FLOW CHARACTERISTICS IN THE AIR BLEEDING MANIFOLD

Teng Jin-fang1 *, He Pan1, Wu Dong-run1, Gu Wei2, Qiang Xiao-qing1
1 School of Aeronautics and Astronautics, Shanghai Jiao Tong University, Shanghai, China
2 AECC Commercial Aircraft Engine Company, Shanghai, China
Email: tjf@sjtu.edu.cn

ABSTRACT
This paper investigates the internal flow characteristics of an aggressive S-shaped air bleeding manifold under typical working condition with the throttle cone opening degree of 15% and the inlet Mach number of 0.34 by experimental and numerical simulation methods. Based on the experimental results of total pressure recovery coefficient, the realizale k-ε model was selected as a suitable turbulence model for the air bleeding manifold numerical simulation by analyzing the numerical results of different turbulence models. The numerical simulation results show that the flow separation occurs at the upper wall and lower wall near the first bend, which also leads to the vortex pair and causes the non-uniformity of the flow field in the exit. The curvature of the end walls and the cross-sectional area variation of air bleeding manifold have greatly influence on the flow structure. The severe deflectionon the lower wall of the first bend is the main factor causing the flow separation and total pressure loss. The mass-averaged total pressure recovery coefficient in exit is 0.9818, and the synthesis distortion index DC(60) in the exit is 0.1334.

INTRODUCTION
Air bleeding manifold is an important component in the active clearance control system of high-pressure turbine. The study of the internal characteristic and the loss mechanism will provide theoretical basis for optimal design of the air bleeding manifold. In this paper, the air bleeding manifold is an aggressive S-shaped duct with variable geometric cross-section. The study on flow characteristics of S-shaped duct in aero-engine is focused on compressor and turbine intermediate duct [1-3], inlet [4-5] and nozzle [6]. It is very famous that AIDA-Aggressive Intermediate Duct Aerodynamics project within the 6th framework program funded by the European Commission two projects dealing exclusively or partly with the topic of intermediate ducts. The outcomes of the AIDA project gives turbine and compressor engineers access to a whole new class of validated “aggressive” duct designs and design tools that will allow a much better optimization of the whole compression and turbine system [7].

The pressure distribution in an S-shaped diffuser depends not only on the area change such as in straight-walled channels, but also upon the curvature of the end walls and hence of the streamlines. S-shaped diffuser has a major effect on the boundary layer behavior at the end walls, which is most critical due to the strongest adverse pressure gradients right after strong wall curvatures [7]. This feature makes the numerical simulation and testing of an aggressive S-shaped diffuser difficult. Wellborn et al. [8] observed the intense mixture of the boundary layer and the core flow in the wind tunnel experiment with the inlet Mach number of 0.6. The counter-rotating vortices convected low momentum fluid of the boundary layer toward the center of the duct, and a large region of streamwise flow separation occurred within the duct. A large number of experimental studies have been carried out on the flow characteristics of S-shaped duct [9-13]. About the numerical simulation study, many researchers have done lots of work to find more accurate turbulence models for numerical simulation [14-18]. Waston et al. [18] calculated the NASA 30/30 geometric model using RANS (S-A, k-ε and SST k-ω model) and zonal large eddy simulation (RANS/LES) approaches. It was found that LES could more reliably predict large degrees of separation, but the mesh requirements and consequent cost render the technique prohibitively expensive for the near future, certainly unacceptably so for routine calculations as part of an ongoing design process. Asghar et al.[19-20] explored the flow characteristics of a baseline S-shaped diffuser using experiment and numerical simulation methods. They concluded that the presence of streamwise and circumferential pressure gradients led to the 3-D flow and the flow distortion in the exit. In addition, different geometry parameters of the S-shaped diffuser would have greatly influence on the flow characteristics. In this paper, the internal flow characteristics of an aggressive S-shaped air bleeding manifold of high-pressure turbine with active clearance.
control (HPTACC) are experimentally and numerically investigated under typical working condition with the throttle cone opening degree of 15% and the inlet Mach number of 0.34.

