J. of Thermal Science Vol.13, No.2

AITEB - An European Research Project on Aero-thermodynamics of Turbine Endwalls and Blades

F. HASELBACH H.-P. SCHIFFER
Rolls-Royce Deutschland, Turbines, Eschenweg 11, 15827 Dahlewitz, Germany
E-mail: Frank.Haselbach@rolls-royce.com

The paper delivers an overview on the European research project AITEB - Aerothermal Investigations on Turbine Endwalls and Blades, which started in year 2000 in the course of the 5. Framework Programme (GROWTH). The aim is to submit an integrated technology and design tool package for the advanced, aerothermal highly loaded design of turbines, especially:

- Experimental/numerical investigation on heat transfer and film-cooling in separated flow for highly loaded blades including advanced trailing edge cooling
- Heat transfer/ improved cooling of turbine endwalls: Experimental/numerical work on cooling of turbine endwalls, shrouds and recessed blade tips.
- Optimised CFD-process (drawing-grid-modelling-postprocessing-risk assessment) in order to derive the "best practice" to use CFD as a time effective tool.

After most of the project life, an overview on the project is delivered. Experimental results of test series at various test sites are compared to numerical simulations of the industrial and university partners.

Keywords: turbines, aerothermal, endwalls, blades, cooling-technology, experiment, CFD.
CLC number: TK474.7 **Document code:** A **Article ID:** 1003-2169(2004)02-0097-012

Introduction

Today's aerothermal and cooling design tools in the gas turbine and aircraft engine industries comprise a state of the art capability in predicting 2D and 3D flows in turbine components, whilst they still suffer from inappropriate accuracy and gaps in knowledge if complex heat transfer problems in 3D-parts of blades (shrouds and platforms) or complex, separated flows and cooling concepts for these areas are targeted. The ever stringent emission limits combined with the competitive pressure lead to ever rising demands for:

- Increased turbine peak cycle temperature
- Improved specific fuel consumption
- Increased component life
- Reduced coolant flows
- Reduced engineering time-scales ('right first time' designs)

The European aero engine and gas turbine industry has to face these challenges through the development and exploitation of new technologies and design tools for new products in order to enhance their role as a global player in this hotly contested market. The six technical work packages of the European AITEB-project, depicted in Fig.1, reflect key technologies for enhanced competitiveness of the European Industry in the turbine domain.

State of the Art

The current work in turbine cooling technology, i.e. the state of the art in these research areas is reflected by a large number of international papers and conference proceedings dealing with the aerothermal problems associated with complex, 3D heat transfer and cooling issues of modern HP- and LP turbines. Among the bulk of them, several papers deliver a comprehensive overview about the specific scientific and technical issues addressed in the current project:

Modern engine design leads to more and more

Received 2003

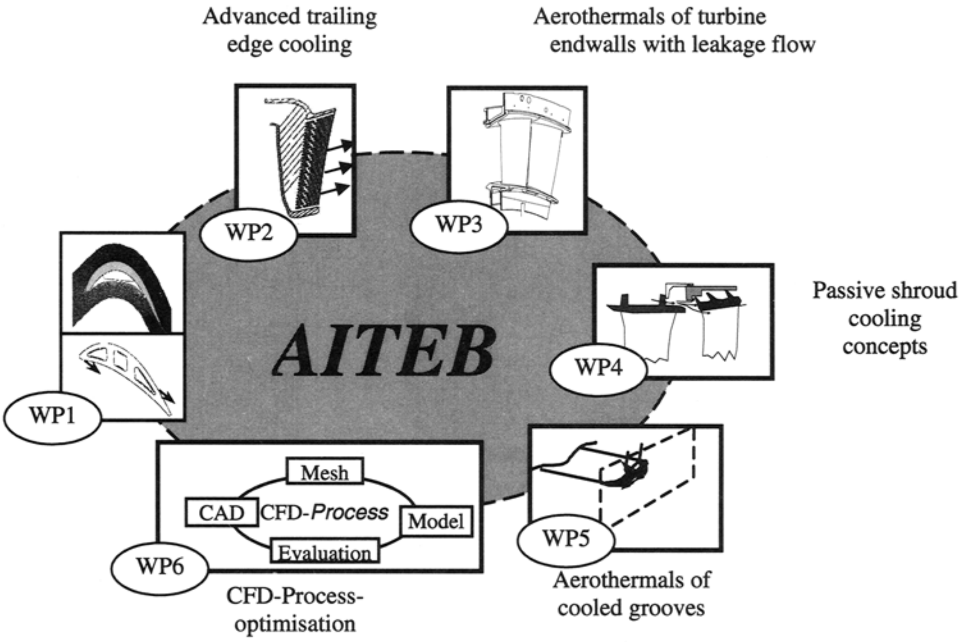


Fig.1 AITEB – Technical workpackes 1~6

highly loaded HP and LP turbine blades. This causes two main technical challenges, i.e. to predict heat transfer in regions with separation, which can no longer be avoided in such highly loaded designs. Examples for such highly loaded LP Turbine designs are to be found already in Engine use, e.g. BR715 LP Turbine, where the technology to improve this turbine was gained successfully through a similar approach to the one targeted in this project. Secondly, especially for film cooled HP turbine blades and vanes, the possible aerodynamic and aerothermal interaction of high loaded blade rows and film cooling needs to be addressed. Here, several basic research investigations (flat plate, steady flow) are reported[1~3], which emphasised the film cooling effectiveness in accelerated and decelerated flow. Effects of curvature and the aerodynamic studies were conducted by several researchers[4~6], showing considerable influence of wall curvature and inflow conditions (e.g. sensitivity to incidence variation, turbulence level, flow unsteadiness). Among these research activities no investigation of very strong adverse pressure gradients (up to separation) combined with film cooling rows is known. Especially the unsteady nature of the problem needs to be addressed more in future research.

