on the Aerodynamic Performance and Flow Field Structure Under Different Incidence Angles

3.2.1. Effect of End-Wall Suction on the Aerodynamic Performance

The average Cpts in the ORI and Case 1 curves at different incidence angles are shown in Figure 18. The suction at differing angles decreases the Cpt. The suction effect remains consistent in the negative 4 incidence angle (N4) and positive 4 incidence angle (P4) conditions, leading to 10% and 7% reductions in Cpt, respectively. The suction under 0° leads to a 16% reduction in Cpt.

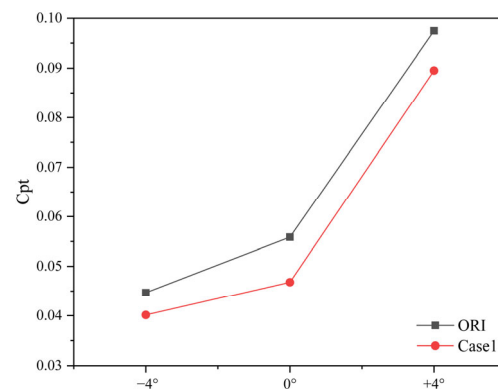


Figure 18. Cpts at different incidence angles.

Figure 19 shows that end-wall suction can reduce the pressure loss under different incidence angles compared with ORI. This can be attributed to the suction force exerted by the suction slot on the accumulated fluid, thereby reducing the flow loss. The main total pressure loss region is from the blade root to 30% H. Under the N4 or zero incidence angle (P0) conditions, the Cpt is significantly lower than that of ORI in the main loss area. As the angle increases from N4 to P4, the high-loss area widens from 30% H to 50% H, and the corner separation becomes more pronounced.

3.2.2. Effect of End-Wall Suction on the Flow Field Structure

Figure 20 shows that the total pressure loss decreases with the end-wall suction. The scope of the high-loss areas is subject to varying degrees of reduction. The end-wall suction under the N4 condition affects the separation inhibition. The concentration degrees of the fluid and high-loss area are smaller than in ORI. At this point, the total pressure loss is the lowest.

Figure 21 shows that the number of separation bubbles in the low-pressure area increases with increasing incidence angles. The corner separation is advanced, and the high-loss area expands toward H. When ORI is under the P4 condition, substantial low-energy fluid accumulation deteriorates the smooth environment. The separation is apparent here, and the flow field is relatively optimal at N4. After suction, P4 cascade separation is suppressed, and the flow field environment improves slightly, although this is not noticeable compared with other angles. Under the N4 condition, corner separation is delayed most effectively, and the total pressure loss is minimized.

