GPPS-TC-2021-0170

Numerical investigation of an aggressive s-shaped compressor transition duct with combined boundary layer suction

Song Huang  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
huangsong@iet.cn  
Beijing, China

Chengwu Yang  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
yangchengwu@iet.cn  
Beijing, China

Shengfeng Zhao  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
zhaoshengfeng@iet.cn  
Beijing, China

Han Ge  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
hange@iet.cn  
Beijing, China

Hongzi Cheng  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
chenghongzhi@iet.cn  
Beijing, China

Xingen Lu  
Institute of Engineering Thermophysics, Chinese Academy of Sciences  
University of Chinese Academy of Sciences  
lux@iet.cn  
Beijing, China

ABSTRACT

Aggressive s-shaped compressor transition ducts are important components in the connection between the upstream boosters and the downstream high-pressure compressors. The flow has a strong three-dimensionality, which is easy to cause flow separation. Therefore, this paper takes an aggressive s-shaped compressor transition duct in a geared turbofan engine as a prototype and proposes a method for controlling the flow separation through combined boundary layer suction. The study found that combined boundary layer suction can reduce the total pressure losses to a greater extent. On the premise that the location of blade suction remains unchanged, the optimal location for the circumferential slot of hub suction is located at 20% of the axial chord length of the strut, whereby the total pressure loss coefficient decreases by about 30% compared with no suction. Besides, when the mass flow rate of suction accounts for 3% of the inlet mass flow rate, compared with the case where all suction is in the hub, the total pressure loss in another case with the mass flow rate of blade suction accounting for 0.5% and mass flow rate of hub suction accounting for 2.5% is further reduced by approximately 1.6%. The distribution of the mass flow rate for combined boundary layer suction has an optimal ratio.

INTRODUCTION

Aggressive s-shaped compressor transition ducts are important components in the connection between the upstream boosters and the downstream high-pressure compressors. Its flow path is an s-shaped flow path with struts and a large radial drop length ratio. The pressure distribution in the aggressive s-shaped compressor transition duct is completely different from that of the conventional straight flow path. The casing pressure is higher than that of the hub at the first turn of the transition duct, and it is opposite at the second turn. The connection, at the middle of the two turns of the hub and casing, generates an adverse pressure and a positive pressure gradient along the flow direction. The adverse pressure gradient allows the fluid to be easily separated near the hub. Coupled with the adverse pressure gradient of the suction surface of struts, the loading near the hub region in the transition duct increases sharply, resulting in a greater aerodynamics loss and an exit wake area, thereby reducing the work of struts and the uniformity of the outlet air, resulting in a rapid increase in the loss of the transition duct and the matching difficulty of the downstream high-pressure compressor (Walker et al., 2013). In recent years, to further promote the development of more competitive and environmentally friendly jet engine technology and reduce the weight and size of the engine. The focus of attention of researchers from various countries has turned to the aerodynamic design and the complex flow mechanism analysis of the aggressive s-shaped compressor.
transition duct with short axial length and large radial drop that breaks through the traditional design limit. Internationally, the EU’s 6th-generation engine framework agreement “Aggressive Intermediate Duct Aerodynamics (AIDA)” program comprehensively explored the aerodynamic design criteria and flow mechanism of transition ducts with a large radial drop length ratio and achieved a series of impressive research results.

