EFFECTS OF GEOMETRIC PARAMETERS AND INFLOW CONDITIONS ON FLOW IN AGGRESSIVE INTER-TURBINE DUCTS

Yumo Mu
National Key Laboratory of
Science and Technology on
Aero-Engine Aero-
thermodynamics, School of
Energy & Power
Engineering, Beihang
University
Beijing, P. R. China

***Weihao Zhang**
National Key Laboratory of
Science and Technology on Aero-
Engine Aero-thermodynamics,
School of Energy & Power
Engineering, Beihang University
Beijing, P. R. China
Collaborative Innovation Centre of
Advanced Aero-Engine
Beijing, P. R. China

Yanmei Mei
National Key Laboratory of
Science and Technology on
Aero-Engine Aero-
thermodynamics, School of
Energy & Power
Engineering, Beihang
University
Beijing, P. R. China

*Corresponding author, Email: zhangweihao@buaa.edu.cn

ABSTRACT

Demands for high efficient and environmental friendly aircrafts drive civil engines towards higher bypass ratios and smaller axial length. Both of these lead to great increase in the mean rise angle of inter-turbine ducts (ITDs), which determines the high curvature and high diffusion rate of its internal flow fields. In order to reduce the flow loss of aggressive ITDs, it is necessary to study the complex flow mechanism in ITDs, explore the factors influencing the flow and performance in ITDs, and provide technical supports to develop advanced turbine design technologies for high-bypass turbofan engines.

In this paper, numerical simulations of a typical aggressive ITD with struts between high and low pressure turbines of a large civil engine are carried out, and a method for comprehensively evaluating the aerodynamic performance of ITDs considering compressibility, area ratio, outlet flow angle is proposed. With this method, the influence of geometric parameters on ITDs flow structure and performance is discussed. The study shows that the area ratio has important influence on the flow field inside the ITDs, and the optimization of area distribution can improve the aerodynamic performance of ITDs. Finally, the influence of inlet aerodynamic conditions such as inlet Mach number and turbulence intensity on flow structure and performance of ITDs is analysed.

INTRODUCTION

The inter-turbine duct (ITD) is often used to connect HP (high-pressure) and LP (low-pressure) turbines in two-spool turbofan engines. In order to shorten axial length to reduce weight and increase bypass ratio to pursue efficiency, more aggressive ITDs with larger mean rise angles are in burning demand. Owing to the geometry of high curvature and large

mean rise angle, the flow has a risk of significant separation which limiting the aerodynamic performance of ITDs. Thus, it is important to study the complex flow mechanism in ITDs.

Work on traditional ITDs can trace back to (Sovran and Klomp, 1967), which proposed the reference sample table S&K diagram with great engineering value for the initial design of ITDs. They carried out detailed experimental research on three types of linear diffusers, and uncovered the relationship between the inlet-to-outlet area ratio and length to height ratio in order to maximum the static pressure lift. However, (Couey et al., 2010) pointed out that the design scope of ITD today is quite different from the experimental data of (Sovran and Klomp, 1967), and supplemented the influence of mean rise angle on aggressive ITD performance.

Extensive researches have been made to reveal the physical mechanism of flow in ITD. The effects of mean rise angle, area rate, structural struts, upstream swirls and wakes have been particularly discussed. (Dominy et al., 1995, 1996, 1998) studied the influence of upstream swirl and wake on flow in ITD channel through experiment and numerical simulation. The results show that the upstream swirl and wake can significantly change the flow structure and the development of secondary flow in ITD channel, although they have little effect on the static pressure coefficient. (Hu et al., 2011; Zhang et al., 2011) studied the influence of inlet swirl distribution on the flow development, finding that the flow loss distribution inside ITD is closely related to the counter-rotating vortex pairs in both shroud and hub regions. In recent research, (Zhang et al., 2018) studied the effects of mean rise angle and area ratio on ITD aerodynamic performance by testing four ITD configurations by varying outlet-to-inlet area ratios and mean rise angles. They found that the adverse pressure gradient near the first bend increased with the increasing of mean rise angle, causing a

strong separation near shroud. When the area ratio increased, not only the shroud separation was maintained, but a significant 2D separation was developed as well.