In modern high loaded transonic turbines the aerodynamic losses of turbine airfoils are mostly covered by the thickness and the wedge angle of the trailing edges. Due to the aerodynamic requirement the thin trailing edges are the life limiting parts of the airfoils.The aerodynamic design requirements in combination with the improved casting technology leads to trailing edge slots with extreme aspect ratios and a huge fillet radius in relation to the internal slot geometry. In most cases, the conventional design tools are not validated for these geometries. Therefore, an improved knowledge of flow and heat transfer in this area is just as necessary as the further developing of the design tools. The flow and heat transfer at turbine endwalls is a subject of continuous research. The flow field close to the endwall hasbeen addressed in several papers. Main objective was the control of secondary flow in order to reduce the turbine losses[7, 8]. Recent papers are also directed to heat transfer problems at the endwalls with enlarged models in subsonic flow[9, 10] and smaller models simulating the correct Mach number[11]. Concerning the passive shroud cooling concepts, which are already in engine applications, research now focuses on a detailed understanding of the flow physics and a better assessment of the cooling effectiveness and the aerodynamic penalties combined with such a cooling concept. One passive shroud cooling concept, namely “rail cooling”, is already routinely applied in state-of-the-art aeroengines, although its design and analysis is only based on empirical methods, leaving significant scope for optimisation. Recent research activities[12~14] suffer from restricted value for general design rules. Furthermore, there are only a small number of experimental investigations dedicated to the influence of a (cooling) leakage flow on the aerodynamics of the main gas path[15]. Especially, the unsteady nature of the problem has not been addressed by most authors.

Most of the modern un-shrouded HPT (High Pressure Turbine) blades use a cooled groove technology at blade tip. The heat transfer at blade tip and the efficiency of the groove cooling impact directly the choice of cycle temperature at turbine inlet, the blade losses, the repartition of blade coolant mass flows and, particularly, the blade life. So, an important effort has been devoted, for many years, to the understanding of aero-thermal phenomena around an un-shrouded HPT blade tip, by conducting experimental[16] or numerical[17] studies. The influence of the groove has been studied in an idealized experimental set-up[18,19], previous to some numerical simulations carried out on real turbine blades[20]. Aerodynamic CFD analysis including cooled groove of blade tip start to be published[21] while nothing on heat transfer simulations or experiments has been reported in the open literature.

Concerning the overall goal in the CFD-validation of WP6, i.e. to reduce engineering time scales by “right first time” designs, the industrial state of the art and the technical approach in the project are as follows: The project, especially WP6, targets on the improvement of the CFD-process in order to enable this process to be an integral part of the engine design loop. Today, CFD is mainly used outside the design loop by computing on a final design to assess design risks. Due to this, major aim is a considerable improvement of the CFD-process from the CAD-model-meshing-computing and post-processing in terms of time scales. This will help to integrate the CFD-process into the design loop and hence reduce time scales and cost and delivering also a sound platform for a “right first time” design.

Discription of the Technical Work

Based on the IACA-project (AER2-CT92-0044), which was mainly focused on 2D unsteady wake-blade interference, and the TATEF-project, predominately targeting the aerothermal performance of turbines, the currently running AITEB project extends and enhances this research. In particular, AITEB focuses on aerothermal turbine design methods and new cooling technologies for complex 3D-components of turbine blades (such as platforms and shrouds). In demarcation to earlier projects, the AITEB project is emphasising on ‘close engine-orientated’ technology development for the complex, 3D-parts of High Pressure (HP), Intermediate Pressure (IP) and Low Pressure (LP) Turbines. The project utilises facilities and experience of the finished IACA- and the related TATEF-project. Especially for turbines, the above mentioned competitive demands lead to the following technical challenges:

- Higher turbine entry temperature, flater temperature traverses (due to modern combustors)
- Highly loaded turbine blades
- High speed low pressure turbines
- Reduced engineering time scales and costs

This is to be achieved in 6 technical work packages, which are explained in the following. As an overview, in order to emphasis their relation to turbine technology aspects, the respective relation of the work packages to the specific turbine module is depicted in Fig.2.

Heat transfer and film cooling in separated flow (WP1)

For the aerothermal design of modern turbomachinery components, the use of Computational Fluid Dynamics (CFD), which has acquired an acceptable level of maturity, becomes more and more essential. Typically, these tools have been validated for heat transfer in High Pressure Turbines (HPT) featuring high Reynolds number and attached flow on the airfoil sections. In the last years, the capability of different CFD codes to predict the heat transfer on these configurations has been investigated where special attention has been paid mainly to mesh sensitivity and turbulent effects. Very little attention has been paid to the heat transfer in separated flows in turbine representative conditions, which is typically avoided in HP Turbine environment. However, as drafted in the state of the art section, highly loaded HPT blading might show regions with main stream deceleration and local separation bubbles. Furthermore, very little attention has been paid to low Reynolds number, large separated flow regions on turbines' representative geometry, which are present in low pressure turbines of today's engines. Consequently, in this work package two tests have been commenced. The first one is dealing with a LPT airfoil (T106). The T106-300 blade section has been used as a generic geometry representative of a typical LPT highly loaded airfoil (see Fig.3). In this investigation, the T106 blade profile is subjected to extremely large incidences in order to have a long separated bubble. Mach and Reynolds number are varied around typical LPT values. Figs.3(b) and (c) show a snapshot of the experimental test data (here hot-wire traverses) in comparison to a steady state CFD prediction. Beside this flow field investigations, static pressure measurements, surface hot-film measurements deriving the state of the boundary layer and heat transfer tests using a constant heat flux method have been performed. Goal of the investigation is besides a more detailed understanding of the underlying physics of such separated flows and the corresponding heat transfer, an improved correlation for the Nusselt-number in reattachment region for low Reynolds-number flows.

In the second test set-up, dealing with a highly loaded HPT, a rotor blade section is investigated. The blade was specially designed to incorporate a separated