In terms of the flow field of the transition ducts, (Bailey, 1997) first studied the internal flow field of a transition duct without struts and upstream and downstream constraints through experiments. This study found that the main factors that determine the internal flow field loss characteristics of the transition duct are the pressure gradient and curvature. The curvature and pressure gradient affect the development of the boundary layer Reynolds shear stress, and the pressure gradient affects the development of the boundary layer shape factor. (Britchford, 1998) examined the influence of the upstream booster on the flow in a conservatively designed transition duct. It was found that when a compressor is installed upstream of the transition duct, the wake induced by the blades mixes with the main flow in the transition duct, forming a streamwise vortex and increasing the total loss. Later, (Ortiz Dueñas et al., 2007) studied the effect of the axial length on the performance of the transition duct. The experiment tested three transition ducts with different axial lengths and reported only a small increase in loss when the axial length of the transition duct reached 74% of that of the length of the current conventional transition duct. However, when the axial length is reduced to 64% of the length of the current conventional transition duct, the flow separation near the hub is triggered, which greatly increased the loss. This is mainly because decreasing the length of the transition enhances the separation of the boundary layer and increases the total pressure loss increases accordingly. (Sonoda et al., 1997, 1998) used experiments and numerical calculations to study the influence of the downstream flow passage shape and the inlet boundary layer thickness on the complex flow field of the transition duct. The results show that compared with a straight passage downstream of the transitional duct, after switching to the curved passage, the boundary layer thickness and the total pressure loss increase significantly due to the effect of the reverse pressure gradient. And the inlet boundary layer thickness affects the shape of the vortex region near the end wall. When the inlet boundary layer is thin, the vortex region at the outlet exists in the form of a single vortex. However, when the inlet boundary layer reaches a certain thickness, a pair of anti-vortices forms in the vortex region, which further aggravates the total loss. (Narayanan et al., 2001) numerically simulated the transition duct connecting the inner and outer bypass with the suction mechanism. The flow field was compared with the one-dimensional calculation results. It is found that the simulation error of the total pressure loss is within 10%. In terms of the design of the transition ducts, (Britchford et al., 2001) studied the influence of the blade lean of the integrated upstream compressor outlet guide vane on the aerodynamic performance of the s-shaped transition duct. The study found that the lift generated by the blade lean can reduce the inverse pressure gradient on the inner wall of the s-shaped transition duct and improve the flow field near the hub. Further, (Walker et al., 2011) integrated the s-shaped transition duct with the upstream compressor outlet guide vane. It successfully reduced the length of the transition duct by 21% through the end wall forward sweep of the blade without deteriorating the overall performance of the flow field. (Wallin and Eriksson, 2006) established a geometric description method of the transition duct expressed by 4 independent parameters, and optimized the transition duct with eight struts by using the response surface model. Without restricting the radial distribution of the outlet airflow, the total pressure loss was reduced by 24% after optimization. However, the optimization loss was reduced by up to 16% by restricting the radial distribution of the outlet airflow. (Ghisu et al., 2007) used the genetic algorithm to optimize the design of the two-dimensional axisymmetric transition duct and controlled the centerline and area distribution rate of the transition duct through 11 variables during the optimization process, resulting in a reduction of the total pressure loss coefficient by 12.5%. In terms of the flow control of the transition ducts, (Karakasis et al., 2010; Naylor et al., 2010) of the White Laboratory in the United Kingdom realized the design of an aggressive transition duct by increasing the radial drop length ratio. The study found that when ΔR/L>0.5, a wide range of corner separation occurs near the trailing edge of the strut in the transition duct. And innovatively proposed to use a non-asymmetric end wall to control the flow separation near the junction of the trailing edge of the strut and the hub to reduce the loss of the aggressive transition duct. (Walker et al., 2013) carried out a numerical study and experimental verification on an s-shaped transition duct with hub suction under the support of the AIDA project. The boundary layer under the aggressive transition duct is adhered to the geometric surface by hub suction. The test proved that compared with the design of the traditional transition duct, it can shorten the axial length of the transition duct by about 30% and reduce the loss by about 20% through hub suction.

However, there is little study has been carried out on the optimal location of hub suction in the aggressive S-shaped duct and no consensus has been reached on the flow mechanism of the aggressive s-shaped compressor transition duct controlled by combined boundary layer suction. Therefore, this paper considers the aggressive s-shaped compressor transition duct of a geared turbofan engine as a prototype and proposes combined boundary layer suction as a means of reducing the total pressure loss. The complex flow mechanism of combined boundary layer suction on the aggressive s-shaped compressor transition duct is revealed.