In this paper, an ITD performance evaluation method considering compressibility, area ratio and outlet flow angle is given. With this method, the improvement of ITD performance caused by area rate optimization is discussed, and the influence of inlet Mach number and turbulence intensity on ITD performance is analysed.

METHODOLOGY

Numerical Methods

The research object of this paper is an aggressive ITD between high and low pressure turbines of a large civil engine with 12 struts, as shown in Fig.1. The main geometry parameters of the ITD are listed in Table 1. In comparison, the mean rise angle of this ITD is 40° while the mean rise angle of ITD in GE90 is only 35°. In that case, the research on such an aggressive ITD is representative, and the results obtained are valuable for reference.

The numerical simulation uses the ANSYS CFX software to solve steady RANS equations with a time pursuing finite volume method. The Shear Stress Transport (SST) turbulence model is employed for turbulence closure. The SST model has been verified by inhouse experimental data, which demonstrates that the SST model can accurately predict the aerodynamic performance and flow details. Total temperature, total pressure and inflow angle are used as the inlet boundary conditions, and static pressure as the outlet boundary condition. Surfaces at the circumferential side of the channels are set as periodic boundary conditions. Due to the strong secondary flows in the aggressive ITD, the vanes of the first stage low-pressure turbine are calculated, in order to achieve better convergence of the calculation results. The interface between the ITD and the vanes adopts the mixing surface method for data transmission.

The domain is modelled with structured, hexahedral grids, generated by Autogrid5 software. The H-type grid topology is used in the main channel, and the O-type grid topology is used near the blade wall. The y^+ of the first layer near the blade wall is less than 2, which meet the requirements of the turbulence model. Grid independency analysis is performed by comparing the inlet mass and the total pressure recovery coefficient of the ITD corresponding to five different sets of grids with the same topology. The five different grid sets are shown in Table 2. The grid set that meet the requirements of grid dependency is Grid 02. The total number of Grid 02 in total is about 1.2 million, and the total number of its ITD grids is about 0.8 million.

ITD Performance Evaluation Method

Due to geometric structure characteristics, the flow through ITD is a diffusion and deceleration process. Under the adverse pressure gradient, the boundary layer near the surface of the hub and shroud may evolve, and then separate, leading to large flow loss. Thus, it is necessary to evaluate

Figure 1 Physical Model of the Aggressive ITD

Table 1 Geometry Parameters of the ITD

Area ratio(AR)	1.45
Length-to-height ratio (L_x / h_1)	2.75
Mean rise angle (θ)	40°

Table 2 Different Grid Sets in the ITD (Axial × Span × Circumferential)

GRID 01	GRID 02	GRID 03	GRID 04	GRID 05
171×41× 65	176×49 ×65	199×57 ×73	235×65 ×89	283×73 ×97

the aerodynamic performance of ITD, and to describe the pressure diffusion rate and the loss of flow through ITD. In addition, since the ITD is connected to a low-pressure turbine, the uniformity of flow parameters at the ITD outlet has a great influence on the low-pressure turbine flow field. Therefore, parameters representing the uniformity of outlet flow are often used to evaluate the performance of the ITD.