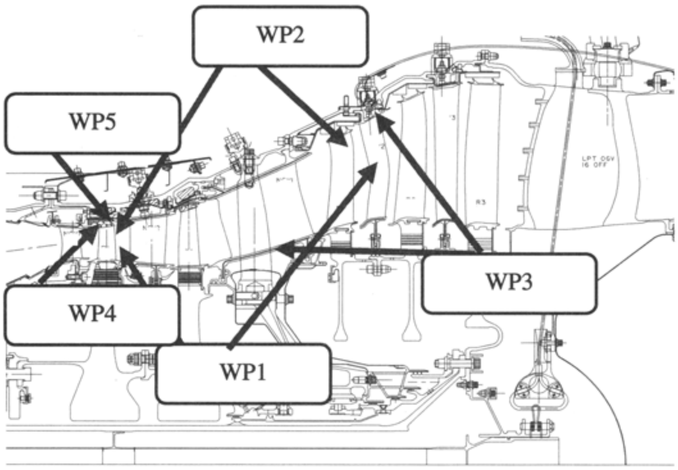
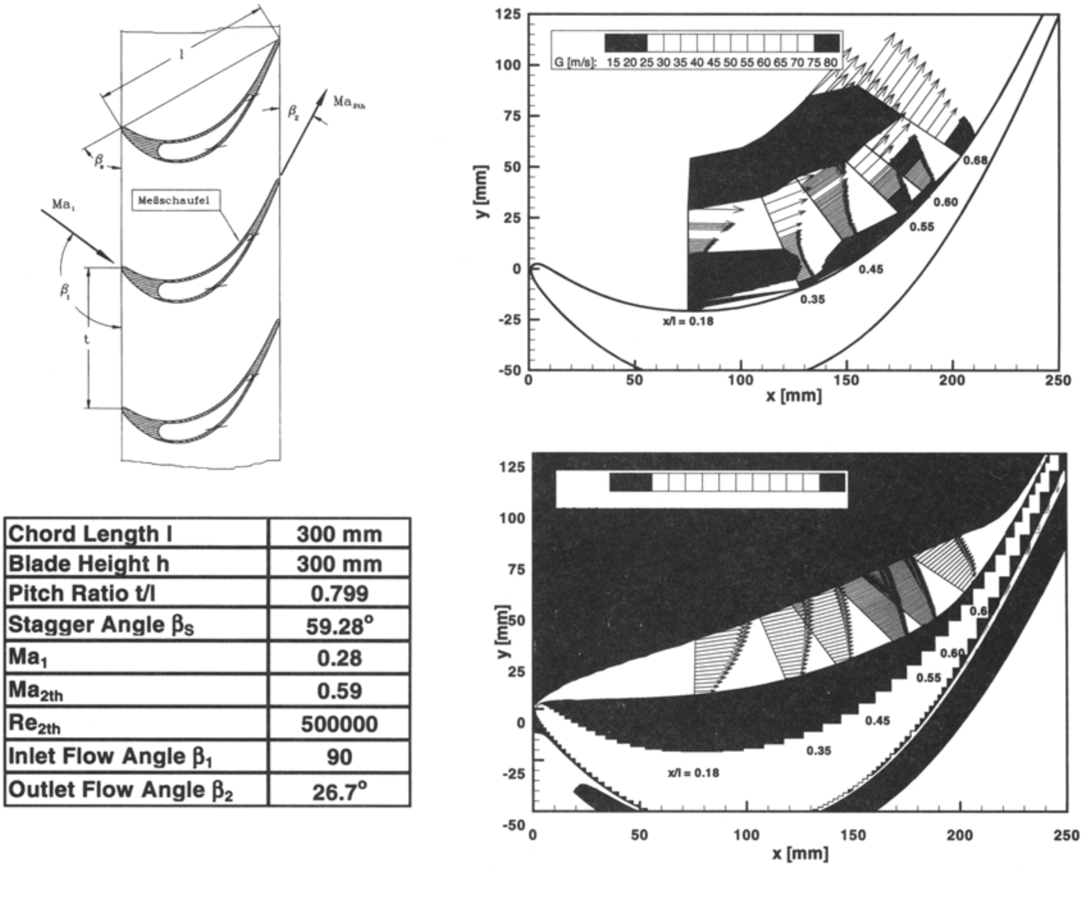


Fig.2 Relation of technical work packages to turbine component



Chord Length l	300 mm
Blade Height h	300 mm
Pitch Ratio t/l	0.799
Stagger Angle β_s	59.28°
Ma_1	0.28
Ma_{2th}	0.59
Re_{2th}	500000
Inlet Flow Angle β_1	90
Outlet Flow Angle β_2	26.7°

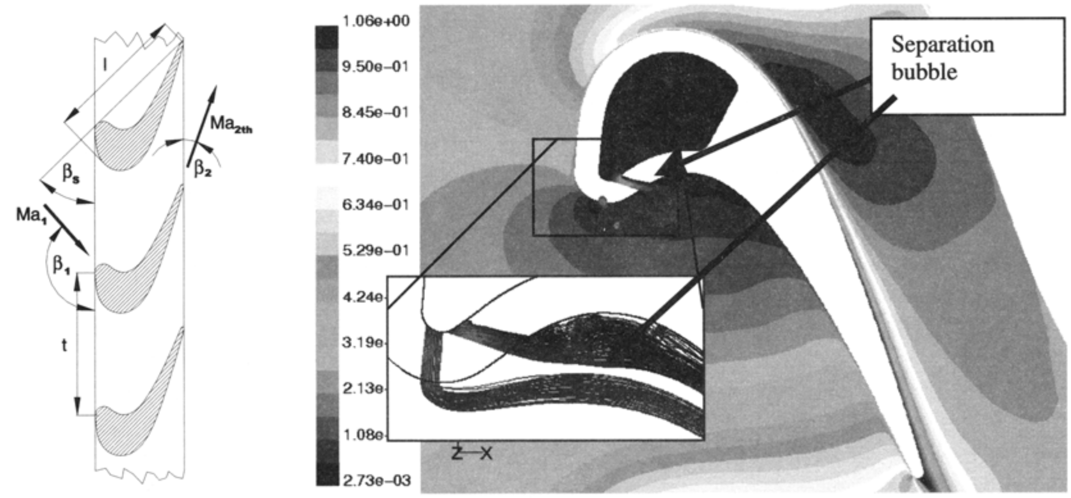
Fig.3 (A) T106 profile, (B) hot-wire measurements and (C) CFD-calculation

flow area on the pressure side leading edge. In the investigation, film cooling in this area with strong adverse pressure gradients will be employed in such a manner that high heat transfer rates and separation are avoided (HPT). The latter might not be achievable, however, a reduction of the high heat transfer rates in the reattachment region is essential for the feasibility of such designs. Fig.4 shows a schematic sketch of the newly design T120 Profile in the high speed linear cascade facility of the University of the German Armed Forces in Munich. Tests will include row characteristics, static pressure measurements, boundary layer traverses with and without simulated wakes in front of the cascade by means of moving bars wake generators. The test will therefore deliver an insight into the aerodynamic characteristics of this blade with and without dedicated film cooling in the pressure side leading edge region.

The aerothermal aspect of this problem is investigated in a complementary low speed experiment at the Technical University in Berlin, where a low speed flat plate with a film cooling row and a separation bubble is subjected to periodic inflow conditions (see Fig.5). Here, the adjacent body superimposes a turbine representative pressure gradient onto the flat plate and generates a separation bubble on the flat plate in the film cooling injection region. The experimental investigations include hot-wire and cold-wire traverses to acquire the time accurate velocity and temperature field and the cooling effectiveness of such a set-up for different blowing rates and positions of the film-cooling ejection relative to the bubble (i.e. injection of the fluid at the front, middle or rear of the separation bubble).

