Effects of Rising Angle on Upstream Blades and Intermediate Turbine Duct (Postprint)

Authors: Liu, J.„ Wang, P.„ Du, Q„ Liu, G., Zhu, J. Q.

Date: 2017-11-02T00:00:00+00:00

Abstract

This paper describes the numerical study on film cooling effectiveness and aerodynamic loss due to coolant and main stream mixing for a turbine guide vane. The effects of blowing ratio, mainstream Mach number, surface curvature on the cooling effectiveness and mixing loss were studied and discussed. The numerical results show that the distributions of film cooling effectiveness on the suction surface and pressure surface at the same blowing ratio (BR) are different due to local surface curvature and pressure gradient. The aerodynamic loss features for film holes on the pressure surface are also different from film holes on the suction surface.

Full Text

Preamble

With advancing requirements, design methodologies, and manufacturing technologies, future aero-engines are characterized by further reductions in fuel consumption and cost alongside increased propulsion efficiency, leading to ultra-high bypass ratio configurations. The intermediate turbine duct (ITD), which connects the high-pressure turbine (HPT) with the low-pressure turbine (LPT), critically impacts the overall performance of such engines. Consequently, mastering the design techniques for aggressive and even super-aggressive ITDs has become increasingly urgent.

Over recent years, numerous research efforts have investigated flow mechanisms in diffuser ducts, yielding many valuable results, though further investigation remains necessary. Using numerical methods, this paper examines how the rising angle (RA) affects ITD performance and flow fields, as well as the adjacent turbines. Eight ITD configurations with identical area ratios and lengths but varying RAs from 8 degrees to 45 degrees are compared.

The investigation reveals that the flow field, particularly the outlet Mach number of swirl blades, is influenced by RA through potential effects—a finding that advises designers to modify HPT rotor blades after altering the ITD. Additionally, as RA increases, the low-velocity region migrates upstream toward the first bend, while pressure loss distribution on the S2 stream surface shows that the hub boundary layer is more sensitive to RA, whereas the casing layer remains nearly constant.

On the other hand, the overall total pressure loss remains nearly equivalent across different RA cases, underscoring the importance of optimization.

Keywords: Intermediate Turbine Duct, Swirl Blades, Rising Angle, Pressure Loss

Introduction

Efficiency and specific fuel consumption (SFC) are two critical indicators of engine performance. To enhance performance and reduce cost, higher efficiency and lower SFC have been pursued relentlessly. To meet these requirements, larger bypass ratio engines have received increasing attention due to their advantages and potential. However, as bypass ratio increases while the power required by the fan remains constant, the radius of LPT blades tends to become larger, increasing the radius difference between HPT and LPT to some extent. This radius difference introduces design concerns, making the ITD—which not only connects HPT and LPT but also diffuses airflow to reduce Mach number with minimal pressure loss—a component that warrants essential and detailed investigation.

ITD performance can be influenced by many factors, such as area ratio (AR), non-dimensional length (ratio of ITD length to inlet height), and others. Among these, upstream flow condition is a particularly significant factor. Dominy et al. [1] investigated the influence of swirl on flow through a diffusing duct. Using the same experimental facility, Dominy et al. [2] studied the influence of upstream blade wakes on duct performance. To the authors' knowledge, this work represents the first experiment addressing the pseudo-steady influence of two-dimensional wakes extending from hub to casing on ITDs. More recently, Hu et al. [3] and Zhang et al. [4] systematically researched the influence of casing swirl and hub swirl distributions, respectively, on ITD flow fields using experimental and numerical methods. Miller et al. [5] published work on the migration and dissipation of flow phenomena in a swan-necked diffuser downstream of a transonic high-pressure turbine stage. The following year, Miller et al. [6] extended knowledge in this area by investigating interaction mechanisms between a high-pressure rotor and a downstream vane located in a diffuser. The influence of upstream HPT rotor gap size variation on ITD flow field is discussed by Marn et al. [7,10], Göttlich et al. [8], and Sanz et al. [9]. Additionally, Axelsson et al. [11] revealed the complex flow structure development within ITDs.

Göttlich et al. [12] and Pullan et al. [13] have studied the flow field within

ITDs. Johansson et al. [14] completed work focusing on a joint experimental and numerical investigation of an intermediate duct configuration. Marn et al. [15] compared flow through two different S-shaped intermediate turbine duct configurations with different diffusion ratios. Hu et al. [16] optimized a baseline ITD geometry using Numeca software to obtain a second ITD with 25% larger mean RA and a third ITD with 20% larger area ratio.