**INVESTIGATED MODEL**

The compressor transition section includes five main design parameters: radial drop length ratio ΔR/L, Outlet-to-inlet area ratio \( A_{\text{out}}/A_{\text{in}} \), inlet-to-outlet height length ratio \( H_{\text{in}}/L \), inlet radius length ratio \( R_{\text{in}}/L \), and strut thickness chord length.
ratio $t/c$, as shown in Figure 1. The physical model in this paper is an aggressive s-shaped compressor transition duct based on a geared turbofan engine. The main design parameters are listed in Table 1.

![Figure 1 Schematic diagram of the aggressive s-shaped compressor transition section](image1)

### Table 1: Main design parameters of the transition section

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial drop length ratio $\Delta R/L$</td>
<td>0.8</td>
</tr>
<tr>
<td>Outlet-to-inlet area ratio $A_{out}/A_{in}$</td>
<td>0.86</td>
</tr>
<tr>
<td>Inlet-to-outlet height length ratio $H_{in}/L$</td>
<td>0.354</td>
</tr>
<tr>
<td>Inlet radius length ratio $R_{in}/L$</td>
<td>1.99</td>
</tr>
<tr>
<td>Strut thickness chord length ratio $t/c$</td>
<td>0.2</td>
</tr>
</tbody>
</table>

**NUMERICAL METHODOLOGY**

In the numerical calculations, the Reynolds averaged Navier-Stokes equations are solved to simulate the steady flow field in the aggressive s-shaped compressor transition duct. The turbulence model uses the calibrated shear-stress transport turbulence model. At the inlet boundary, a uniform standard total pressure and total temperature related to the axial flow direction are imposed. At the outlet, the average static pressure is imposed. The solid wall adopts an adiabatic non-slip boundary condition. Figure 2 shows a schematic diagram of the grid of the aggressive s-shaped compressor transition duct is given. The number of streamwise grid nodes is 497, the number of radial grid nodes is 73, and the number of circumferential grid nodes is 107. The O grids are generated to improve the orthogonality near blade surfaces, whereas H grids are applied to the inlet, blade passage, and exit. At the solid wall, $y+<1$, the total number of grid nodes is about 3.88 million, which satisfies the grid-independence requirement.

![Figure 2 Schematic diagram of the grid of the aggressive s-shaped compressor transition duct](image2)

To verify the credibility of the numerical calculations in this paper, numerical simulations are carried out for the compressor transition duct in literature (Bailey, 1997). The geometric parameters of experiments and strut locations are shown in Figure 3. There are eight struts in the transition duct, the blade shape is NACA 65, the chord length is 190.9mm, and the maximum thickness-to-chord-length ratio is 0.12. In the numerical simulations, the experimentally measured
velocity distribution at location 1 in Figure 3(a) is assigned at the inlet, and the standard atmospheric pressure is assigned at the outlet. Figure 4 shows the dimensionless streamwise velocities at the traverse stations 2 and 11, which are normalized using the average streamwise velocity of 28.3 m/s at the inlet. It can be seen that the streamwise velocity distribution trend obtained by the numerical simulations accords with the experimental values. The streamwise velocity deviation near 20% of the span height at exit is mainly caused by the current RANS turbulence model cannot accurately simulate the influence of the curvature of the transition ducts. In summary, the overall error of the numerical simulation method in this paper is relatively small, which satisfies the accuracy of engineering calculations and could be used to predict aerodynamic performance and analyze the flow mechanism of the aggressive s-shaped compressor transition duct.