1 EXPERIMENTAL SETUP

1.1 Test Facility

The experimental wind tunnel diagram is shown in Fig.1. The experimental device power source is a three phase alternating current asynchronous motor with rated power of 1500kW, which can drive the high pressure ratio centrifugal blower to pressurize the air, so as to provide high speed and high pressure flow conditions for the experimental section through the import valve, the diffusion section, the stable pressure section and the contraction section. The stable pressure box is an 800mm*800mm square with three damping screens and a honeycomb flow straightening grid. The experimental section is a 100mm *100mm square channel to simulate the flow condition of the engine fan bypass outlet.

Fig.1 Schematic diagram of the wind tunnel

The air bleeding manifold 3-D test model for this experiment is shown in Fig.2. The throttle cone was connected to the end of air outlet pipe. In order to reduce the influence of the throttle cone on the flow field of exit plane, a cylindrical duct with 4.5 times length of the air bleeding manifold was installed between the throttle and the exit of the air bleeding manifold. The experiment facility consists of the inlet flow straightening section, air bleeding manifold and exhaust throttle section. The air bleeding manifold test model was made with the geometry ratio of 1:1 according to the geometric design scheme, and connected to the facility of the wind tunnel through the contraction section 2. The cross-sectional shape of the air bleeding manifold is gradually changed from quasi-arch at the inlet to circle at the exit. The duct curvature changes drastically and the cross-sectional size is small. As an expansion duct, figure 3 shows the cross-sectional area distributions of the air bleeding manifold, where the abscissa axis (streamwise position) x and ordinate axis (cross-sectional area) A are respectively nondimensionalized by the length of the air bleeding manifold (l0) and the exit area (Aexit).

Fig.2 Air bleeding manifold 3D model

Fig.3 Area distribution of the air bleeding

1.2 Instrumentation and Data Acquisition

In this experiment, the exit opening degree of the air bleeding manifold is 15% and the inlet Mach number is 0.34. The experiment is conducted using the pitot tube to acquire the total pressure, the surface static pressure holes to obtain the static pressure and the thermocouple to measure the total temperature at the stable pressure section. Moreover, the calibrated five-hole probes are used to obtain the flow field at the measuring station (only measure the parameters in the meridian plane at Z=0). As shown in Fig.4, the measuring stations are located in the inlet (X=0, measuring station 1) and the exit (X=235mm, measuring station 2) of the air bleeding manifold. 10 measuring points are placed in the inlet and 12 measuring points are placed in the exit, according to Fig.5.
In this experiment, the total pressure recovery coefficient is used to quantitatively analyze the aerodynamic performance of the air bleeding manifold. The inlet Mach number is calculated based on the measuring data of the total pressure and static pressure at the inlet. The formulas for calculation are as follows:

$$\delta^* = \frac{p_{\text{out}}^*}{p_{\text{in}}^*}$$  \hspace{1cm} (1)

$$Ma = \sqrt{\frac{2}{k-1} \left( \frac{p_{\text{in}}^*}{p_{\text{out}}^*} \right)^{\frac{k-1}{k}} - 1}$$  \hspace{1cm} (2)

Where, $p^*$ is the total pressure, $p$ is the static pressure and the subscripts in and out respectively represent the inlet and exit of the air bleeding manifold.

### 2 NUMERICS

In the following, the numerical method, the numerical mesh, and the boundary conditions used are presented. The numerical mesh of the components including air bleeding manifold, extension section and throttle cone was generated by the ICEM module in software ANSYS. The grid is shown in Fig.6, where the structure grid and O-type topology were used to refine the grid in boundary layer and improve the grid orthogonality. The software ANSYS®Fluent 16.1 was used to solve the steady 3-D Reynolds-Averaged Navier-Stokes (RANS) equations. The spatial discretization was performed by the second-order upwind method. The ideal compressible air was taken as the calculation working medium. The boundary conditions are listed in Table 1. To validate the mesh independence, five meshes respectively with 1.3, 1.8, 2.3, 4 and 8 million cells were tested. Finally, mesh with 2.3 million cells reached the mesh independence when analyzing the overall performance and the flow field detail. Therefore, 2.3 million cell mesh was adopted for the following analysis.