Göttlich [17] summarized previous work on ITDs, reviewing flow evolution through intermediate turbine diffusers and systematically discussing the influence of different effects. This work provided a meaningful foundation for subsequent investigations, offering a good starting point and correct direction for ITD development. In addition, other useful investigations [18]-[26] on ITDs have been completed by researchers.

Upstream flow condition is an important and apparent factor for ITD performance that can be realized through experimental facilities and has received much study, as introduced above. Another factor, equally vital and attracting much attention but with less investigation, is the duct geometry itself.

Norris and Dominy [27] used both experimental and numerical results to show differences between two S-shaped ducts with the same area ratio and radial offset but different axial lengths. Couey et al. [28] developed an original annular-diffuser performance chart for ITDs, which has been widely used for diffuser-type duct design since Sovran and Klomp [29] reported it. Using numerical methods, Couey et al. assessed the interaction of duct slope and pitch-wise turning with area ratio and length using approximately sixty different duct configurations. Their investigation concluded that the influence of slope and exit pitch angle can be as significant as the area ratio and length parameters traditionally used to correlate duct performance. A combination of new area ratio-length and slope-length parameters was found to segregate ducts between separated and non-separated cases, leading to a modification of the Sovran and Klomp chart.

A more detailed investigation of the effects of mean RA and area ratio on ITD aerodynamics was conducted by Zhang et al. [30] in a large-scale, low-speed, annular wind-tunnel facility at the National Research Council of Canada. Their study found that the duct mean rise angle determined the severity of adverse pressure gradient in the casing’ s first bend, whereas the duct area ratio mainly governed the static pressure rise in the second bend.

In the above studies, Norris and Dominy examined the flow field of two ducts with the same area ratio and radial offset but different axial lengths. A more detailed work was done by Couey et al., who simulated nearly sixty models to find the influence of duct slope and pitch-wise turning parameters on ITD. They focused on the relationship between overall performance and these key design parameters but without further investigation of corresponding flow field details. Additionally, as the authors pointed out, without upstream blades, the real coupled influence was not captured. Because upstream wake is the main source of pressure loss, it should receive more attention when evaluating ITD performance accurately. Another question that concerned the writers is whether the throat area of the LPT vane remains unchanged, which should be a more important parameter for vanes.

Moreover, almost all attention has focused on ITD performance, while how nearby turbines, especially upstream blades, are influenced has received much less focus.

In this paper, the ITD model is designed with upstream swirl vanes (which can supply wake and inlet swirl angle) and downstream LP nozzle vanes (which can supply a real downstream flow field with potential and blockage effects), as shown in Figure 1 [Figure 1: see original paper]. Eight configurations are selected using numerical methods. All model configurations have the same AR and axial length but different radial offsets. Performance comparison and flow field analysis for these models will be presented to identify the influence of ITD radial offset on turbine performance and flow field. It is hoped that the results will provide sensible guidance when more aggressive ITD designs are ready for use in future advanced engines.

Fig. 1 Definition of ITD model

Numerical Model and Boundary Conditions

The original conservative ITD prototype is a well-designed duct positioned between HPT and LPT with an RA of 8 degrees from an aero-engine. To further improve LPT power output, this paper aims to study the ultimate potential of such ITDs.

To investigate the influence of RA on ITD flow field separately, parameters other than RA are kept unchanged across models. Inlet conditions and swirl blades remain the same, as does the axial outlet position (inlet of LPT nozzle). The parameters of ITD and LP nozzle are listed in Table 1 . Numerical models with RAs of 8° (A8), 15° (A15), 20° (A20), 25° (A25), 30° (A30), 35° (A35), 40° (A40), and 45° (A45) are investigated in parallel. However, the inner and outer walls of the ITD outlet section are modified to ensure equivalent AR. Meanwhile, the hub and casing curvature of the LPT nozzle are adapted to the modified duct. As shown in Figure 2 [Figure 2: see original paper], A8, A20, A30, and A40 are presented.