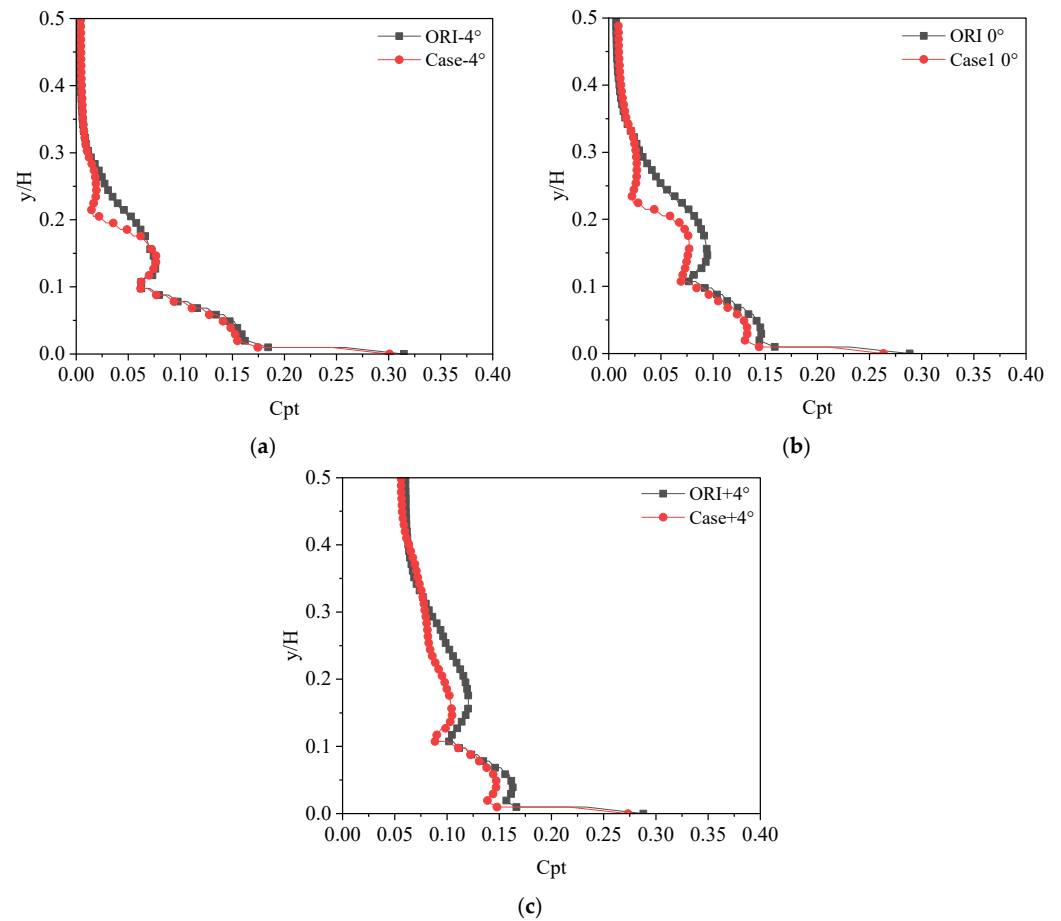


Figure 19. Distribution of Cpts at different incidence angles. (a) N4, (b) P0, (c) P4.

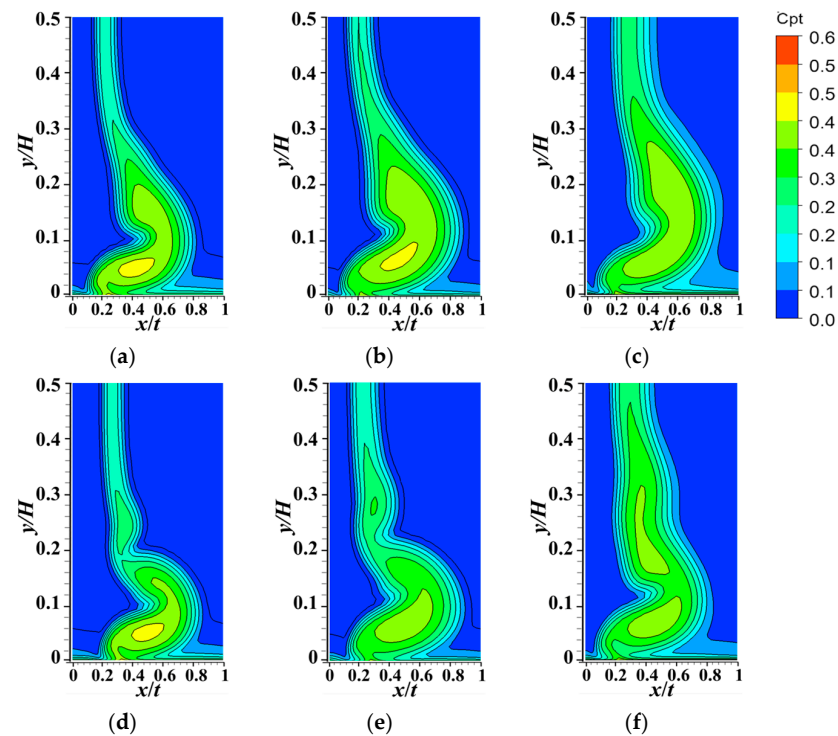


Figure 20. Cpt contours under different angles. (a) ORI-N4, (b) ORI-P0, (c) ORI-P4, (d) Case 1-N4, (e) Case 1-P0, (f) Case 1-P4.