In this paper, the air bleeding manifold is an aggressive S-shaped duct with variable geometric cross-section, the flow structure is complicated. Therefore, it is important to select the appropriate turbulence model to ensure the accuracy of the numerical calculation. Four turbulence models widely used in the aeronautical field were selected: Spalart-Allmaras (S-A) model, SST $k-\omega$ model, Realizable $k-\varepsilon$ model and turbulence stress (RSM) model. The results were comparatively analyzed to determine the proper turbulence model. In addition, the Realizable $k-\varepsilon$ model and the turbulence stress (RSM) model used the enhanced wall treatment. The value of $y+$ was [0, 10], which satisfied the computational requirements of the four turbulence models.

As is shown in Table 2, the total pressure recovery coefficient calculated by the experimental result is taken as the comparison with the results of the four turbulence models. It can be seen from the table 2 that the total pressure recovery coefficients in the exit calculated by the four turbulence models are similar, and the error between the computational and experimental results is within the range of 2%. Moreover, the results of RSM and the Realizable $k-\varepsilon$ model are much closer to the experimental results, and the difference between S-A model and experiment is more obvious, due to the conservative prediction of the vortex using S-A model.

### Table 1 Computational boundary condition

<table>
<thead>
<tr>
<th>Boundary parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>total pressure-inlet /Pa</td>
<td>116570</td>
</tr>
<tr>
<td>total temperature-inlet /K</td>
<td>320.84</td>
</tr>
<tr>
<td>velocity direction-inlet axial direction</td>
<td></td>
</tr>
<tr>
<td>static pressure- outlet /Pa</td>
<td>101000</td>
</tr>
</tbody>
</table>

### Table 2 Total pressure recovery coefficient

<table>
<thead>
<tr>
<th></th>
<th>Exp.</th>
<th>S-A</th>
<th>SST</th>
<th>Realizable $k-\varepsilon$</th>
<th>RSM</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta^*$</td>
<td>0.9665</td>
<td>0.9853</td>
<td>0.9821</td>
<td>0.9818</td>
<td>0.9814</td>
</tr>
<tr>
<td>$\Delta %$</td>
<td>1.94</td>
<td>1.61</td>
<td>1.58</td>
<td>1.54</td>
<td></td>
</tr>
</tbody>
</table>
Fig. 7 shows the comparison of the contour of the total pressure recovery coefficient at the exit cross section calculated by the four turbulence models. The figure shows that the influence of the high pressure region near the upper wall calculated by S-A is the largest, and the distribution of the total pressure recovery coefficient calculated by SST \( k-\omega \) and Realizable \( k-\varepsilon \) model is in much better agreement than other turbulence model, while the high pressure region near the upper wall calculated by RSM shows some difference. This is because the RSM avoids the Boussinesq’s eddy viscosity hypothesis and directly solves the turbulence stress term of the Reynolds equation.

**Fig. 7 Contour of total pressure coefficient at the exit for different turbulence models**

Fig. 8 presents a comparative analysis of the streamline on central symmetry section \((Z=0)\) for different turbulence models. The figure shows that the numerical results of turbulence models are close to each other without flow separation, whereas the numerical results are greatly affected by the turbulence model when the flow separation occurs. Four models are all able to capture the separation line S1 and S2 at the first bend near the upper wall, and the separation line S3 at the first bend near the lower wall. However, there are differences in the specific location of the separation line S1 and S2 and the development of the downstream channel in each turbulence model. Moreover, the relative position and the developing trend of separation line S1 and S2 of S-A and RSM turbulence model are similar, which are slightly different from the other two models. The separation line of SST \( k-\omega \) model is much closer to the one of Realizable \( k-\varepsilon \) model, and the separation line S1 at the upper wall near the first bend will yield the back flow region. The results show that the SST \( k-\omega \) model and Realizable \( k-\varepsilon \) model have more similar analysis of streamline development and flow field detail.

**Fig. 8 Streamline on central symmetry plane \((Z=0)\)**

In summary, the total pressure recovery coefficient and flow separation results of SST \( k-\omega \) and Realizable \( k-\varepsilon \) turbulence model are similar. Because the Realizable \( k-\varepsilon \) prediction results is closer to the experimental results than the SST \( k-\omega \), in this paper, we finally chose realizable \( k-\varepsilon \) model to simulate the internal complex flow as well as to study the flow field details of the air bleeding manifold.