Table 1 Parameters of ITD

It should be noted that the LPT nozzle blade numbers of all models are kept the same (44 for swirl blades and LPT nozzle). As mentioned above, the authors aim to distinguish RA’ s influence on the ITD. To achieve this goal, the same throat area of the LPT nozzles must be realized to guarantee equivalent mass flow rates. Usually, there are two ways to realize this requirement: increasing the blade number of the nozzle while keeping all blade profiles unchanged, or altering the blade profile to some extent on one or multiple sections while keeping the blade number unchanged. Although the first method makes model modification easier, the accompanying unexpected relatively larger weight of the LPT nozzle and increased profile loss may eliminate efforts to further improve turbine performance. Therefore, the second strategy is adopted. The throat area remains constant at 0.0198 m^2 in all studied cases, achieved by altering the nozzle’ s outlet metal angle, stagger angle, and ratio of outlet passage width to blade thickness. The unguided turning angle is amended to smooth the blade surface. Boundary conditions for all models are the same, with a constant mass flow of 8.52 kg/s at the inlet and a back pressure of 240 kPa. The Reynolds number is maintained at 3×10 to avoid its influence on boundary layers.

Commercial CFD software Numeca is used. The topology for building model mesh is O4H, and matching periodicity technology is employed to obtain an accurate flow field in blade passages. Figure 3 [Figure 3: see original paper] shows the numerical model and structured grids of A40 as an example created by IGG. The mesh at the blade root and leading edge is magnified locally to show detailed mesh features. The mesh orthogonality is good enough to reflect the flow field, and Y+ near the wall is set below 1 to fulfill the requirements of the Shear Stress Transport (SST) turbulence model. With the same blade count, a full non-matching frozen rotor between swirl vanes and LPT nozzle is chosen to provide steady upstream wake for the ITD.

Mesh quantity is an important factor for numerical results, making a mesh independence test necessary. Figure 4 [Figure 4: see original paper] shows the relationship between total pressure ratio and mesh quantity for A8. This regularity applies to other models as well. From this chart, mesh count does not influence pressure ratio when it exceeds 3 million, indicating that the flow field is independent of mesh count. Therefore, each model is simulated with more than 3 million meshes, and their computation results are ready for comparison.

Fig. 2 The sketch of different RA ITDs

Fig. 3 Numerical Model and Mesh of A40

Fig. 4 Test of Mesh Independence

Results and Discussion

All models are simulated with varying RA. In the following sections, numerical results are systematically presented. According to simulation results, RA is an important factor that can change ITD performance and nearby turbine blade characteristics.

After simulation, separation occurs in A45 and cannot be diminished by modifying the LPT nozzle blade profile. Therefore, this paper advises that RA should be set smaller than 45° to prevent separation. A45 will not be discussed further to avoid severe numerical error.

Figure 5 [Figure 5: see original paper] shows yaw angle and Mach number at the ITD inlet section, Plane C1. From these figures, compared with Mach number, yaw angle is less influenced by RA variation, indicating that angle is more stable under potential field effects. Except near the hub region, the initial A8 case has a larger yaw angle in the main flow region than other cases. This can be attributed to the relatively smaller concave curvature along the inner wall of A8. More specifically, in A8, the tangential velocity is smaller near the hub region, which reduces the yaw angle slightly. For the same reason, the yaw angle above 50% passage height in the A40 model is the smallest, particularly at 95% passage height.

At the ITD inlet section, it is evident that larger RA is accompanied by smaller Mach number. While the A40 case exhibits evident Mach number unevenness, its A8 counterpart has a more even Mach number distribution. Therefore, it is important to focus on this characteristic to achieve a more uniform flow field.

Detailed flow field on the S3 stream surface is useful for observing local flow characteristics comprehensively. Figure 6 [Figure 6: see original paper] presents the ITD inlet (Plane C1) Mach number. In smaller RA cases, the high Mach number region occupies almost the entire passage height. However, the range of low Mach number increases gradually as RA increases steadily. When RA reaches its peak in case A40, Mach numbers higher than 0.7 are only seen above approximately 85% passage height. Stemming from the pressure side of the swirl blade, the low-velocity region develops, and the core of low-energy fluid remains close to its pressure side.

According to Mach number and yaw angle variations, ITD RA affects HPT rotor deviation angle. Consequently, HPT performance is influenced, which should be forecasted during the turbine design process.