Advanced trailing edge film cooling (WP2)

In modern highly loaded transonic turbines the aerodynamic losses of airfoils are principally governed by the thickness and wedge angle of the trailing edge. Improvements in casting technology has resulted in cooled trailing edges of reduced thickness which has consequently resulted in an aerodynamic improvement. A further consequence is that the trailing edge cooling slots have high aspect ratios and large fillet radii in relation to the internal slot geometry. The existing design tools are only partly applicable or validated for these modern cooling slot geometries. Furthermore, the trailing edge region of a turbine airfoil is typically the life limiting feature of the airfoil and therefore it is vital that a better understanding of the flow phenomena and heat transfer characteristics of this complex region needs to be obtained. Typically, the blade trailing edge, from aerodynamic point of view should be as thin as possible. This causes a conflict with cooling requirements as difficulties of manufacturing result from the integration of internal cooling passages in the thin trailing edge region. One typical cooling technique, which enables thin trailing edges, is obtained by cutting material from the pressure side of the trailing edge forming a characteristic step. As the cutback uncovers the internal coolant passage a slot is created from which air can emerge to generate a cooling film on the trailing edge cutback. In order to increase the integrity of the trailing edge region, the structure is stiffened by rib or pin-fin



(a) T120 profile in the high speed test facility

(b) Numerical prediction of the flowfield (Mach number) and the path-lines of coolant flow (small box) on pressure side

Fig.4

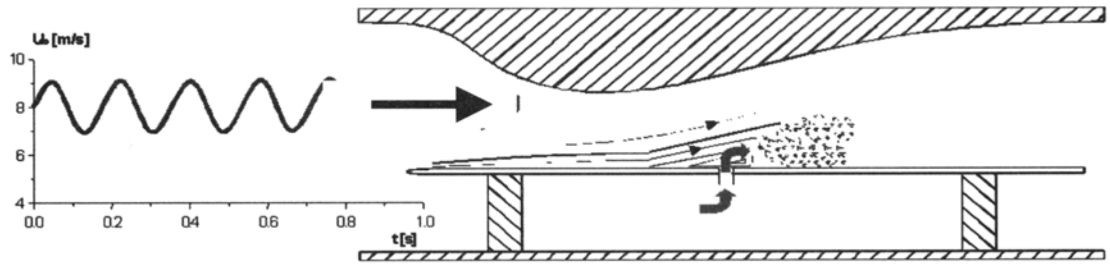
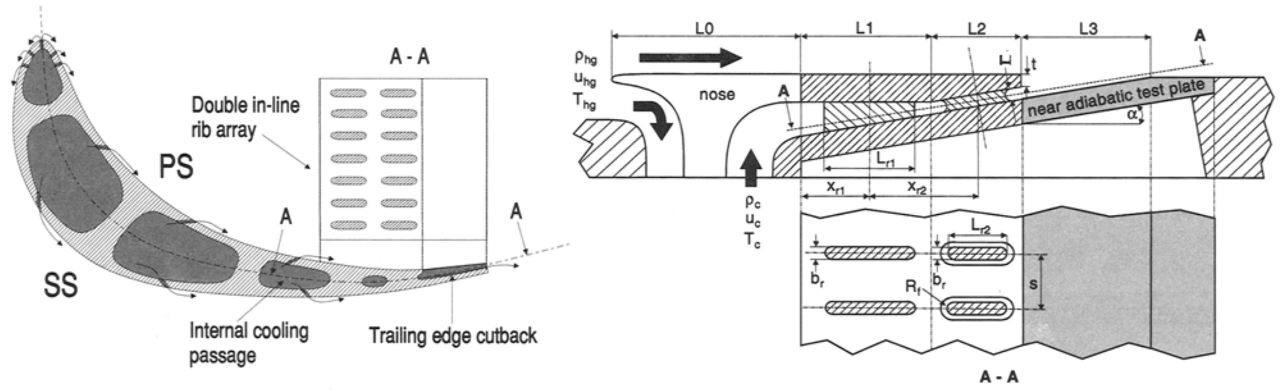


Fig.5 Low speed test set-up of the flat plate with separation bubble and film-cooling injection (and periodic inflow)



(a) Cross sectional view of a turbine blade with pressure side trailing edge cutback

(b) Experimental test set-up with generic representation of trailing edge region[23]

Fig.6

arrays located inside the internal cooling passage connecting the pressure and the suction side. Furthermore these arrays work as turbulence generators to enhance the convective heat transfer in the cooling passage and control the blade cooling mass flow. Fig.6(a) shows a cross-sectional view of a turbine blade with pressure side trailing edge cutback. There have been a number of publications dealing with two dimensional film cooling through slots, but so far the influence of rib or pin-fin arrays on film cooling has not been taken into account. The work package concentrates on the experimental and numerical investigation of a typical trailing edge configuration, where coolant ejects from a slot, disrupted by a double in-line rib array as shown in Fig.6(a). All experimental work was carried out on a scaled-up trailing edge model integrated in the atmospheric hot wind channel of the Institut für Thermische Strömungsmaschinen (ITS) at the Universität Karlsruhe, Germany. The trailing edge model investigated is scaled up by a factor of 10 and consists of a double in-line rib array. Each row is composed of 7 equally spaced ribs, depicted in Fig.6(b). The overall width of the cooling slot is 180 mm and the lateral distance between two ribs is 24 mm. In contrast to the first rib row, it was decided to provide the ribs in the second row with fillets in order to consider any influence due to the fillet radii on the flow field downstream the ejection slot. For a realistic boundary layer thickness of the main flow, the wall affected hot gas is sucked off by a variable boundary layer bleed so that a new boundary layer starts to develop on the tip of the nose. Fig.7 shows as an example the measured film-cooling effectiveness on the cut-back region after the slot for different blowing rates. As can be seen from the figure, CFD underestimates the near slot cooling effectiveness and overestimates the far region effectiveness. This is mainly due to poor mixing prediction in the steady flow calculations.

In order to get insight into the internal heat transfer, in addition to these test, the Univ. of Florence investigates the pin-fin arrangements in large scale with the use of thermochromic liquid crystal technique and particle image velocimetry (not shown here).