The static pressure coefficient is used to describe the pressure diffusion of the flow through ITD, representing the static pressure lift from the inlet section to the outlet section, as shown in Eq. (1). Assuming that the flow through the ITD is an isentropic process, the ideal static pressure coefficient can be obtained by combining one dimensional continuity equation and Bernoulli equation, as shown in Eq. (2). In case of incompressible flow, the ideal static pressure coefficient is a single-value function of the area, representing the maximum diffusing rate of flow through ITD. In practice, the flow through ITD is not an isentropic process, so the actual static pressure coefficient is smaller than the ideal coefficient. The ratio between the actual and ideal static pressure coefficient is defined as diffusion efficiency, as shown in Eq. (3).

$$Cp_{incompressible} = \frac{P_2 - P_1}{P_1^* - P_1} \quad (1)$$

$$Cp_{ideal} = 1 - \left(\frac{A_1}{A_2} \right)^2 \quad (2)$$

$$\varepsilon = \frac{C_p}{C_{p_{ideal}}} \quad (3)$$

With Mach number increasing, the airflow density changes, and the larger the Mach number, the stronger the compressibility of the airflow. It is generally considered that when the Mach number is less than 0.3, the airflow is incompressible. As the inlet Mach number of ITD is generally between 0.3 and 0.6, it is not so accurate to predict the maximum pressure diffusing rate of the flow through ITD based on the incompressible assumption. For compressible flow, the static pressure coefficient can be expressed as a function of Mach number and total pressure, as shown in Eq. (4). For isentropic flow, the total pressure at the inlet and outlet remains unchanged, and the one-dimensional flow equation can be simplified as the relationship between area ratio and Mach number, as shown in Eq.(5).

$$C_{p_{compressible}} = \frac{\sigma^* \pi(Ma_2) - \pi(Ma_1)}{1 - \pi(Ma_1)} \quad (4)$$

$$A_1 q(Ma_1) = A_2 q(Ma_2) \quad (5)$$

In addition, the influence of flow direction is neglected in the derivation of the ideal static pressure coefficient. That is, the direction of airflow is considered to be orthogonal to the inlet and outlet section. Thus, the inlet to outlet area ratio refers to the flow area, and the ideal static pressure coefficient is a function of the flow area ratio. The curvature of the shroud and hub lines varies greatly, and a large outlet angle (Φ) often exists between the flow direction and the orthogonal direction of the outlet, which makes the solving complicated with the definition of flow area. Therefore, sometimes it is more convenient to substituting the cross-sectional area for flow area. Fig. 2 (up) shows the relationship between ideal static pressure coefficient, flow area ratio (AR) and inlet Mach number. Fig. 2 (down) shows the relationship between the ideal static pressure coefficient and the cross-sectional area ratio (AR'), inlet Mach number and outlet angle. Generally, the smaller the area ratio is, the more important it is to consider the influence of inlet Mach number and outlet flow angle.

Table 3 shows the ideal static pressure coefficients with and without considering compressibility when the area ratio is 1.45 under different inlet Mach numbers. It can be seen from the table that when the inlet Mach number is 0.6, ignoring the compressibility of the air flow will lead to a deviation of 24.43% in the ideal static pressure coefficient. Therefore, a big deviation in predicting the maximum diffusing pressure can be achieved theoretically when the gas flows through the ITD according to the incompressible assumption.

The flow loss through ITD can also be estimated by total pressure recovery coefficient, as shown in Eq. (6). The following discussion involves the variation of inlet flow conditions such as inlet Mach number and turbulence intensity, which have great influence on flow in ITD. Therefore, the total pressure recovery coefficient based on

Figure 2 Relationship between Ideal Static Pressure Coefficient, Area Ratio, Inlet Mach number and Outlet Angle

Table 3 AR=1.45 Static Pressure Coefficient at Different Inlet Mach number

	Ma=0.3	Ma=0.4	Ma=0.5	Ma=0.6
$C_{p_{ideal,incompressible}}$	0.524	0.524	0.524	0.524
$C_{p_{ideal,compressible}}$	0.562	0.589	0.619	0.652
$\Delta C_{p_{ideal}}$	7.25%	12.40%	18.13%	24.43%

the inlet kinetic energy cannot reasonably describe the development of flow loss along the path, so it can be replaced by total pressure loss coefficient, as shown in Eq. (7). Since the essence of total pressure is the high-quality energy of the flow, the total pressure recovery coefficient can be used to represent the work capacity loss after flowing through ITD. Another expression of the flow loss through ITD is cascade efficiency, which is expressed in the form of total enthalpy ratio, as shown in Eq. (8). Compared with the total pressure recovery coefficient, the cascade efficiency takes into account the influence of airflow velocity in the ITD, and to some extent represents the kinetic energy loss of the flow through ITD, so it can measure the flow loss in the ITD more accurately.