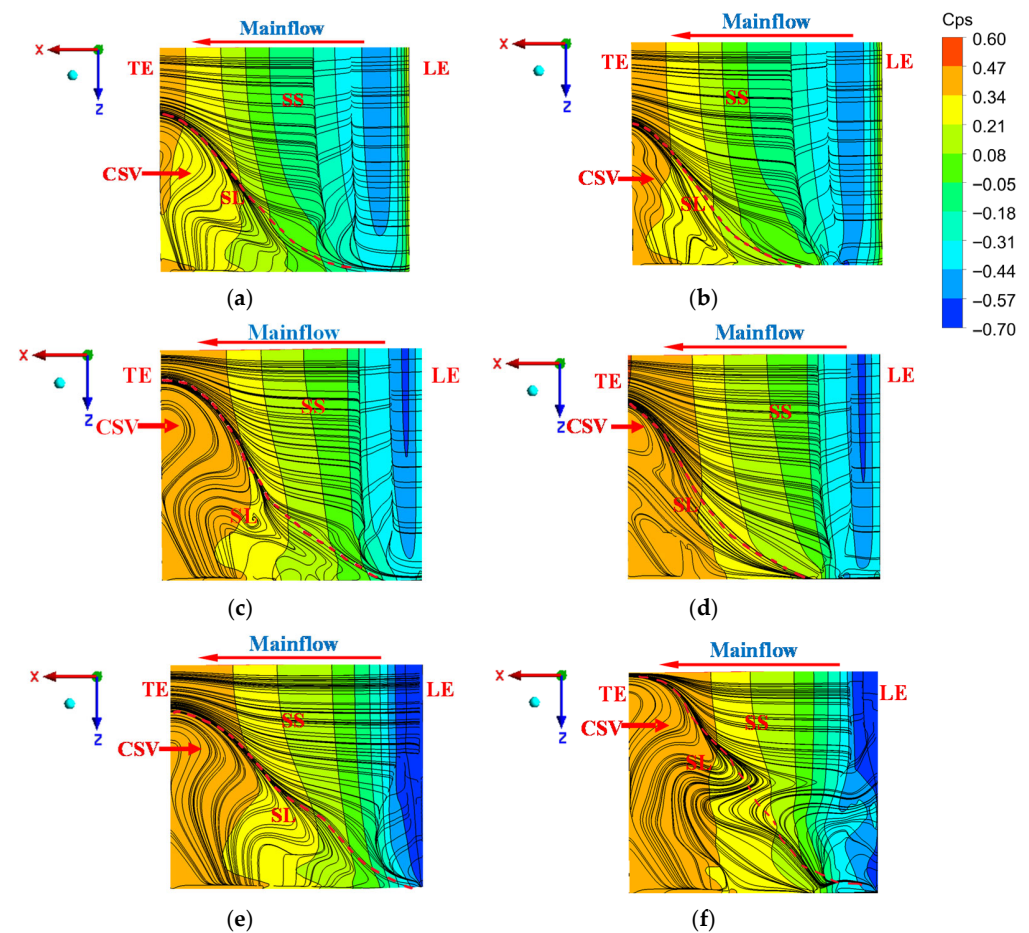


Figure 21. Cps contours of the blade surface under different angles. (a) ORI-N4, (b) Case 1-N4, (c) ORI-P0, (d) Case 1-P0, (e) ORI-P4, (f) Case 1-P4.

Figure 22 shows that with the increase in the angle, the level of fluid accumulation rises at the LE. Along the spanwise direction, the high-loss region increases. Simultaneously, the flow field gradually deteriorates. The operating condition with the smallest low-energy fluid volume and the most optimal structure is N4. After end-wall suction, the fluid volume in all cases decreases by varying degrees under different angle conditions compared with ORI. This configuration is known to reduce loss and elevate the flow environment. However, the peak after suction is larger than in ORI, and the range and peak of the high-loss region reach the maximum under the P4 condition.

Figure 23 displays the vortex structure under different angle conditions. The ORI's separation area expands as the incidence angle gradually expands. The concentrated shedding vortex (CSV) develops near the TE, significantly improving the pressure loss. The velocity of concentrated fluid increases under suction compared with ORI, and the separation phenomenon is delayed. End-wall suction markedly inhibits the flow loss; the effect is most evident under the N4 condition. Under the P0 condition, the attenuation effect on the PV is particularly significant, which significantly upgrades the flow field structure.

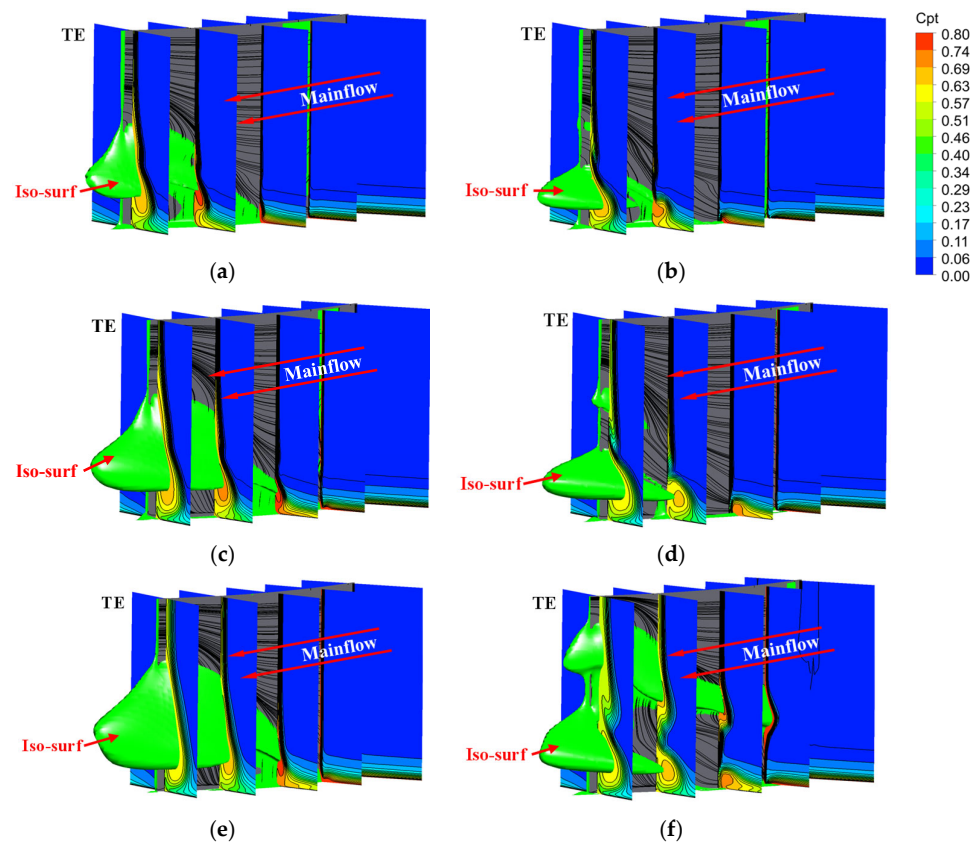


Figure 22. Distribution of low-energy fluid mass under each condition. (a) ORI-N4, (b) Case 1-N4, (c) ORI-P0, (d) Case 1-P0, (e) ORI-P4, (f) Case 1-P4.