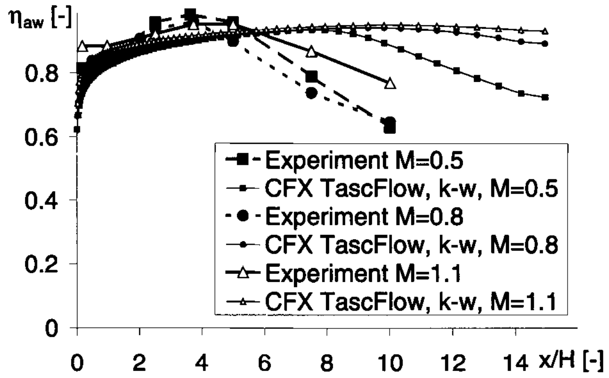


Fig.7 Comparison of measured and computed cooling effectiveness on the cut-back region (taken from [23])

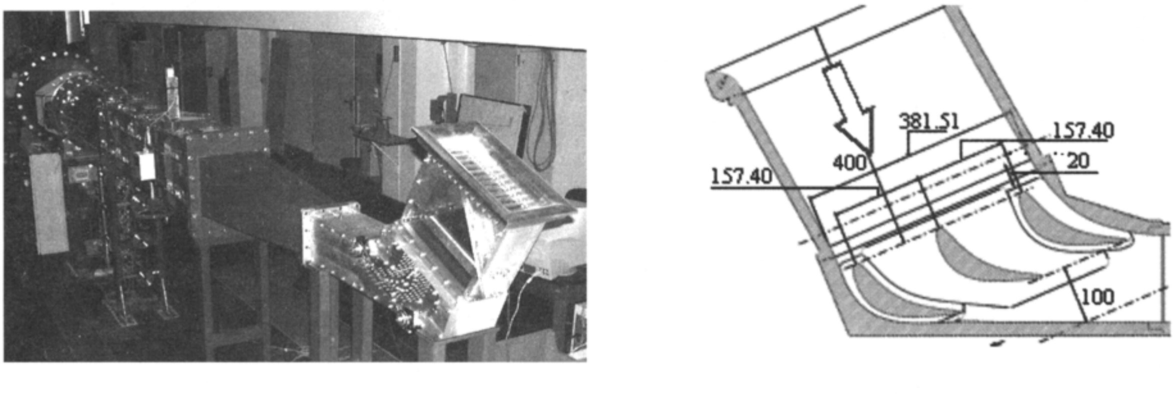
Flow and heat transfer on turbine endwalls (WP3)

The actual trend in combustor design results in more and more flat temperature profiles. Consequently higher temperatures are to be expected close to the platforms of the NGV. Additionally integrated blade casting leads to clusters of blades, which are prone to thermo-mechanical fatigue. Therefore a sophisticated design of the platform cooling system is mandatory. However, this requires detailed information about the heat transfer on the hot-gas side of all walls. Recently performed experimental investigations concerning platform cooling suggest that detailed measurements of some specific phenomena with a well adapted generic model in combination with numerical simulation are the only way to solve remaining problems. Consequently this experiment has been drafted. In the experimental set-up a linear cascade of 3 nozzle guide vanes is tested with leakage flow injection at various pitch and swirl angles with different mass flow rates, see Figs.8(a) and (b). The tests will comprise aerodynamical and heat transfer measurements in order to determine both the aerodynamic penalty and the cooling effect of the secondary flow. The aerodynamic measurements include traverses, boundary layer measurements and Particle Image Velocimetry. For the aerothermal tests the heat transfer coefficient on the endwall will be derived by means of a constant heat flux technique and an infrared camera.

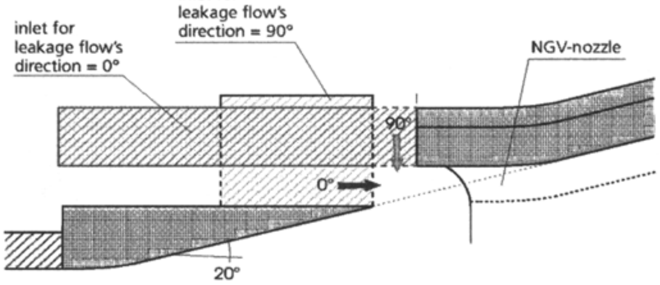
Passive shroud cooling concepts (WP4)

The cooling of turbine blade shrouds has become a necessity. Active shroud cooling (i.e. by means of internal cooling passages in the shroud) is effective, but requires thick, heavy shrouds. Passive shroud cooling, on the other hand, reduces the driving gas temperatures by ejecting coolant from stationary parts, making use of local 3D flow structures to transport the coolant to where it is needed. The "Passive Shroud Cooling Concept Investigations" of the AITEB project focuses on the understanding and improvement of passive shroud cooling. Main objective is to develop a thorough understanding of the physical phenomena associated with the passive shroud cooling concept currently in use, to develop a design methodology to optimise this passive shroud cooling concept and to develop alternative concepts. The technical objectives are as follows: To measure the cooling effectiveness of "rail cooling" (passive shroud cooling) of the shroud of a model HP turbine and to gain an understanding of the development and interaction of the primary and cooling flow fields, to develop an improved method of passive shroud cooling, to provide a data base for the validation of state-of -the-art CFD codes, to use CFD calculations to enhance the understanding and to generate design rules for the future.

The aerodynamics of the original shroudless low speed MT1 turbine at Cambridge (UK), as tested in the 3rd Framework IACA programme, were assessed for their usage as basis for the AITEB passive shroud cooling investigations. The aim was to design a shrouded rotor blade, based on the low speed MT1, whilst keeping as much dimensions as possible of MT1 stage the same as in the previous study. Similarity with the previous investigations not only offers cost benefits, but also opportunities for comparison with the previous film-cooling study. The design of the shroud for the experimental investigations is based on a plain-sided shroud design with three fins, as currently in use in engine applications. As in the engine design, the three fins are inclined forward and are stepped between the first and second seal. The gaps between neighbouring shrouds are modeled in the experiments. A sketch of the test section of this rotating rig (1 stage) and the coolant configuration can be seen in Fig.9. Figs.10(a) and (b)



(a) Picture of the test facility (test section in front) (b) Experimental test set-up at DLR Goettingen



(c) detail of the test section (side view- showing the slot ejection)