Static pressure distributions on swirl blade surfaces are compared in Figure 7 [Figure 7: see original paper], with black dotted lines representing static pressure isolines. According to Figure 7(a), blade surface static pressure increases when RA is larger. High-pressure regions (greater than 292 kPa) extend from areas near the casing to the entire blade surface, indicating that fluid decelerates near the blade in larger RA models. Low-pressure corner regions (less than 254 kPa) locate at the blade trailing edge near the root in A8 but move near the casing in A30 and A40. As the swirl blade passage is horizontal and the blade profile was designed without twisting, the pressure distribution is influenced by stream-wise adverse pressure gradient. Low-pressure corner area will change blade deviation angle. The suction surface pressure feature varies. Low-pressure corner region is not observed; instead, high-pressure corner region develops from areas near the casing in A8 to areas near the root in A40. Low-pressure regions narrow down from the entire blade surface in A8 to the trailing edge area in A40 (the blank area at the trailing edge).

In subsonic turbine stages, on one hand, more attention should be given to the downstream flow field; on the other hand, different RA can simultaneously cause changes to the upstream flow field. With larger ITD outlet radius in the investigated models, the inlet flow for the LPT nozzle differs. The yaw angle and velocity distribution would present various features that affect downstream turbine performance.

More specifically, flow field on the S2 stream surface of the duct will be discussed. If separation occurs within the S2 stream surface, the three-dimensional flow field must be poor and the design process should be restarted. Pressure loss distribution can provide objective guidance on whether the duct wall curvature design is acceptable.

Total pressure coefficient Cpt is defined here as:

$$C_{pt} = \frac{P_t - P_{t0}}{P_{t0}}$$

where P_t is local total pressure and P_{t0} is the mass-averaged inlet total pressure of ITD. Cpt represents total pressure loss, with larger values indicating higher total pressure loss.

Figures 8 [Figure 8: see original paper] and 9 [Figure 9: see original paper] show Cpt and Mach number contours for different models, with only A8 and A40 presented. Dashed white lines indicate stream-wise planes, S3 surfaces, C1 to C5 correspondingly. C1 to C5 planes are approximately perpendicular to streamlines along the ITD, intended for detailed flow field analysis.

Within the ITD, ultra-high total pressure loss spots mainly concentrate on the hub wall as ITD RA decreases (red dashed ellipse areas in Figure 8). This indicates a thicker hub boundary layer on the inner wall of smaller radial offset ITDs (noting that no separation occurred in any models except A45). The inner wall curvature affects the low-velocity region, which moves gradually toward the first bend of the ITD in higher ITDs (dashed blue ellipses in Figure 9). As ITD mean radius increases, ITD wall curvature differs, causing variable positions of peak adverse pressure gradient along the inner wall. Consequently, low-energy fluid mainly congregates near the first bend of the ITD.

Pressure loss characteristics near the casing differ from those near the hub. The high total pressure loss area near the casing remains almost constant, extending from swirl blade to LPT nozzle (Figure 8). However, a more evident and lower velocity area appears within the ITD (Figure 9) and locates near the LPT nozzle for all models. As RA increases, the adverse pressure gradient increases along the outer wall within the ITD, and the peak magnitude region remains near the LPT nozzle.

Boundary layers on duct walls are severely influenced by RA, as discussed above. To further investigate how wall shear stress is affected, streamlines and static pressure coefficient Cps are presented simultaneously.

Static pressure coefficient Cps is defined as:

$$C_{ps} = \frac{P_s - P_{s0}}{P_{t0} - P_{s0}}$$

where P_s is local static pressure, and P_{t0} and P_{s0} are mass-averaged total and static pressure at the ITD inlet, respectively. Cps can distinguish how the static pressure field changes due to various factors while ruling out velocity effects.

In all ITDs, the LPT nozzle is located before the second bend of the ITD, while the first bend starts immediately downstream of the ITD inlet section (the starting position of the figures). From Figure 10 Figure 10: see original paper, completely different limiting streamlines appear on the hub wall. For A8, smooth surface limiting streamlines near the inlet concentrate near the hub wall, while streamlines before the LPT nozzle are nearly perpendicular to the axial direction, indicating extremely large circumferential velocity compared with axial velocity. This phenomenon suggests that a relatively extremely thick and low-velocity boundary layer concentrates near the ITD hub wall.