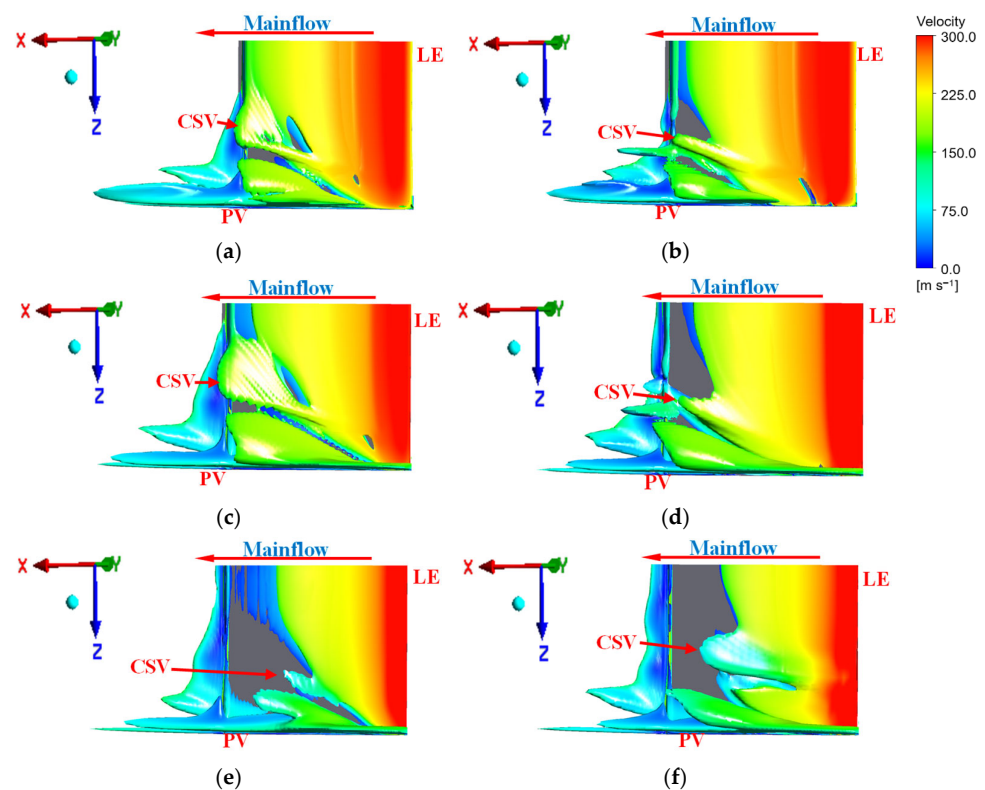


Figure 23. Three-dimensional vortex structures under different angles. (a) ORI-N4, (b) Case 1-N4, (c) ORI-P0, (d) Case 1-P0, (e) ORI-P4, (f) Case 1-P4.

Figure 24 demonstrates the limiting streamlines and temperature distributions on the SS and end-wall surface of the ORI and Case 1 at different incidence angles. The results indicate that the suction suppresses the corner separation gradually forming along the flow, while simultaneously delaying the onset of CSV. The temperature on the SS and end wall at the separation region is extremely high. This is due to the accumulation of low-energy fluid, which creates a recirculation at the TE, reducing heat exchange with the main flow. As the incidence angle increases from negative to positive, the temperature in the high-temperature region gradually rises and expands from the TE. In Case 1, the suction effect first increases and then decreases with increasing angle. The temperature-suppressing effect on the high-temperature region is not significant under the P4. Consequently, the suction applied near the LE suppresses the high-temperature region expansion more effectively than the ORI, with the most significant effect observed under P0 conditions.