Fig.8

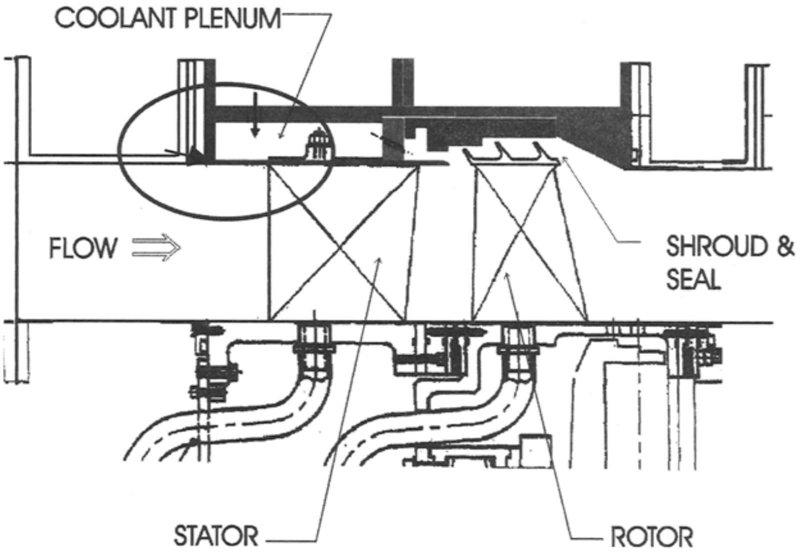
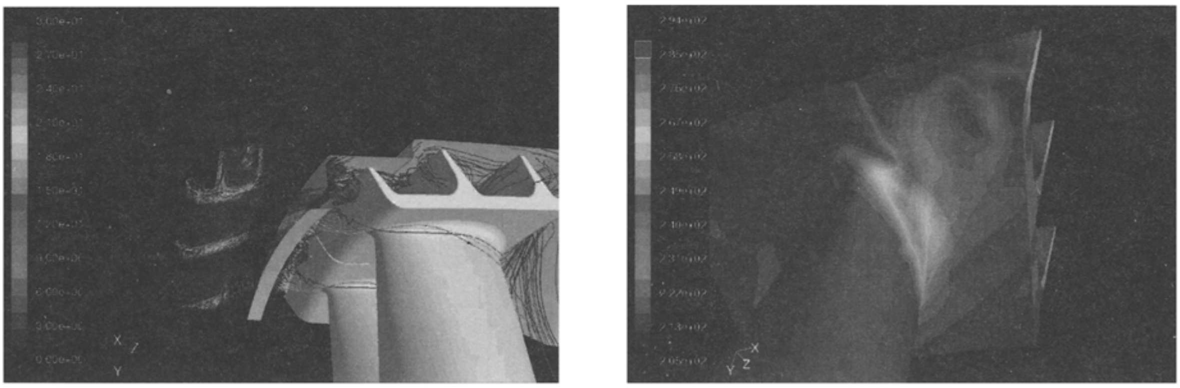


Fig.9 Schematic sketch of the test facility showing the 1-Stage turbine and the coolant injection "above" the NGV-platform to the rotor shroud



(a) (b)

Fig.10 CFD-prediction of A) coolant path-lines and B) adiabatic wall temperature

show results of CFD-prediction of wall temperature and the coolant path-lines. From this CFD-prediction, it can be seen that the coolant coming from the platform-slot above the NGV is predominantly going over the shroud. Furthermore, a sucking of the coolant into the blade low pressure regions (suction side) seems to be obvious. Objective of the test is to confirm the CFD findings and to investigate more favorable areas of coolant injection to achieve better "passive cooling effectiveness" on the gas washed surface of the shroud.

Investigations of turbine blade tip with cooled groove (WP5)

In advanced engine technology, the design of the High Pressure Turbine rotor blade is the key issue for the entire engine parameters such as engine fuel consumption, component life, engine weight, duration and cost of engine development programme. For modern un-shrouded HPT blades, the cooling of the groove at blade tip is a necessity. The efficiency of this cooling impacts directly the choice of cycle temperature at turbine inlet, the blade losses, the repartition of blade coolant mass flows and, particularly, the blade life. So, an important effort has to be devoted to the understanding of aero-thermal phenomena around HPT blade tip with cooled groove, to the build-up of associated data bases and to the validation of CFD methods for such phenomena. In this part of AITEB, aerothermal measurements on heat transfer and aerodynamics of a cooled groove are performed at the Von Karman Institute (VKI) in Brussels. Figs.11(a)~(c) show the test facility used for this linear cascade measurements. A corresponding CFD-prediction of the wall heat flux with and without coolant can be seen in Figs.12(a) and (b). The key information of this work package will be the heat transfer coefficient measured for

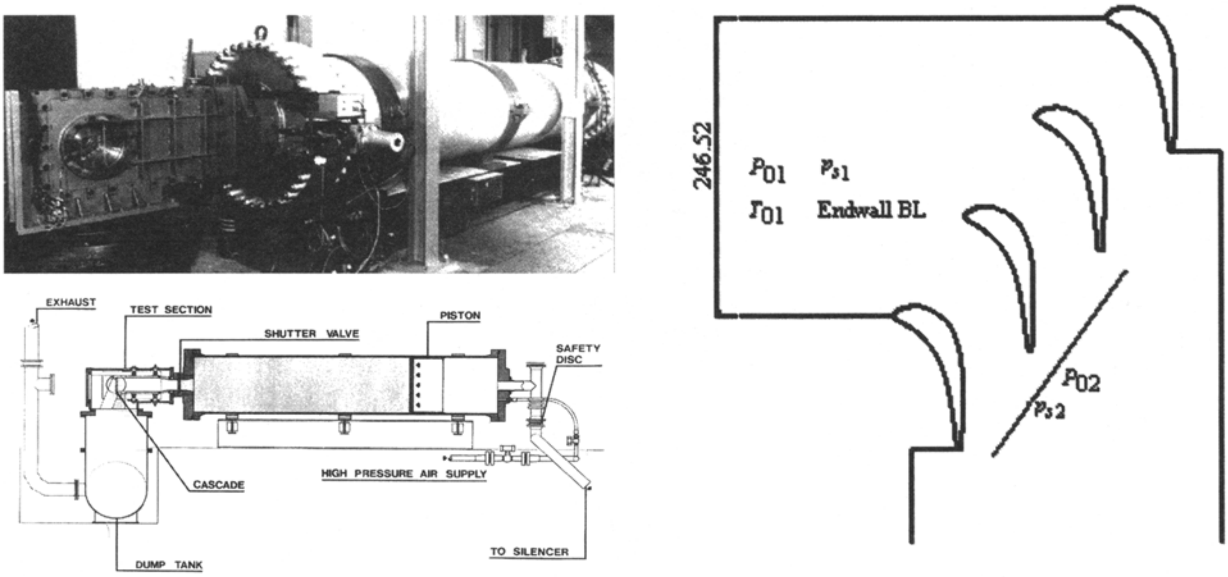


Fig.11 VKI compression tube tunnel CT-2 and test set-up for aerothermal investigations in cooled groove blade tip