![Figure 3 Schematic diagram of the compressor transition duct of Loughborough University](image3)

![Figure 4 Dimensionless streamwise velocities at different flow directions](image4)

**RESULTS AND DISCUSSION**

**Influence of combined boundary layer suction on the aggressive transition duct**

**Location of combined boundary layer suction**

For the boundary layer suction structure, two methods can be used. One involves using an actual suction slot structure to simulate the effect of boundary layer suction, and the other uses a simplified physics model to simulate the effect of boundary layer suction. The actual suction slot structure can simulate the influence of the suction angle on the performance of the blade and the flow field in the suction slot. However, due to the three-dimensionality of the blade and the complicated sectional shape of the suction slot, this method requires a lot of work to generate the suction slot through grid blocks, which is time-consuming for engineering calculations. The simplified physics model is used to simplify the suction slot into a curved surface parallel with the blade surface. The boundary layer suction is performed along the normal direction of the surface, regardless of the flow inside the suction slot. Research by (Dang et al., 2003) of MIT indicates that the calculated flow field using the real suction geometry model and the simplified physical model can basically be in good agreement, and the numerical calculation results are also very close, which is very suitable for engineering research.

According to the results in the literature (Zhang, 2007), the best suction location of the blade surface is located near the location where the strength of the separated vortex reaching the highest. Therefore, on the basis that the total suction mass flow rate occupying 3% of the inlet flow rate, the location of the suction hole of the blade is fixed at 80% of the chord length of the struts. The span height of the holes ranges from 5% of the span height to 50% of the span height, the number...
is eight, the diameter is 6.8mm, and the suction flow rate is fixed at 1% of the inlet mass flow rate. The mass flow rate of the hub suction is fixed at 2% of the inlet mass flow rate. The hub suction locations are the inlet of the aggressive s-shaped compressor transition duct, 20% of the chord length of the strut, 40% of the chord length of the strut, 60% of the chord length of the strut, and 80% of the chord length of the strut, respectively. They are marked as Case 5A_MIX, Case 5B_MIX, Case 5C_MIX, Case 5D_MIX, and Case 5E_MIX, as shown in Figure 2. For the aggressive s-shaped compressor transition duct that does not take any suction, it is recorded as Case 5, which is the prototype.

To analyze the effect of different hub suction locations on the aerodynamic performance and flow mechanism of the aggressive s-shaped compressor transition duct. Figures 5 and 6 show the total pressure loss coefficient and the radial distribution of the total pressure loss coefficient of combined boundary layer suction under different hub suction locations. It can be seen that the combined boundary layer suction reduces the total pressure loss of the aggressive s-shaped compressor transition duct. Compared with Case 5, under the premise that the suction location of the blade remains unchanged, the best location for the circumferential hub suction slot is 20% of the axial chord length of the strut, and the total pressure loss coefficient is reduced by about 30%. When far away from this location, the suction effect decreases. The decrease in the total pressure loss coefficient of Case 5E_MIX is mainly due to the decrease near the tip region. Compared with the Case 5E_MIX, the total pressure loss coefficient of Case 5B_MIX is further reduced near 75% of the span height, which is mainly due to the reduction in the radial migration of the fluid on the blade surface and the decrease in the mixing strength of the wake caused by the corner separation. The detailed reasons are discussed in the following analysis.

![Figure 5 Total pressure loss coefficient of combined boundary layer suction under different hub suction locations](image1)

![Figure 6 Radial distribution of the total pressure loss coefficient of combined boundary layer suction under different hub suction locations](image2)

Figure 7 shows the total pressure loss coefficient along the flow direction for combined boundary layer suction under different hub suction locations. Compared with Case 5, the total pressure loss of Case 5B_MIX when passing through the axial position of the hub suction is significantly reduced, but the total pressure loss almost does not change suddenly when passing through the axial position of the suction hole on the blade surface. For Case 5E_MIX, the change in total pressure loss is smaller than that of Case 5B_MIX. In short, the reduction in the total pressure loss of the combined boundary layer suction is mainly due to the hub suction. The growth rate of the total pressure loss is reduced in the transition duct located at 40%~60% of the streamwise length, and there is almost no difference in the growth rate at the remaining axial locations, which means that the combined boundary layer suction under different hub suction locations in the s-shaped transition duct hardly affects fluid mixing loss upstream and downstream of the s-bend.