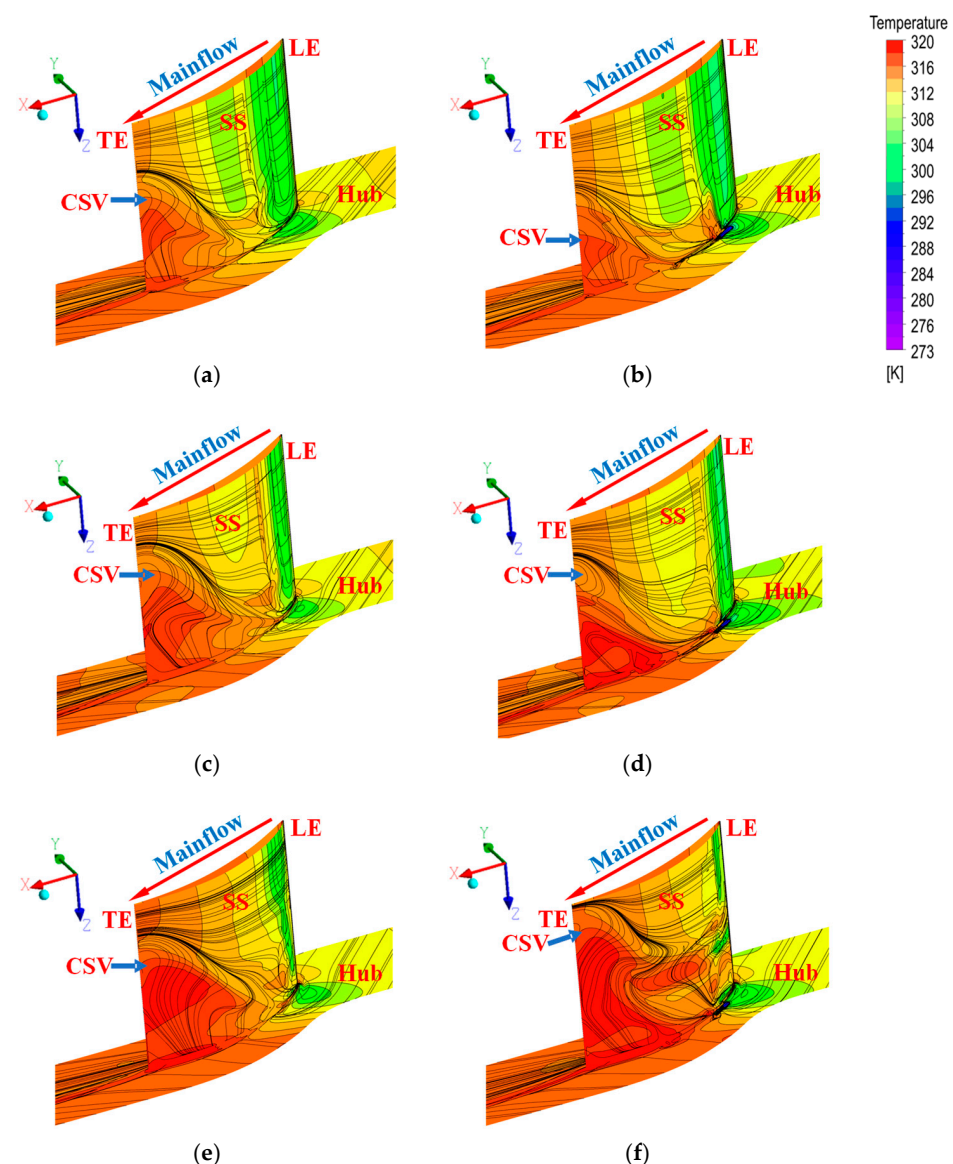


Figure 24. Limiting streamlines and temperature distribution under different angles. (a) ORI-N4, (b) Case 1-N4, (c) ORI-P0, (d) Case 1-P0, (e) ORI-P4, (f) Case 1-P4.

Figure 25 depicts the temperature contour of the outlet for the prototype and Case 1 under three incidence angles. Due to the corner separation, low-energy fluid accumulates extensively in the region, resulting in reduced heat transfer. Consequently, the extent of the

high-temperature region in the corner varies with operating conditions. As the incidence angle gradually increases from N4 to P4, the high-temperature region expands for both the ORI and Case 1, with the most severe expansion in the P4 along the spanwise and pitch-wise direction. However, the high-temperature region in Case 1 is smaller than that of the ORI under all three angles. In Case 1, the low-energy fluid accumulating in the corner is extensively drawn away, increasing the fluid velocity and reducing the high-temperature fluid volume. The end-wall suction suppresses corner separation while also inhibiting the development of the high-temperature region.