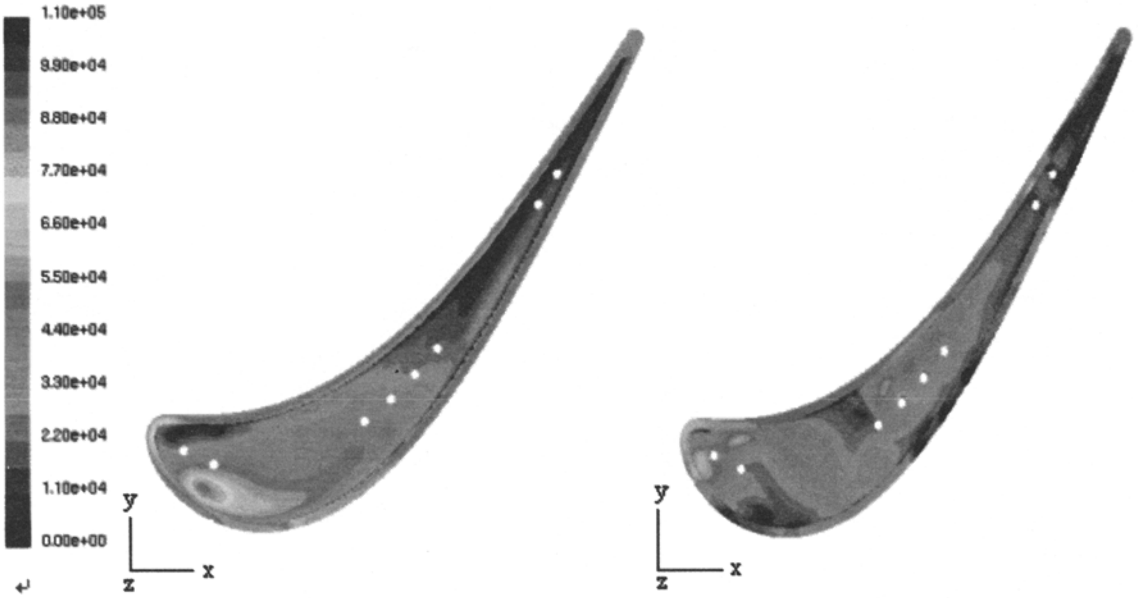


Fig.12 Predicted wall heat flux without source term cooling A) & with cooling flows B)

three different configurations (without groove=plain tip, with groove and with cooled groove) and the comparison to the corresponding CFD simulations with different flow solvers and turbulence modelling.

CFD-process (WP6)

Looking to the ever rising demand for reduced engineering time scales for engine development the use of numerical tools in the design process of single components as well as the total system is more and more important. The design process is typically an iterative procedure in which the starting design of any component has to be updated repeatedly in order to satisfy all technical requirements. The integration of numerical tools in this process includes a significant effort in grid generation adaptation whenever the geometrical boundaries are changed. Therefore, there is a considerable demand of a practical method for the reduction of the time consuming process of re-modelling the grid after a geometrical variation of a component. As part of AITEB, the Univ. of Karlsruhe and ALSTOM developed a method, which is able to significantly reduce the time needed for CFD-pre-processing. A method for re-modelling is described in [23] and, as an example, applied to film-cooled trailing edges of turbine blades for subsequent CFD-studies. The starting point of the described method is a commercial grid-generating program called GAMBIT (version 2.04) in which the entire model can be described by a sequence of string-commands. During the conventional set up of a TE-model in GAMBIT's graphical user interface (GUI) the command sequence is automatically recorded by a so called 'journal-file'. By modifying single commands in this file various design changes can be realised in the TE-region of the turbine blade. Furthermore it is also possible to adapt the computational grid in order to fit special requirements like grid density in the near wall area for heat transfer simulation. The command sequence in the journal–file is updated automatically after the input of all specific design and grid related parameters in a macro-based datasheet in MS-EXCEL. Because of the resulting significant speed-up for the model generation, the described method is especially suited for frequent design changes in an process of optimisation as well as extensive parametrical studies which are necessary for example to improve film-cooling in the TE-region. Figs.13 and 14 depict an outline of the process and some meshing example. For further detail please refer to [23].