$$\sigma = \frac{P_2^*}{P_1^*} \quad (6)$$

$$\sigma' = \frac{P_1^* - P_2^*}{P_1^*} \quad (7)$$

$$\eta = \frac{h_2^* - h_2}{h_2^* - h_{2ad}} = \frac{T^* - T_2}{T^* - T_{2ad}} = \frac{v_2^2}{v_{2ad}^2}. \quad (8)$$

RESULTS AND DISCUSSION

Effect of Area Rate

The geometry is one of the main factors influencing the performance of ITD. The previous inhouse research (Mei et al.) found that the curvature of meridional plane have an impact on the flow loss of ITD. In fact, the meridional plane modification design not only changes the local curvature of hub and shroud, but also affects the area rate along the flow direction of ITD channel. These two factors can have great influence on pressure distribution and flow loss of ITD. Next, the influence of area rate on ITD performance and flow is discussed in detail.

The ITD studied in this paper contains 12 struts. As the rate of strut thickness to chord length is large, the decrease of flow area caused by struts cannot be ignored. Fig. 3 shows the modification of meridional plane and geometric design of struts. Based on the original geometry Case_00 and curvature modification geometry of the meridional plane Case_01 (Mei et al, 2017), the modification Case_02 is added. The shroud profile design of Case_02 retains that of Case_01, but the strut thickness distribution and hub profile are adjusted to optimize the distribution of circulation area. Fig. 4 shows the axial distribution of the area rate of each case. Case_02 modify the geometry of the struts, and the position of the maximum thickness of the struts is marked.

Table 4 shows the aerodynamic performance parameters of all geometry. By comparing case_02 with case_01 and case_00, it can be seen that the overall performance of case_02 geometry has been significantly improved, which has smaller flow loss, higher diffuser efficiency, more uniform airflow at the outlet surface. As a result, area rate has a great impact on the performance of ITD.

Fig.5&6 shows the surface limiting streamline and surface static pressure distribution on shroud, hub and strut of all geometry. From the distribution of limiting streamlines near shroud and hub, it can be seen that the modification design has little effect on flow near hub, but flow separation at the pressure side near shroud was suppressed, and the influence area decreased obviously downstream of the separation at the trailing edge of the strut. Fig.6 shows that after Case_02 modification, the corner separation at the trailing edge of the suction side is significantly weakened, the separation area at the pressure side of the strut is reduced, and the radial flow is improved. The shroud curve of Case_01 and Case_02 are the same, and the flows on shroud are similar. However, due to different area rates, the development of secondary flow structures on shroud surfaces of Case_01 and Case_02 are quite different.

Fig.7 shows the distribution of total pressure and Mach number along the flow direction. It can be seen from the figure of Case_02 that the total area with low total pressure and Mach number is significantly decreased. Compared with

Figure 3 Modification of Meridional Profile (left) and Geometric Design of Struts (right)

Figure 4 Area Rate Axial Distribution of each Geometry

Table 4 Aerodynamic Performance Parameters of Different ITD Geometry

ITD	σ	η	C_p	ε	Outlet SKE
Case_00	0.9927	0.9394	0.387	0.642	0.126
Case_01	0.9936	0.9568	0.381	0.632	0.134
Case_02	0.9944	0.9787	0.429	0.711	0.081

the distribution of Case_01, downstream of the separation point on pressure side of the struts, the loss of Case_02 increases more slowly, and the area rate has a great influence on the development of the flow structure in the channel.