Variation of the saddle point location near the LPT nozzle leading edge with increasing RA implies different inlet angle performance. Comparatively, limiting streamline skewness near the ITD inlet is more overt when RA increases. Span-wise pressure gradient changed by the first and second bends of the ITD induces this difference. With larger RA, higher span-wise pressure gradient can be found at both the first and second bends within the ITD, causing low-momentum boundary layer flow accumulated near the hub at the first bend position to mix with the mainstream flow. Such flow movement may result in less low-momentum flow accumulating near the hub wall, thereby reducing streamline skewness. Cps contours reaffirm this variation.

Similarly, flow movement near the outer wall differs between cases. The limiting streamline on the shroud of A8 is smoother compared with its A40 counterpart, formed by the smaller stream-wise pressure gradient near the first bend. Comparing limiting streamlines on hub and shroud, they are more easily influenced near the inner wall.

Figure 11 [Figure 11: see original paper] shows Cpt on the S2 stream surface at 50% span. The dissipation of trailing edge shed vortex generated by upstream swirl vanes can be observed clearly. However, since RA varies among cases, the actual length (not axial length) of the ITD is also modified. Therefore, even though swirl vane design remains unchanged across all cases, different circumferential positions of upstream trailing edge induced wake can be scrutinized at the LPT nozzle inlet. Under this influence, the incidence angle of the LPT nozzle will undoubtedly change, consequently altering LPT nozzle performance. Therefore, it is necessary for designers to optimize blade profile and wall curvature, especially parts near the ITD outlet section, to improve downstream turbine performance.

Finally, total pressure losses of all models (except A45) at different planes (Figure 8) are compared using Cpt in Figure 12 [Figure 12: see original paper].

C1-C2 represents pressure loss from Plane C1 to C2. Apparently, pressure loss is most severe between C1 and C2 for all models, near the first bend of the ITD. In this region, upstream wake is strongly dissipated, increasing pressure loss. After C2, pressure loss is generated by wake dissipation and boundary layer accumulation. While pressure loss exhibits a relatively weak relationship with RA, larger RA does not necessarily lead to more severe pressure loss. For example, Cpt of A20 is lower than all other models. Therefore, RA is not the only decisive factor for pressure loss, and other reasons should be considered. In this paper, wall curvatures are modified by RA while length and area ratio remain constant. Consequently, to reduce loss, it is strongly advised that wall optimization should be performed after RA is determined.

Fig. 5 Yaw angle and Ma at ITD inlet section (Plane C1)

Fig. 6 ITD Inlet Ma (From Downstream)

Fig. 7 SP Distribution on Swirl Blade Surface

Fig. 8 Cpt of S2 Stream Surface

Fig. 9 Ma of S2 Stream Surface

Fig. 10 Limited streamlines and Cps on hub and shroud

Fig. 11 Cpt at 50% Passage Height

Fig. 12 Cpt of different models

Conclusions

In this paper, the authors investigate the influence of RA on upstream blades and ITD using numerical methods. Based on the same original model from a real engine, seven ITDs are simulated. However, separation is found in A45 due to excessive RA, so it is advised that RA should be smaller than 45° to prevent separation. After analysis, several conclusions are drawn:

Larger RA will cause the high-loss area (also the low Mach number area) near the hub to move toward the ITD inlet and increase the risk of separation within the ITD.

At the ITD inlet section (outlet of swirl vanes), relatively lower Mach number area distributes near the inner wall region, and lower yaw angle area separates near the outer casing when RA increases, indicating a stronger potential effect of ITD. Therefore, it is essential for designers to modify or optimize HPT rotor blade profile when improving ITD. More specifically, the outlet angle and stagger angle should be larger to maintain a relatively constant angle, especially near the shroud.

With various RAs, limiting streamlines on walls present individual characteristics. More specifically, they are more twisted near the first bend, indicating that the flow field near the first bend is affected most. However, total pressure loss analysis shows that loss is not necessarily larger in ITDs with higher outlet radius, making an acceptable loss level with large RA possible. Cpt is superiorly high when the ITD is too aggressive.

In conclusion, RA selection plays a key role in ensuring a well-designed ITD. Since the flow thread through the ITD will determine downstream flow field quality and hence downstream component aerodynamic characteristics, this flow movement characteristic should be kept in mind when studying ITDs for engine application. An important finding of this paper is that total pressure loss can be controlled in larger RA ITDs, while optimization is essential.

Acknowledgements

Thanks for the support from the National Natural Science Foundation of China (No. 51406204).