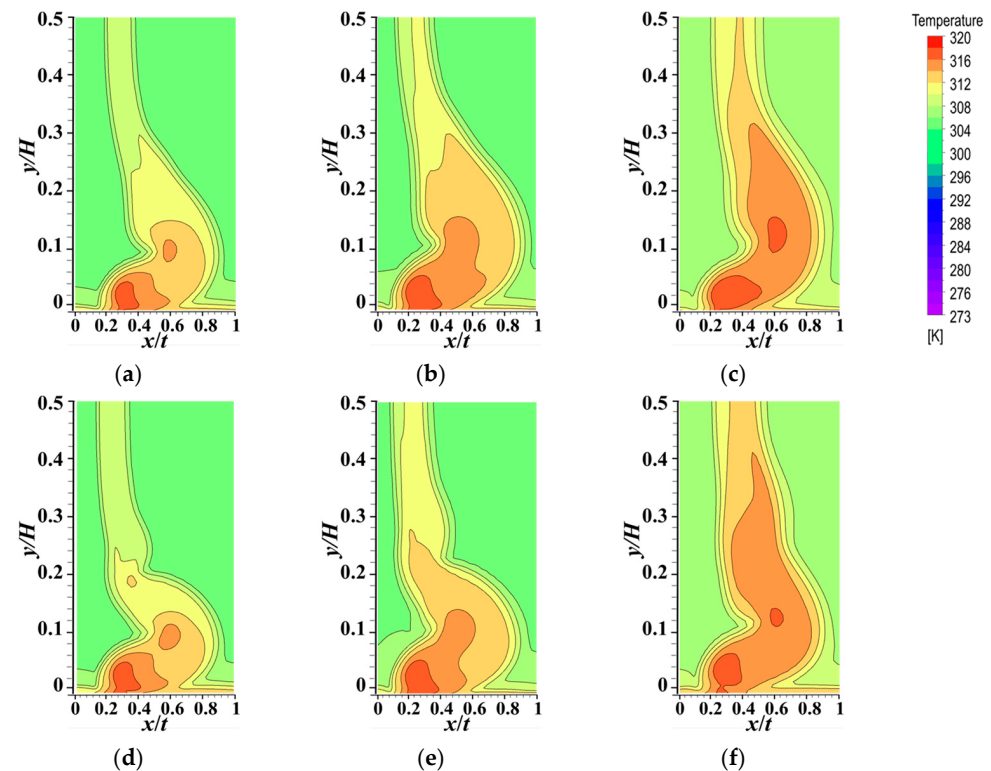


Figure 25. Temperature contours of each case outlet under different angles. (a) ORI-N4, (b) ORI-P0, (c) ORI-P4, (d) Case 1-N4, (e) Case 1-P0, (f) Case 1-P4.

To demonstrate the slot effect on the flow field more clearly, Figure 26 displays the vortex schematic diagrams of ORI and Case 1, where the black contour lines represent ORI and the red contour lines represent Case 1. First, the end-wall suction arrangement disperses the suction side vortex (SSV), thereby reducing the pitch-wise separation vortex. Second, it significantly reduces the CSV height, further reducing the wing-tip vortex. Meanwhile, the pitch-wise separation vortex reduction causes the PV to approach the suction surface, thereby increasing its influence range in the axial direction. Under the compression of the CSV and PV, the corner vortex (CV) located near the TE of the root is significantly reduced and adheres closely to the corner. The better flow field dramatically enhances the cascade diffuser performance.

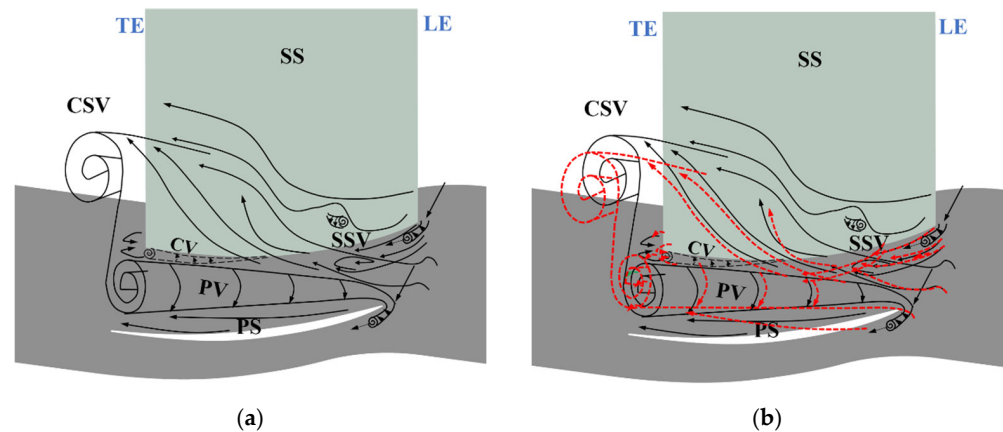


Figure 26. Comparison of vortex structures between the optimal case and ORI. (a) ORI, (b) Case 1.

4. Conclusions

This study investigated the influence of different suction positions at varying incidence angles on the aerodynamic performance and flow field structure of a compressor. The following conclusions were drawn:

- (1) The high-loss area is primarily distributed from the blade root to 40% H. The corner separation occurs when the compressor operates, and the PV in the boundary layer deteriorates the flow field structure. The accumulation of low-energy fluids brings about a significant decrease in compressor efficiency.
- (2) Applying suction at the LE significantly delays the corner separation, reducing the high-loss area. At the same time, it reduces the fluid temperature in the corner region, suppresses recirculation at the TE of the end-wall, lowers the surface temperature, and reduces the range of the high-temperature region. The Cpt is reduced by 16% in this case. Suction located in the middle of the cascade is not significantly different from that at the LE and may cause backflow. Suction at the TE increases the range of total pressure loss and worsens the flow field structure. Therefore, suction is best at 10% c to 20% c at the LE.
- (3) The Cpt is small when the incidence angle is negative. The low-energy fluid cluster blockage becomes more obvious as the angle increases. With the angle increasing from N4 to P4, the suction effect first rises and then decreases, and the high-temperature region gradually expands. The cascade diffusion capacity is better under P0 compared with other angles. However, regardless of the incidence angle, end-wall suction can reduce the total pressure loss, inhibit corner separation, and improve the flow field structure. End-wall suction reduces the volume of corner low-energy, high-temperature fluid, and its effect on suppressing surface temperatures is most pronounced under N4 operating conditions.