Based on above analysis, it can be concluded that the area rate has a great influence on the development of secondary flow structure in aggressive ITD, which further affects the flow loss distribution. Fig.8 shows the mean static pressure axial variation in ITD passage and on meridional end wall profile. It can be concluded that the variation of pressure is consistent with the change of area rate. The inlet sections of Case_01 and Case_02 have larger diffuser pressure than Case_00, of which Case_01 has the longest diffuser part and the largest reverse pressure gradient after the trailing edge of the strut. Although the distribution of endwall pressure on the meridional plane is mainly affected by local curvature, the influence of area rate cannot be ignored. It can be seen from figure of the pressure variation on meridional end wall profile that, though under the sameshroud profile, the pressure variation of Case_01 and Case_02 differ greatly. The pressure variation curve on the surface of Case_02 is smoother and the local adverse pressure gradient is smaller. The surface pressure variation

Figure 5 Shroud (up) and Hub (down) Surface Limiting Streamline

Figure 6 Suction Side (up) and Pressure Side (down) of Strut Surface Limiting Streamline

Figure 7 Total Pressure (up) and Mach number (down) Distribution along Flow Direction

difference of the hub is mainly reflected near the second bend and the adverse pressure gradient on the hub surface of all geometry downstream of the second bend are large.

Fig. 9 shows the variation of secondary kinetic energy (SKE) and flow loss in ITD. It can be seen that flow separation appears at the leading edge of Case_00 struts, the SKE at the

inlet section is large, and the development of secondary flow energy of Case_01 and Case_02 is relatively slow. After the max-thickness of the strut, the secondary flow of Case_00 decrease slightly, but it is still significantly greater than that of Case_01 and Case_02. Comparing the variation of Case_01 and Case_02 secondary flow, it can be seen that the secondary flow corresponding to Case_02 can be slightly larger than Case_01. In fact, from Fig.8 the average static pressure variation in ITD, it can be seen that after the max-thickness of the strut, the positive pressure gradient of Case_01 is the largest, therefore the development of secondary flow is the slowest. After the trailing edge of the strut, each geometry has a large adverse pressure gradient, while Case_01 has the longest diffuser and Case_02 has the shortest. Therefore, at the outlet of the ITD, the secondary flow energy of Case_01 is the largest, and the secondary flow energy of Case_02 is the smallest. It can be seen that the variation of static average pressure in the passage has a great influence on the development of the secondary flow, and the secondary flow loss and the mixing dissipation of the secondary flow and the mainstream constitute most of flow loss in the aggressive ITD. From Fig. 9 it can be seen from the axial development of flow loss in ITD that the flow loss develops fastest in Case_00 and the loss is relatively small in Case_01 and Case_02. According to the flow field analysis, this is mainly because the ITD of Case_01 and Case_02 weaken the separation at the leading edge of the strut. The geometry difference between Case_01 and Case_02 is mainly reflected after the max-thickness of the strut. According to the flow field analysis above, after the max-thickness, separated flows start to appear in the itds of Case_01 and Case_02. From the max-thickness of the to the trailing edge of the strut, the area rate of both Case_01 and Case_02 decreases, then increases rapidly after the trailing edge. The increasing gradient of area rate after trailing edge of strut is almost the same, and the variation of flow loss is also similar.

Figure 8 Static Pressure Coefficient in ITD Channel (left) and on Shroud and Hub Surface (right)

Figure 9 Secondary Kinetic Energy (left) and Total Pressure Recovery Coefficient (right) in ITD Channel

Effect of Inlet Conditions

Besides the geometry factors mentioned above, inlet conditions can also greatly affect ITD performance. The incoming flow conditions contain many aspects, and their influence on flow in ITD is also different. In this chapter, based on Case_02 geometry, the effect of various inlet aerodynamic conditions on ITD performance and flow field is further discussed.