References

[1] Dominy, R. G., Kirkham, D. A.. 1995. The Influence of Swirl on the Performance of Inter-Turbine Diffusers. VDI Berichte 1186, pp. 107 122.

[2] Dominy, R. G., Kirkham, D. A.. 1996. The Influence of Blade Wakes on the Performance of Inter-Turbine Diffusers, ASME J. Turbomach., 118, pp. 347 352.

[3] Hu, S. Z., Zhang, Y.F., Zhang, X.F., and Vlasic, E.. 2011. Influences of Inlet Swirl Distributions on an Inter-Turbine Duct, Part I: Casing Swirl Variation. Proceedings of ASME Turbo Expo 2011. Vancouver, British Columbia, Canada. GT2011-45554.

[4] Zhang, Y.F., Hu, S.Z., Zhang, X.F., and Vlasic, E.. 2011. Influences of Inlet Swirl Distributions on an Inter-Turbine Duct, Part II: Hub Swirl Variation. Proceedings of ASME Turbo Expo 2011. Vancouver, British Columbia, Canada. GT2011-45555.

[5] Miller, R. J., Moss, R. W., Ainsworth, R. W., and Harvey, N. W.. 2003. The Development of Turbine Exit Flow in a Swan-Necked Inter-Stage Diffuser. Proceedings of ASME Turbo Expo 2003. Atlanta, Georgia, USA. GT2003-

[6] Miller, R. J., Moss, R. W., Ainsworth, R. W., and Harvey, N. W.. 2004. The Effect of an Upstream Turbine on a Low-Aspect Ratio Vane. Proceedings of ASME Turbo Expo 2004. Vienna, Austria. GT2004-54017.

[7] Marn, A., Göttlich, E., Pecnik, R., Malzacher, F. J., Schennach, O., and Pirker, H. P.. 2007. The Influence of Blade Tip Gap Variation on the Flow Through an Aggressive S-Shaped Intermediate Turbine Duct Downstream a Transonic Turbine Stage - Part I: Time-Averaged Results. Proceedings of ASME Turbo Expo 2007. Montreal, Canada. GT2007-27405.

[8] Göttlich, E., Marn, A., Pecnik, R., Malzacher, F. J., Schennach, O., and Pirker, H. P.. 2007. The Influence of Blade Tip Gap Variation on the Flow Through an Aggressive S-Shaped Intermediate Turbine Duct Downstream a

Transonic Turbine Stage - Part II: Time-Resolved Results and Surface Flow, Proceedings of ASME Turbo Expo 2007. Montreal, Canada. GT2007-28069.

[9] Sanz, W., Kelterer, M., Pecnik, R., Marn, A., and Göttlich, E., 2009. Numerical Investigation of the Effect of Tip Leakage Flow on an Aggressive S-Shaped Intermediate Turbine Duct. Proceedings of ASME Turbo Expo 2009. Orlando, Florida, USA. GT2009-59535.

[10] Marn, A., Göttlich, E., Malzacher, F., and Pirker, H. P.. 2009. The Effect of Rotor Tip Clearance Size onto the Separation Flow Though a Super-Aggressive S-shaped Intermediate Turbine Duct Downstream of a Transonic Turbine Stage. Proceedings of ASME Turbo Expo 2009 Orlando, Florida, USA. GT2009-59934.

[11] Axelsson, L.-U. Johansson, T.G.. 2008. Experimental Investigation of the Time-Averaged Flow in an Intermediate Turbine Duct. Proceedings of ASME Turbo Expo 2008. Berlin, Germany. GT2008-50829

[12] Göttlich, E., Marn, A., Malzacher, F. J., Schennach, O., and Heitmeir, F.. 2007. Experimental Investigation of the Flow Through an Aggressive Intermediate Turbine Duct Downstream of a Transonic Turbine Stage. Proceedings of the Seventh ETC, Athens, p. 383.

[13] Pullan, G., Denton, J., and Dunkley, M.. 2003. An Experimental and Computational Study of the Formation of a Streamwise Shed Vortex in a Turbine Stage. ASME J. Turbomach., 125, pp. 291 297.

[14] Peter, B.V., Johansson, T.G., and Axelsson, L.-U.. Numerical and Experimental Analysis of the Flow in an Aggressive Intermediate Turbine Duct. 2009. Proceedings of ASME Turbo Expo 2009. Orlando, Florida, USA.GT2009-

[15] Marn, A., Göttlich, E., Malzacher, F., and Heitmeir, F.. 2009. Comparison Between the Flow Through an Aggressive and a Super-Aggressive Intermediate Turbine Duct. Montreal, Canada. ISABE-2009-1259.