### 3 FLOW CHARACTERISTICS ANALYSIS

Based on the CFD numerical results, the flow field details which are difficult to obtain by experimental measurement have been analyzed. Firstly, the overall flow characteristics of the experimental model have been obtained. Then the mechanism of flow loss and the secondary flow in the air bleeding manifold has been investigated.

#### 3.1 Flow Characteristics

Fig. 9 exhibits the contour of Mach number at the symmetry plane. As shown in the figure, near the upper wall, the flow velocity is decelerated, and the boundary layer grows rapidly in the flow channel due to the concave curve at the first bend. In contrast, the flow accelerates and suppresses the development of boundary layer because of the convex curve at the second bend. Near the lower wall, due to the presence of severe deflection at the first bend, a large low-velocity region is formed. The low velocity flow moves towards the center of the flow channel along streamwise direction, and finally stabilized near 60% passage height in exit plane. In the front part of the mainstream, the flow velocity is significantly decelerated because of the expansion of the duct. By contrast, in the after part, the flow velocity remains roughly unchanged because of the development of the low-velocity region near the lower wall.
Fig. 9 Contour of Ma at the symmetry plane (Z=0)

Fig.10 presents the static pressure distribution of the upper and lower wall. According to the figure, the axial length is dimensionless and the range is [0, 1]. The wall static pressure is nondimensionalized as $p/p_\text{in}^*$, where $p$ represents the local static pressure, and $p_\text{in}$ represents the total pressure of the inlet. The static pressure has a maximum value of 0.989 and a minimum value of 0.971 at the first and the second bend near the upper wall, which corresponds to the low velocity and high velocity position near the upper wall in Fig. 9. Near the lower wall, the static pressure along the channel of the first bend is rapidly raised in general. And with the further development of flow separation, the change of static pressure becomes stable.

3.2 Mechanism Analysis

To investigate of the distortion degree of the exit plane, the synthesis distortion index $DC(60)$ is introduced, the formula is shown in equation (3):

$$DC(60) = \frac{P_{\text{out,av}}^* - P_{\text{min}(60,av)}^*}{P_{\text{out,av}}^* - P_{\text{out,av}}^*}$$

Table 3 Axial location of planes along streamwise direction

<table>
<thead>
<tr>
<th>Axial direction/ x/l₀</th>
<th>Plan eA</th>
<th>Plan eB</th>
<th>Plan eC</th>
<th>Plan eD</th>
<th>Plan eE</th>
<th>Plan eF</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00</td>
<td>0.25</td>
<td>0.50</td>
<td>0.70</td>
<td>0.85</td>
<td>1.00</td>
<td></td>
</tr>
</tbody>
</table>

Fig.11 shows the streamline distribution inside air bleeding manifold. It can be seen from the figure that the upper wall and lower wall have severe duct curvature variant and the flow channel is greatly expanding, which can lead to flow separation and complex flow structure corresponding to the secondary flow. On the upper wall, separation lines S1 and S2 are generated, and they flow backward when reaching the lower wall. When S1 reaches the lower wall, it turns back towards the lower wall, and there is possibility to form vortex. While S2 backflows and quickly converges with separation line S3 developing steadily along the axial. On the lower wall, there is separation line S3, which converges with S1 and S2 along streamwise direction. In general, the flow separation region on the lower wall is much larger than that on the upper wall, and the mainstream region flows better than the separation zone.
Fig.12 displays the contour of static pressure at several planes along streamline. As shown in the figure, in plane A, the static pressure is circumferentially uniform and the flow quality is better. In plane B, there is a dramatic change of the curvature on the lower wall profile, where exists the low static pressure zone. Besides, the static pressure near the upper wall is higher than that of the lower wall. Therefore, pressure gradient occurs in the radial direction of the plane, and the flow separation appears on the upper wall surface, which is consistent with the position at the separation line S1. In plane C, the pressure gradient in the radial direction of the plane is further expanded, while a significant adverse pressure gradient occurs at the first bend of the upper wall along the axis. The upper wall flow separation is further developed, which is corresponding to the formation of the separation line S2. In plane D, with the flow at the upper wall accelerating, static pressure significantly attenuated, radial distribution of the pressure changes in gradient direction, which corresponds to the separation lines S1 and S3 merging in the low-velocity region at the bottom, and moving towards mainstream region. In plane E, owning to expansion of the radial influence of the low-velocity region at the bottom, the scale of the low-pressure region on the upper wall is decreased, and the separation zone near the lower wall continues to diffuse and reaches the entire lower wall. In the exit plane F, with the further recovery of the static pressure on the upper wall, the static pressure distribution is more uniform along radial direction, and the flow is steady along the axial direction. In addition, the separation zone and the main flow region are relatively stable.