Effect of Inlet Mach number

With the variation of flight conditions, the inlet Mach number of aggressive ITD can change significantly. The ITD is located behind the high-pressure turbine, and generally the inlet Mach number of ITD can vary between 0.3 and 0.6. Thus, it is necessary to evaluate the influence of inlet Mach number on the performance and flow field.

Generally, with the increase of inlet Mach number, the total pressure recovery coefficient and cascade efficiency decrease, and the flow loss increases. At the same time, with inlet Mach number increasing, the static pressure coefficient increase, and the diffusing ability of aggressive ITD get better. From the above discussion, it is pointed out that the ideal static pressure coefficient is a function of inlet to outlet area ratio and inlet Mach number, and ignoring the compressibility of the flow will lead to a large deviation in the evaluation of the ideal static pressure coefficient.

Table 5 shows the performance parameters of aggressive ITD under inlet Mach number varies from 0.35 to 0.55. It can be concluded that with the increase of inlet Mach number, the compressible diffusion efficiency decreases, while incompressible diffusion efficiency increases. Thus, when comparing ITD performance of different inlet Mach number, neglecting compressibility may lead to relatively big error. In fact, from the perspective of outlet flow quality, with the

Table 5 Aerodynamic Performance Parameters of Different Inlet Mach number in Case_02 (ε_c stands for compressible, ε_i stands for incompressible)

Ma_i	σ	η	C_p	ε_c	ε_i	Outlet SKE
0.35	0.9964	0.9800	0.415	0.723	0.792	0.080
0.45	0.9944	0.9787	0.429	0.711	0.819	0.081
0.55	0.9924	0.9755	0.447	0.704	0.853	0.084

inlet Mach number increasing, the outlet secondary flow energy increase, and outlet flow uniformity become worse.

Fig.10 shows total pressure and Mach number distribution along the flow direction at different inlet Mach number. It can be seen with the increasing of inlet Mach number, the location and influence range of the low total pressure region remain unchanged, but the total pressure difference to main flow get larger. This is the same with Mach number distribution, when inlet Mach number increases, the location and influence range of the low Mach number region remain unchanged, but the Mach number difference to main flow also get larger.

Fig.11 shows the average static pressure coefficient in ITD channel and on the surface of the shroud and hub along the axial direction with different inlet Mach number. It can be seen from the variation of average static pressure coefficient of different sections in ITD channel that with the increase of the inlet Mach number, the ability to diffuse of the ITD improves, and the adverse pressure gradient in flow direction increases. Moreover, the larger the inlet Mach number is, the more rapidly the static pressure coefficient increases. It can be seen from figure of the static pressure coefficient on shroud and hub, with inlet Mach number increasing, the static pressure coefficient increase correspondingly. The static pressure coefficient on the surface of the shroud were more affected by inlet Mach number variation, but generally, the pressure gradient did not change much. In conclusion, the change of inlet Mach number within a certain range has some influence on the flow parameters at the outlet, but the flow structure in ITD is basically unaffected because the pressure gradient changes little.

Fig.12 shows the flow loss and secondary kinetic energy in ITD channel under different inlet Mach number. It can be seen that with inlet Mach number increasing, flow loss in ITD increase correspondingly. As discussed above, the flow parameters difference between the area with greater loss corresponding to low total pressure and low Mach number regions and the main flow increase with the increase of inlet Mach number. Therefore, the increase of inlet Mach number mainly affects the friction loss inside the boundary layer, separation flow and other low-energy fluids, as well as the mixing loss between low-energy fluids and the mainstream. The large velocity difference between flows will enhance the shear, resulting in increased flow loss. As can be seen from the SKE diagram, the change of inlet Mach number within a certain range has little influence on the secondary flow structure in ITD. However, when inlet Mach number increase

Figure 10 Total Pressure (up) and Mach number (down) Distribution along Flow Direction

Figure 11 Static Pressure Coefficient in ITD Channel (left) and on Shroud and Hub Surface (right)

Figure 12 Secondary Kinetic Energy (left) and Total Pressure Loss Coefficient (right) in ITD Channel
to 0.55, the secondary flow structure in ITD channel is enhanced after the trailing edge of the strut, leading to the increase of SKE in the high-loss region.