In the future, we will further optimize the suction schemes and active control strategies to continuously enhance the aerodynamic performance of the compressor and improve the flow field structure. Meanwhile, we will conduct in-depth analyses of the interaction mechanisms among multiple vortex systems and perform systematic experimental verification to consolidate the reliability of the numerical results and clarify the inherent correlations between key physical quantities. In addition, unsteady jet control and different structural forms of suction slots will be systematically investigated. Furthermore, thermal effects, as an important factor that influences compressor performance, will also be incorporated into the scope of subsequent research.

Author Contributions: Resources, software, supervision, W.X.; visualization, writing—original draft, Y.W.; methodology, project administration, S.Z.; funding acquisition, data curation, G.R.; formal analysis, validation, S.W. All authors have read and agreed to the published version of the manuscript.

Funding: This project is supported by the Doctoral Research Initiation Project of the Liaoning Provincial Natural Science Foundation (Grant ID:2025-BS-0334) and the Liaoning Provincial Department of Education Basic Research Youth Project (Grant ID: JYTQN2023069, LJKMZ20220565).

Data Availability Statement: All relevant data are included as figures or tables in the main article. Further inquiries can be directed to the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

The following abbreviations are used in this manuscript:

Ma	Mach number
CSV	concentrated shedding vortex
ORI	labyrinth seal prototype blade
LE	leading edge
TE	trailing edge
PS	pressure side
SS	suction side
Cpt	total pressure loss coefficient
Cps	static pressure coefficient
AVD	axial velocity density
PV	passage vortex
SSV	suction side vortex
CV	corner vortex

References

1. Hergt, A.; Meyer, R.; Engel, K. Experimental Investigation of Flow Control in Compressor Cascades. In *ASME Turbo Expo 2006: Power for Land, Sea, and Air*; American Society of Mechanical Engineers: New York, NY, USA, 2008; Volume 6, pp. 231–240.
2. Liu, H.P.; Li, D.Y.; Chen, H.L.; Zhang, D.F. Effects of Vortex Generator Jet on Corner Separation/Stall in High-Turning Compressor Cascade. *Trans. Tianjin Univ.* **2016**, *22*, 555–562. [[CrossRef](#)]
3. Zhong, J.J.; Yang, L.; Han, S.B. Numerical Investigation of the Secondary Flow Control in an Annual Compressor of Transonic Cascade with Different Length Endwall Fences. *J. Eng. Thermophys.* **2012**, *33*, 765–769.
4. Ma, Y.; Teng, J.; Zhu, M.; Qiang, X. Effective strategy of non-axisymmetric endwall contouring in a linear compressor cascade. *Proc. Inst. Mech. Eng.* **2022**, *236*, 1473–1487. [[CrossRef](#)]
5. Ding, Y.Y.; Wang, J.; Jiang, B.Y.; Xiao, Q.H.; Yang, X.P.; Wu, L.Y.; Xie, B.C. Numerical investigation of the effect of blade distortion laws on the corner flow separation of the axial-flow fan. *Aerosp. Sci. Technol.* **2023**, *138*, 108296. [[CrossRef](#)]
6. Zhong, J.J.; Wu, W.Y.; Han, S.B. Research Progress of Tip Winglet Technology in Compressor. *J. Therm. Sci.* **2021**, *30*, 18–31. [[CrossRef](#)]
7. Zhang, P.; Cheng, R.X.; Li, Y.H. Numerical Study on the Corner Separation Control for a Compressor Cascade via Bionic Herringbone Riblets. *Aerospace* **2024**, *11*, 90. [[CrossRef](#)]
8. Cao, Z.Y.; Song, C.; Gao, X.; Zhang, X.; Zhang, F.; Liu, B. Effects of Pulsed End-Wall Air Injection on Corner Separation and Vortical Flow of a Compressor Cascade. *Eng. Appl. Comput. Fluid Mech.* **2022**, *16*, 879–903.
9. Cui, W.W.; Liu, Y.Q.; Liu, F.S.; Ruan, C.L.; Yang, L.S.; Li, L.T.; Yao, F.; Wang, X.L.; Wang, C.P. Effect of Geometric Variation of Root Fillet on the Flow Characteristic of a Transonic Compressor Rotor. *J. Therm. Sci.* **2023**, *32*, 1407–1420. [[CrossRef](#)]
10. Zhang, Y.J.; Dong, X.; Zhang, Y.F.; Lu, X.G. Flow Control of Hub Corner Stall in a Highly Loaded Axial Compressor Cascade. *Int. J. Heat Fluid Flow* **2019**, *78*, 108434. [[CrossRef](#)]
11. Chen, S.W.; Gong, Y.; Zeng, C. Pulsed Suction Towards Unsteady Active Flow Control in an Axial Compressor Cascade Including Clearance Leakage Effects. *Appl. Therm. Eng.* **2023**, *219*, 119654. [[CrossRef](#)]
12. Guo, C.J.; Yang, X.D.; Han, J.A.; Zhong, J.J. Effect of a Bionic Blade Rib on the Loss Characteristics of a Highly Loaded Compressor Cascade. *Aerosp. Sci. Technol.* **2024**, *145*, 108848. [[CrossRef](#)]