For further analysis of the flow of vortex formation mechanism, Fig.13 exhibits the corresponding velocity vector distributions at several planes along streamwise direction. In the figure, the Z-axis represents spanwise direction. In addition, to analyze the flow field structure of the airflow along each direction more clearly, some necessary corresponding streamlines are added. As shown in the figure, when the airflow reaches plane B, under the effect of radial pressure gradient, there is a vertical velocity component downward from the upper wall as well as a tendency of symmetrical diffusion on both sides, indicating the exist of crosswise pressure gradient. When the airflow reaches plane C, it initially forms a counter-rotating vortex pair that is symmetrically distributed along the Z-axis. The reason is that the flow velocity component with the direction from the bottom to the top generated by the backflow near the lower wall converges with the velocity component from the top to the downon the lower wall, and then forms a vortex pair under the influence of the transverse gradient. When the airflow reaches plane D, the flow direction trend of velocity vector is similar to that of plane B, which is merged at the bottom of the duct. Meanwhile, the streamwise vortex rapidly expands in the channel, and the influence scale increases significantly. As the flow channel expands, the vortex core moves towards the symmetry plane along spanwise direction (Z-axis). In addition, there is an upward trend of the vortex core along the radial direction, which implies that the streamwise vortices spread to the center of the channel. In plane E and F, the vortex expands rapidly in the channel, and the influence region increases obviously. The vortex core moves towards center along Z axis and away from the symmetry surface. This is due to the mainstream transportation and vortex rotation, the high-momentum fluid of the mainstream is involved and mixed with the low-momentum fluid within the vortex, which makes the effect region of the vortex much larger.

Fig.14 shows the contour of Mach number at several planes along streamwise direction. As shown in the figure, in plane A, the distribution of Mach number along the circumferential direction is more uniform, and the quality of airflow is better. In addition, the boundary layer is not fully developed at this time. In plane B, the flow velocity of the mainflow region decreases due to the increase of flow passage area. Meanwhile, the boundary layer of the upper and lower wall becomes thicker and more uniform along the axial direction. And there is a low velocity zone near the top of the upper and lower walls. In plane C, the boundary layer of the upper wall is thickened further due to the formation of the leeside at the first bend which suppresses the flow velocity. In the vicinity of the lower wall, the pressure is reduced and the low velocity zone is formed near the wall, which is induced by the duct curvature variant and the rapid increase of the flow cross-sectional area. In the low velocity separation region, the low-momentum fluid is further accumulated, and the radial range of the low velocity zone increases along streamwise direction. The flow velocity presents inhomogeneity along the circumferential direction. In addition, there are flow separation, backflow and strong three-dimensional vortex at the region, corresponding to Fig. 13. In plane D and F, the low velocity zone at lower wall is further enlarged, and the main stream is continuously compressed caused by the rotation of the vortex. The high-momentum fluid of the mainstream in the duct is involved and mixed with the low-momentum fluid within the vortex, which intensifies the circumferentially non-uniformity of the flow parameters and increases the flow loss. The Mach number in the exit plane is more uniform below the 74% radial height, and a high velocity flow region exists above the 74% radial height, suggesting that the experimental module performs not well in regulating the uniformity of the main flow field. Moreover, the mixture of the separation zone and the mainstream is the main factor of the flow loss.
To summarize the evolution of the separation lines, the reason of the separation line near the upper wall is that the duct curvature variant leads to the formation of the leeside near the wall, which then induces the decrease of the flow velocity and the increase of the static pressure. Combining the influence of the adverse pressure gradient along the axial, the boundary layer separation is generated at the first bend on the upper wall. The reason for the separation line on the lower wall is that the intense curvature of the lower wall and the severe expansion of the duct lead to generation of the high pressure and low velocity zone, which cause the flow stagnation. As the radial pressure gradient is not sufficient to keep the flow moving along the wall, the flow separation occurs near the wall. To figure out the generation and evolution of vortex, it can be seen that a pair of symmetrical streamwise vortex is formed on both sides of the Z axis near the axial position on the lower wall corresponding to the position of the first bend on the upper wall. The flow path corresponds to the separation line S1 in Fig. 12. With the convergence of separation line S1 and S3, the streamwise vortex moves towards both sides and the center of main channel under the influence of mainstream transportation and the rotation of the vortex itself. The fundamental cause of vortex is closely related to the spatial
interaction of the axial pressure gradient, the radial pressure gradient, and the crosswise pressure gradient.