Figure 8 shows the limiting streamlines of the suction surface and hub of the strut of combined boundary layer suction under different hub suction locations. Compared with no suction, the combined boundary layer suction greatly reduces the range of corner separation. Compared with Case 5, the corner separation of the strut in Case 5B_MIX almost disappears, which greatly reduces the secondary flow loss, and consequently reduces the total loss coefficient. However, Compared with Case 5, Case 5E_MIX slightly reduces the spanwise range of the corner separation of the strut. From the hub location of about 60% chord length of the strut, fluid migrates upwards due to the effect of the radial migration. However, after passing through the suction holes on the blade surface, the radial migration ability is weakened. The fluid flows along the streamwise direction on the blade surface.
Figure 7 Total pressure loss coefficient along the flow direction of combined boundary layer suction under different hub suction locations

Figure 8 Limiting streamlines of the suction surface and hub of the strut of combined boundary layer suction under different hub suction locations

Figure 9 shows the streamwise vortices on different planes of combined boundary layer suction under different hub suction locations. Compared with those of Case 5, the strength and range of the wall vortex (WV) of Case 5B_MIX are greatly reduced, and the corner vortex (CV) at the exit of the strut formed by mixing the WV and the horseshoe vortex (HV) near the hub almost disappears. In other words, the strength and range of the suction surface corner vortex (SSCV) and the pressure surface corner vortex (PSCV) of the downstream plane VII are small, and thus the total loss is greatly reduced. However, compared with Case 5, it can be seen from plane VII that there are two pairs of vortex core regions with opposite vortices in Case 5E_MIX. The PSCV and SSCV near the hub do not mix with the shedding vortex near the tip of the blade. Therefore, the loss near 75% of the span height is greatly reduced.

Figure 9 Streamwise vortices on different planes of combined boundary layer suction under different hub suction locations

Figure 10 shows the static pressure coefficient of the suction surface at different spanwise heights of the combined boundary layer suction under different hub suction locations. It can be seen that the static pressure coefficient of the blade surface increases rapidly after the suction hole on the blade surface, forming a larger "secondary diffusion capacity", and
then accelerating, the static pressure coefficient decreases to the trailing edge. For different combined boundary layer suction conditions, the value of the static pressure coefficient at the trailing edge location is almost the same under the same span height. At 50% span height, the static pressure coefficient at the trailing edge is -0.3.

Figure 10 Static pressure coefficient of the suction surface at different spanwise heights of combined boundary layer suction under different hub suction locations

Mass flow rate of combined boundary layer suction

Under the condition of a certain total mass flow rate of combined boundary layer suction, which accounts for 3% of the inlet mass flow rate, we explore the ratio of the mass flow rate of blade suction to hub suction to see whether there is an optimal ratio for combined boundary layer suction. Figure 11 shows the total pressure loss coefficient under the different mass flow rates of combined boundary layer suction. It can be seen that compared with hub suction, combined boundary layer suction reduces the total pressure loss. In particular, under the scheme of blade suction accounts for 0.5% of mass inlet flow rate, and hub suction accounts for 2.5% of mass inlet flow rate, the total pressure loss is reduced by about 1.6%. It can be inferred that compared with hub suction, the combined boundary layer suction can further reduce the loss of the transition duct, but the distribution ratio of the mass flow rate of blade suction to hub suction needs to be considered. Figure 12 shows the radial distribution of the total pressure loss coefficient under the different mass flow rates of combined boundary layer suction. It can be seen that compared with the blade without suction, under the scheme of blade suction accounts for 0.5% of mass inlet flow rate and hub suction accounts for 2.5% of mass inlet flow rate, the reduction of the total pressure loss coefficient is mainly from the region above the middle span. It is mainly because the corner separation of the strut is further reduced. The radial migration of the fluid almost disappears. Under the scheme of blade suction accounts for 2.0% of mass inlet flow rate and hub suction accounts for 1.0% of mass inlet flow rate, the total pressure loss coefficient increases near the hub region. This is mainly due to the increase in momentum exchange caused by the blade suction, which gradually increases the mixing loss in the hub region. The detailed reasons are analyzed below.