13. Li, Y.H.; Wu, Y.; Zhou, M.; Su, C.B.; Zhang, X.W.; Zhu, J.Q. Control of the Corner Separation in a Compressor Cascade by Steady and Unsteady Plasma Aerodynamic Actuation. *Exp. Fluids* **2010**, *48*, 1015–1023. [[CrossRef](#)]
14. Zhang, H.D.; Yu, X.J.; Liu, B.J.; Wu, Y.; Li, Y.H. Control of Corner Separation with Plasma Actuation in a High-Speed Compressor Cascade. *Appl. Sci.* **2017**, *7*, 465. [[CrossRef](#)]
15. Li, L.T.; Song, Y.P.; Chen, F.; Liu, H.P. Flow Control Investigation of Steady and Pulsed Jets in Bowed Compressor Cascades. In *Turbo Expo 2016: Turbomachinery Technical Conference and Exposition*; American Society of Mechanical Engineers: New York, NY, USA, 2016; Volume 49699, p. V02AT37A025.
16. Matejka, M.; Popelka, L.; Safarik, P.; Nozicka, J. Influence of Active Methods of Flow Control on Compressor Blade Cascade Flow. In *Turbo Expo 2008: Power for Land, Sea, and Air*; American Society of Mechanical Engineers: New York, NY, USA, 2008; Volume 43161, pp. 591–599.
17. Cao, Z.Y.; Gao, X.; Zhang, T.; Liu, B. Flow Mechanism and Aspiration Strategies in an Ultra-Highly Loaded Supersonic Compressor Cascade. *Aerosp. Sci. Technol.* **2020**, *104*, 105989. [[CrossRef](#)]
18. Wang, H.J.; Mao, X.C.; Liu, B.; Qing, Y.S. Combined Flow Control with Full-Span Slot and End-Wall Boundary Layer Suction in a Large-Camber Compressor Cascade. *Aerosp. Sci. Technol.* **2021**, *119*, 107121. [[CrossRef](#)]
19. Cao, Z.Y.; Song, C.; Gao, X.; Zhang, X.; Liu, B. Computational Study on the Boundary Layer Suction Effects in the Supersonic Compressor Flow with a Bowed Blade. *J. Therm. Sci.* **2022**, *31*, 511–528. [[CrossRef](#)]
20. Moussavi, S.A.; Ghaznavi, A. Effect of boundary layer suction on performance of a 2 MW wind turbine. *Energy* **2021**, *232*, 121072. [[CrossRef](#)]
21. Liu, Y.W.; Sun, J.J.; Lu, L.P. Corner Separation Control by Boundary Layer Suction Applied to a Highly Loaded Axial Compressor Cascade. *Energies* **2014**, *7*, 7994–8007. [[CrossRef](#)]
22. Methel, J.; Forte, M.; Vermeersch, O.; Casalis, G. Experimental investigation on the effect of forward-facing steps and gaps combined with wall suction on boundary layer transition. *Exp. Fluids* **2022**, *63*, 21. [[CrossRef](#)]
23. Zhao, B.; Li, K.Y.; Wang, Y.P.; Wang, Z.Y. Theoretical Analysis of Convective Heat Flux Structure in the Incompressible Turbulent Boundary Layer on a Porous Plate with Uniform Injection and Suction. *Int. J. Therm. Sci.* **2022**, *171*, 107264. [[CrossRef](#)]
24. Zhang, B.; Liu, B.; Wang, H.; Mao, X.C. Effect of Endwall Suction on Aerodynamic Performance of Compressor Cascade with Tip Clearance at a Large Incidence Angle. *Proc. Inst. Mech. Eng. Part A J. Power Energy* **2021**, *235*, 1332–1343. [[CrossRef](#)]
25. You, F.H.; Li, X.J.; Lu, Q.; Zhang, H.G.; Chu, W.L. Numerical Investigation into a New Method of Non-Axisymmetric End Wall Contouring for Axial Compressors. *Energies* **2022**, *15*, 3305. [[CrossRef](#)]
26. Fan, Y.Q.; Tian, Z.T.; Zheng, J.Z.; Xin, J.C.; Lu, H.W. Effect of End Wall Boundary Layer Suction on the Performance of High Load Helium Compressor Cascade. *Prog. Nucl. Energy* **2024**, *170*, 105116. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.