Fig.15 presents the contour of total pressure recovery coefficient at several planes along streamwise direction. As seen from the figure, the total pressure recovery coefficient of the main area is near 0.99 along streamwise direction, which indicates that the flow characteristics of the mainstream are good and the flow loss is low. In plane B, the boundary layer of the wall is rather thick, and the total pressure recovery coefficient is obviously decreased which is induced by the viscosity loss in the boundary layer. Whereas, the total pressure recovery coefficient is uniform along the circumferential direction, and the overall value is relatively high. In plane C, the total pressure recovery coefficient in a large area near the lower wall is obviously decreased, and the overall total pressure recovery coefficient is decreased. This is because that the drastic change of the curvature of the lower wall promotes the generation of the flow separation and vortex, and the high-momentum loss caused by fluid friction leads to the decrease of the total pressure recovery coefficient. In plane D-F, under the effect of rotating vortex, the surrounding high velocity mainstream flow is involved into the low velocity separation zone, which improves the uniformity of the total pressure recovery coefficient along the circumferential direction. The mass-averaged total pressure recovery coefficient in plane F is 0.982. In summary, through analysis of the contour of the Mach and the total pressure recovery coefficient along streamwise direction, it can be found that the total pressure loss of the air bleeding test module is mainly from the flow separation generated by intense adverse pressure gradient, which is caused by the duct curvature variation at the first bend of the upper wall and the first and second bend of the lower wall, and the expansion of the duct area. The flow loss has a great influence on the mainstream, and leads to the circumferential inhomogeneity of the flow field. The synthesis distortion index DC(60) in the exit plane is 0.1334.

4 CONCLUSIONS

In this paper, the experiment of the HPTACC air bleeding manifold test module is firstly performed to obtain the total pressure recovery coefficient. The experiment is conducted under typical working condition with opening degree of 15% and the inlet Mach number of 0.34. Based on the experimental result of total pressure recovery coefficient, the realizable k-ε model is selected as more suitable turbulence model for the air bleeding manifold numerical simulation by analyzing the numerical results of different turbulence models. The numerical simulation analysis conclusions are as follows:

(1) The combination of the expansion channel and strong duct curvature variation of air bleeding manifold causes the obvious deceleration of the main airflow. And the overall velocity of the second half is roughly unchanged due to the low-velocity region near the lower wall. The strong expansion angle at the first corner of the lower wall yields a large low-velocity region, which diffuses to the center of the flow channel and finally stabilizes near the 60% passage height in the exit plane.

(2) In the air bleeding manifold, there is obvious flow separation and secondary flow at the upper and lower near the first bend which caused by obvious change of curvature, and a pair of counter-rotating vortex structures are also formed. Under the effect of the mainstream transportation and vortex rotation, the vortex continues to spread downstream.

(3) The main factors causing the total pressure loss of the air bleeding manifold test module are boundary layer separation and secondary flow effect, which are induced by the interaction of the intense deflection at the beginning of the first bend of the lower wall and the flow channel expansion. High energy loss caused by fluid friction leads to the decrease of total pressure recovery coefficient. The mass-averaged total pressure recovery coefficient in the exit plane is 0.9818, and the synthesis distortion index DC(60) in the exit plane is 0.1334.
ACKNOWLEDGMENT

This work is supported by the Research Program of Shanghai Municipal Commission of Economy and Information, AECC Commercial Aircraft Engine Co., LTD (No. CXY-2016-005), the United Innovation Center (UIC) of Aerothermal Technologies for Turbomachinery, and “2011 Aero-Engine Collaboration Innovation Plan” of China. The authors express the sincere gratitude to Lu Huawei from Marine Engineering College of Dalian Maritime University for his help in the experiment.

REFERENCES