Effect of Inlet Turbulence Intensity

As ITD is located downstream of the high-pressure turbine, the inlet condition is complex. The inlet turbulence intensity can change over time, thus influencing the development of boundary layer and the mixing between flows, and then affecting the flow field and performance of ITD. In this section, three levels of inlet turbulence intensity (1%-5%-10% corresponding to low-medium-high turbulence intensity) are selected, in order to discuss the effect of inlet turbulence intensity to ITD performance.

Table 5 shows the aerodynamic performance parameters of ITD under different inlet turbulence intensity. With inlet turbulence intensity increasing, the total pressure recovery coefficient and cascade efficiency decrease, and the flow loss in ITD increase. Meanwhile, the static pressure coefficient increases with the increase of turbulence intensity of the coming flow, and the diffusing ability of aggressive ITD is

Table 5 Aerodynamic Performance Parameters of Different Inlet Turbulence Intensity (ε_c stands for compressible)

Case_02	σ	η	C_p	ε_c	Outlet SKE
Tu=1%	0.9946	0.9794	0.429	0.711	0.082
Tu=5%	0.9944	0.9787	0.429	0.711	0.081
Tu=10%	0.9942	0.9754	0.431	0.715	0.071

improved. At the same time, with the increase of inlet turbulence intensity, the secondary flow at the outlet can be reduced significantly, and the outlet flow gets more uniform. However, in general the performance of ITD does not change much with the turbulence intensity. When inlet turbulence intensity varies from 1% to 5%, the performance of ITD changes little. When turbulence intensity is as high as 10%, comparing with the middle turbulence level of 5%, the cascade efficiency decreases by only 0.33%, the diffusion efficiency increases by 0.4%, but the static pressure coefficient almost remains the same. In contrast, the SKE at the outlet surface decreases greatly by 12.3%.

Fig.13 shows the distribution of the average static pressure coefficient in the passage and the static pressure coefficient on shroud and hub under different inlet turbulence intensity. It can be seen from the figure that the variation of flow turbulence has little effect on the pressure distribution in ITD channel. The flow loss in aggressive ITD mainly comes from the separation flow near shroud, and the variation of inlet turbulence has little influence on the development of the boundary layer on shroud surface. As the pressure gradient is basically unchanged, the influence of incoming turbulence on the anti-separation ability of the boundary layer is limited.

Fig.14 shows the flow loss in ITD channel and the variation of secondary flow energy along the axial direction under different inlet turbulence intensity. It can be seen from the figure that when turbulence intensity is relatively low like 1% and 5%, the flow loss and secondary flow structure basically remain unchanged with the increase of turbulence of incoming flow. However, when inlet turbulence increases to 10%, flow loss in ITD channel increases obviously, and the SKE after the trailing edge of the strut decreases. As a result, when the incoming turbulence level increases to high enough, the mixing between flows and the total flow loss increases, and the outlet flow gets more uniform.

Figure 13 Static Pressure Coefficient in ITD Channel (left) and on Shroud and Hub (right)

Figure 14 Secondary Kinetic Energy (left) and Total Pressure Loss Coefficient (right) in ITD Channel

CONCLUSIONS

In this paper, the effects of geometric factors and inflow aerodynamic conditions on flow field and performance of ITD are studied. Meanwhile, methods for evaluating the different aspects of aggressive ITD performance are discussed.