Figure 11 Total pressure loss coefficient under the different mass flow rates of combined boundary layer suction

Figure 12 Radial distribution of the total pressure loss coefficient under the different mass flow rates of combined boundary layer suction

To study the effect of the different distribution of combined boundary layer suction mass flow rate on the performance and flow mechanism of the aggressive s-shaped transition duct. Figure 13 shows the limiting streamlines on the suction surface and hub of the strut under the different mass flow rates of combined boundary layer suction. It can be seen that compared with the blade without suction, under the scheme of blade suction accounts for 0.5% of mass inlet flow rate, the
radial migration range of the limiting streamline upward in the hub region almost disappears, so that the secondary flow loss is reduced. However, compared with the blade without suction, under the scheme of blade suction accounts for 2.0% of mass inlet flow rate, the fluid on the blade surface will no longer migrate upward in the radial direction after flowing through the suction holes on the blade surface. However, the excessive suction flow rate near the hub region causes a local separation vortex to form. As a result, the separation vortex that develops along the flow direction falls off, and the loss in the hub region increases.

Figure 13 Limiting streamline on the suction surface and hub of the strut under the different mass flow rates of combined boundary layer suction

Figure 14 shows the streamwise vortices on different locations under the different mass flow rates of combined boundary layer suction. As shown in Figure 14, compared with the blade without suction, under the scheme of blade suction accounts for 0.5% of mass inlet flow rate, due to the fluid on the blade surface hardly migrates upward in the radial direction, it can be seen from plane VII that the high vorticity near the blade tip region is reduced, and the distribution along the span height is more uniform, so the total pressure loss is reduced. However, under the scheme in which blade suction accounts for 2.0% of mass inlet flow rate, a separation vortex on the blade surface develops along the streamwise direction. The range of high vorticity on plane VII near the hub region increases, enhancing the total pressure loss near the hub region, and the effect of combined boundary layer suction slightly deteriorates.

Figure 14 Streamwise vortices on different locations under the different mass flow rates of combined boundary layer suction

Figure 15 shows the static pressure coefficient of the suction surface at different spanwise heights under the different mass flow rates of combined boundary layer suction. It can be seen that when the total suction mass flow rate is constant, as the proportion of the mass flow rate of blade suction increases, the increase rate in the static pressure coefficient of the blade surface after the hub suction slot slows down, which means that the expansion capacity decreases. However, the increase in the suction flow rate on the blade surface improves the secondary diffusion capacity of the blade. Compared with the situation at 5% and 50% of span height, the secondary diffusion amplitude is higher at 25% of span height. Under the different mass flow rates of combined boundary layer suction, the static pressure coefficient reaching the trailing edge of the blade is almost the same, indicating that the distribution of mass flow rate of combined boundary layer suction has little effect on the overall static pressure rise capacity across the blade. It mainly changes the flow loss characteristics by influencing the flow structure on the blade surface.
CONCLUSIONS

1 Combined boundary layer suction can reduce the total pressure loss of aggressive s-shaped compressor transition ducts. Compared with the prototype, when the mass flow rate of the blade suction and mass flow rate of hub suction are 1% and 2%, respectively. Under the premise that the location of the blade suction is fixed at 80% of the axial chord length of the strut, the best streamwise location for the circumferential slot of hub suction is 20% of the axial chord length of the strut. The total pressure loss coefficient is greatly reduced by about 30%. As the locations move away from the optimal configuration, the effect of combined boundary layer suction decreases.