[16] Hu, S. Z., Zhang, X. F., Benner, M., Gostelow, P., and Vlasic, E..2010. Geometric Optimization of Aggressive Inter-turbine Duct. Proceedings of the 14th International Mechanical Engineering Congress & Exposition. Vancouver, British Columbia, Canada. IMECE 2010-37323.

[17] Göttlich, E.. 2011. Research on the aerodynamics of intermediate turbine diffusers. Prog Aerospace Sci (2011), doi:10.1016/ j.paerosci. 2011.01.002

[18] Satta F., Simoni D., Ubaldi M., Zunino P., Bertini F., and Spano E.. 2007. Velocity And Turbulence Measurements in a Separating Boundary Layer with and without Passive Flow Control. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. 2007 221: 815, DOI: 10.1243/09576509JPE457.

[19] Axelsson, L.-U., Arroyo Osso, C., Cadrecha, D., and Johansson, T. G. 2007. Design, Performance Evaluation and Endwall Flow Structure Investigation of

an S-Shaped Intermediate Turbine Duct. Proceedings of ASME Turbo Expo 2007. Montreal, Canada. GT2007-27650.

[20] Wallin, F., Osso, C. A., and Johansson, T. G.. 2008. Experimental and Numerical Investigation of an Aggressive Intermediate Turbine Duct: Part 1-Flow Field at Design Inlet Conditions. 26th AIAA Applied Aerodynamics Conference, 18-21 August 2008, Honolulu, Hawaii. AIAA

[21] Osso, C. A., Wallin, F., and Johansson, T. G.. 2008. Numerical Investigation of an Aggressive Intermediate Turbine Duct -Part 2: Flow field under Off-Design Inlet Conditions. 26th AIAA Applied Aerodynamics Conference, 18 21 August 2008, Honolulu, Hawaii. AIAA

[22] Wallin, F., Eriksson, L. E.. 2006. Response Surface-based Transition Duct Shape Optimization. Proceedings of ASME Turbo Expo 2006. Barcelona, Spain. GT2006-

[23] Lavagnoli, S., Yasa, T., Paniagua, G., and Duni, S.. 2010. Aerodynamic Analysis of an Innovative Low Pressure Vane Placed in a S-Shape Duct. In Proceedings of ASME Turbo Expo 2010. Glasgow, UK. GT2010-22546.

[24] Karakasis, M. K., Edward, M. J. N., Miller, R. J., and Hodson, H. P.. 2010. The Effect of an Upstream Compressor on a Non-Axisymmetric S-Duct. In Proceedings of ASME Turbo Expo 2010. Glasgow, UK. GT2010-

[25] Spataro, R., Göttlich, E., Lengani, D., Faustmann, C., Heitmeir, F.. 2013. Development of a Turning Mid Turbine Frame with Embedded Design - Part I: Design and Steady Measurements. Proceedings of ASME Turbo Expo 2013. San Antonio, Texas, USA. GT2013-95279.

[26] Spataro, R., Göttlich, E., Lengani, D., Faustmann, C., Heitmeir, F.. 2013. Development of a Turning Mid Turbine Frame with Embedded Design - Part II: Unsteady Measurements. Proceedings of ASME Turbo Expo 2013. San Antonio, Texas, USA. GT2013-95280.

[27] Norris, G., Dominy, R. G.. 1997. Diffusion Rate Influences on Inter-Turbine Diffusers, IMechE J. of Power and Energy, 211 Part A, pp. 235 242.

[28] Couey, P. T., McKeever, C. W., and Malak, M. F.. 2010. Computational Study of Geometric Parameter Influence on Aggressive Inter-Turbine Duct Performance. Proceedings of ASME Turbo Expo 2010. Glasgow, UK. GT2010-

[29] 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, Elsevier, Amsterdam, pp.270 319, 1967.

[30] Zhang, Y.F., Hu, S. Z., Mahallati, A., Zhang, X.F., and Vlasic, E.. 2014. Effects of Area Ratio and Mean Rise Angle on the Aerodynamics of Inter-Turbine Ducts. Proceedings of ASME Turbo Expo 2014. Düsseldorf, Germany. GT2014-27207