Firstly, the effects of flow compressibility, area ratio and outlet angle on the parameters assessing the performance of ITD are discussed. For typical working conditions, ignoring the compressibility can bring a 15% deviation of static pressure coefficient, while ignoring the effect of 20-degree outlet angle may lead to 10% deviation. Secondly, after optimization on the area rate of ITD channel, the outlet flow uniformity and the aerodynamic performance is effectively improved. After area rate optimization, the static pressure coefficient is improved by 12.6%, and outlet SKE is reduced by 39.5% comparing to previous geometry. Thirdly, the effects of inlet conditions on flow field in ITD channel are discussed. When the inlet Mach number varies from 0.35 to 0.55, the diffusion efficiency is reduced by about 1.9%, while outlet SKE increases by about 5%. When the inlet turbulence intensity is relatively low (1% and 5%), its variation has little impact on the flow structure and pressure distribution, however when the incoming turbulence level increases to high enough (10%), the mixing loss will increase, and the outlet SKE will decrease by about 12.3%.

ACKNOWLEDGEMENT

The authors would like to acknowledge the support of the National Science Foundation of China (No. 51406003).

NOMENCLATURE

AR	flow area ratio
AR'	cross-sectional area ratio
Cp	static pressure coefficient
h1	inlet height
ITD	inter-turbine duct
LE	leading edge
Lx	duct axial length
p	pressure
p*	total pressure
SKE	secondary kinetic energy
T	temperature
T*	total temperature
TE	trailing edge
Tu	turbulence intensity
v	flow velocity

ε_c	compressible diffusion efficiency
ε_i	incompressible diffusion efficiency
Φ	outlet flow angle
η	cascade efficiency
θ	mean rise angle
σ	total pressure recovery coefficient
1	inlet
2	outlet

REFERENCES

- Couey P. T., McKeever C. W., Malak M. F. et al. (2010). Computational study of geometric parameter influence on aggressive inter-turbine duct performance. ASME Paper GT2010-23604. pp.1647-1656. doi: 10.1115/GT2010-23604.
- Dominy R. G., Kirkham D. A. (1995). The influence of swirl on the performance of inter-turbine diffusers. VDI Berichte Nr. 1186, pp.107-122.
- Dominy R. G., Kirkham D. A. (1996). The influence of blade wakes on the performance of inter-turbine diffusers. ASME Journal of Turbomachinery 118(2), pp.347-352. doi:10.1115/1.2836649.
- Dominy R. G., Kirkham D. A., and Smith A. D. (1998). Flow Development through Inter-Turbine Diffusers. ASME Journal of Turbomachinery 120(2), pp. 298-304. doi:10.1115/1.2841406.
- Hu S., Zhang Y., Zhang X., Vlasic E. (2011). Influences of Inlet Swirl Distributions on an Inter-Turbine Duct Part II: Hub Swirl Variation. ASME Paper GT2011-45554. pp.631-641. doi: 10.1115/GT2011-45554.
- Mei Y., Zhang W., Liu H., Zou Z. (2019). Flow Mechanisms and Optimization Design Within a Super-Aggressive Intermediate Turbine Diffuser, Journal of Engineering Thermophysics 40(1), pp. 1-11.
- Sovran G., Klomp E. D. (1967). Experimentally determined optimum geometries for rectilinear diffusers with rectangular conical or annular cross section. Fluid mechanics of internal flow, pp.270-319.
- Zhang Y., Hu S., Zhang X., Vlasic E. (2011). Influences of Inlet Swirl Distributions on an Inter-Turbine Duct Part I: Casing Swirl Variation. ASME Paper GT2011-45555. pp.619-630. doi: 10.1115/GT2011-45555.
- Zhang Y., Hu S., Mahallati A., Zhang X., Vlasic E. Effects of Area Ratio and Mean Rise Angle on the Aerodynamics of Interturbine Ducts. (2018). ASME Journal of Turbomachinery 140(9), pp.091006.1-11. doi: 10.1115/1.4039936.