2 Under the premise of a fixed total mass flow rate of combined boundary layer suction. Compared with the scheme of blade suction accounts for 0% of mass inlet flow rate, the total pressure loss of the scheme where blade suction accounts for 0.5% of inlet mass flow rate and hub suction accounts for 2.5% of inlet mass flow rate is further reduced by about 1.6%. However, when the mass flow rate of blade suction exceeds 0.5% of the inlet mass flow rate, the total pressure loss increases. It can be inferred that compared with hub suction, the combined boundary layer suction further reduces the total pressure loss of the s-shaped compressor transition duct, but the distribution ratio of the mass flow rate of blade suction to hub suction needs to be considered.

3 Combined boundary layer suction mainly affects the radial upward migration range of the limiting streamline in the hub region, thereby changing the development of secondary flow loss close to the tip region along with the downstream of the strut. When the high vorticity region near the blade tip region is reduced and the distribution along the span height is more uniform, the total pressure loss is small. Besides, at different span heights, the static pressure coefficient of the suction surface of the strut varies with different combined boundary layer suction schemes, the combined boundary layer suction can significantly increase the secondary diffusion capacity near the hub region.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \omega )</td>
<td>Total pressure loss</td>
</tr>
<tr>
<td>( \omega_s )</td>
<td>Streamwise vortex</td>
</tr>
<tr>
<td>( C_p )</td>
<td>Static pressure coefficient</td>
</tr>
<tr>
<td>( S )</td>
<td>Dimensionless span height</td>
</tr>
<tr>
<td>( U )</td>
<td>Meridional velocity</td>
</tr>
<tr>
<td>( \Delta R )</td>
<td>Radial drop height</td>
</tr>
<tr>
<td>( L )</td>
<td>Length of the s-shaped compressor transition duct</td>
</tr>
<tr>
<td>( A_{out} )</td>
<td>Outlet area</td>
</tr>
<tr>
<td>( A_{in} )</td>
<td>Inlet area</td>
</tr>
<tr>
<td>( H_{in} )</td>
<td>Inlet height</td>
</tr>
<tr>
<td>( R_{in} )</td>
<td>Inlet radius</td>
</tr>
<tr>
<td>( t )</td>
<td>Strut thickness</td>
</tr>
<tr>
<td>( c )</td>
<td>Strut chord length</td>
</tr>
<tr>
<td>( u_s )</td>
<td>Percentage of the mass flow rate of blade suction</td>
</tr>
<tr>
<td>( z )</td>
<td>Axial location of the s-shaped compressor transition duct</td>
</tr>
<tr>
<td>( C )</td>
<td>Axial length of the s-shaped compressor transition duct</td>
</tr>
<tr>
<td>( \text{Exp} )</td>
<td>Experiment results</td>
</tr>
<tr>
<td>( \text{Cal} )</td>
<td>Calculation results</td>
</tr>
</tbody>
</table>

Figure 15 Static pressure coefficient of the suction surface at different spanwise heights under the different mass flow rates of combined boundary layer suction
ABBREVIATIONS
CV Corner vortex
SS Suction surface
WV Wall vortex
PSCV Pressure side corner vortex
SSCV Suction side corner vortex
HV Horseshoe vortex
LE Leading edge
TE Trailing edge
AIDA Aggressive intermediate duct aerodynamics

ACKNOWLEDGMENTS
The authors wish to acknowledge the financial support of the Technology Project of China (Grant No. 2019-II-0004-0024), the Civil Aircraft Scientific Research Project (Grant No. MJ-2017-D-29), and the National Natural Science Foundation of China (Grant No. 51836008) for this work.

REFERENCES
Zhang Hualiang (2007) Investigation on application of dihedral/swept blade and boundary layer suction to control vortex Configuration In Compressor Cascades, Dissertation, Harbin Institute of Technology.