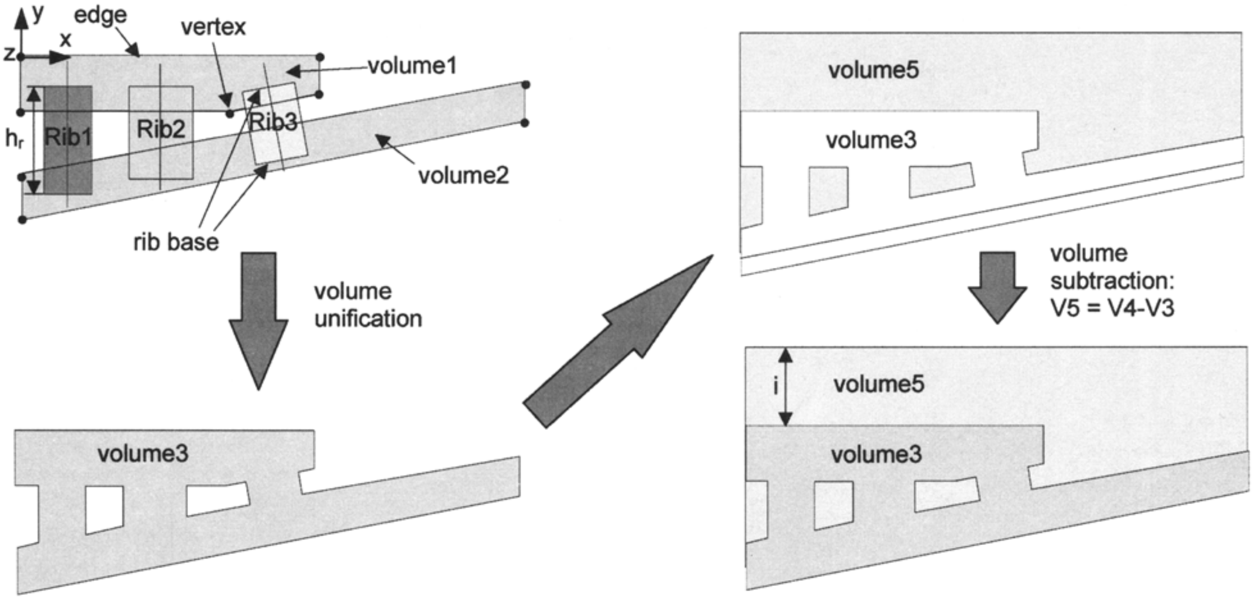
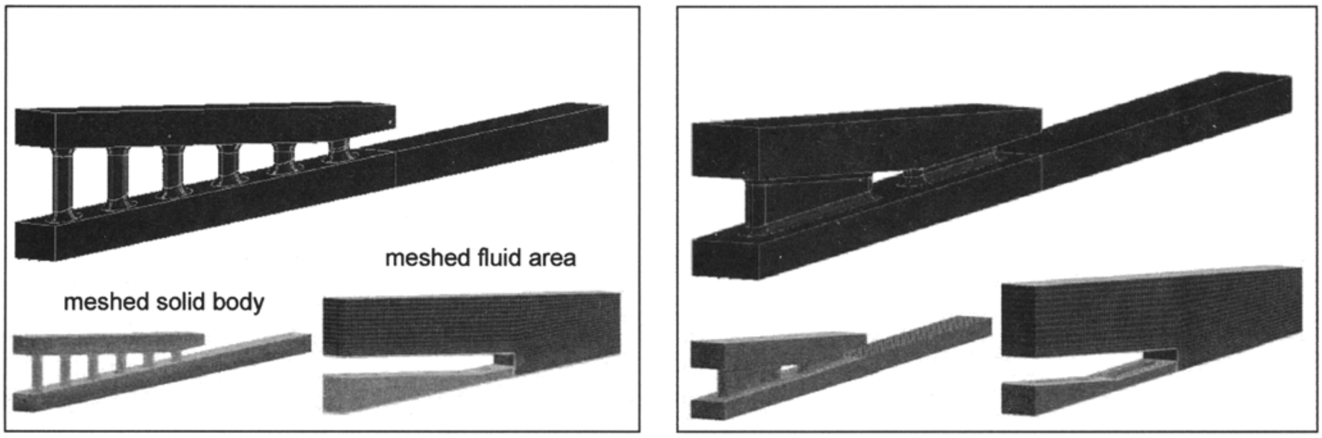


Fig.13 Automative mesh generation process (outline) taken from [23]



(a) a two-fin array (b) generated with the trailing edge mesh template[23]

Fig.14 Examples of pin-fin array

Project Outcome to Date

The expected deliverables, such as new cooling concepts, models and correlations and improved design tools will help the European gas turbine industry to remain competitive and to meet the increasing global market requirements. This will improve aircraft/aero-engine performance and environmental friendliness and contribute to the EU-policies in the aspects of competitive growth and increasing air transport capabilities. The project outcome will deliver a sound basis for sustainable employment and growth, whilst protecting the environment by reduced fuel consumption and emissions. The optimised CFD-process will improve time scales and efficiency of the CFD-application in an industrial environment. Additionally, such results will also be of major benefit to turbines contained within proposed technology platforms. From AITEB, it is expected to achieve enhanced competitiveness and sustainable growth of the European gas turbine industry due to:

- Increased engine performance due to higher temperature capability of turbines or lighter component designs
- Reduced fuel burn because of reduced cooling flows
- Reduced maintenance effort because of increased component life
- Reduced engine development costs due to reduced engineering time-scales (right first time)

The results of the comprehensive investigation can be integrated into the products of the industry partners at any time of the project.

Conclusions

The described AITEB project represents a concerted effort of the European gas turbine industry to answer the challenges in the turbine cooling domain. The project is running since February 2000 and with an anticipated running time of 4 years. The overall structure of the project and details of the technical work packages have been described. Although emphasis is put on the overall project, some first results and details of the technical work packages of the global project have been presented.

Already, a considerable amount of publications emerged out of the running project, which one can refer to for more detailed information in the technical challenges and solutions. The overall aim of this paper is to submit an overview on the research activities.

Acknowledgements

All reported work was performed within the European research project "Aerothermal Investigation on Turbine Endwalls and Blades" (AITEB, 5FP, G4RD - CT - 1999 - 00055) funded under the 5th Framework GROWTH programme. The permission for the publication by the partners and the Commission services is gratefully acknowledged by the author.

References

[1] Kruse, H. Effects of Hole Geometry, Wall Curvature and Pressure Gradient on Film Cooling Downstream of a Single Row. AGARD Conference Proceedings No.390, 1985

[2] Hay, N, Lampard, D, Saluja, C L. Effects of the Condition of the Approach Boundary Layer and of the Mainstream Pressure Gradients on the Heat Transfer Coefficient on Film-Cooled Surfaces. Journal of Engineering for Gas Turbines and Power, 1985, 107: 99—104

[3] Teekaram, A J H, Forth, C J P, Jones, T V. Film Cooling in the Presence of Mainstream Pressure Gradients. Transactions of the ASME, 1991, 113: 484—492

[4] Schwarz, S G, Goldstein, R J. The Two-Dimensional Behavior of Film Cooling Jets on Concave Surfaces. Journal of Turbomachinery, 1989, 111: 124—130

[5] Abuaf, N, Bunker, R, Lee, C P. Heat Transfer and Film Cooling Effectiveness in a Linear Airfoil Cascade. ASME paper 95-GT-3, 1995

[6] Ardey, S, Fottner, L. Flowfield Measurements on a Large Scale Turbine Cascade with Leading Edge Film Cooling by Two Rows of Holes. ASME paper 97-GT-524, 1997

[7] Duden, A, Fottner, L. The Secondary Flow Field of a Turbine Cascade with 3D Airfoil Design and Endwall Contouring at Off-Design Incidence. ASME Paper 99-GT-211, 1999

[8] Duden, A, Raab, I, Fottner, L. Controlling the Secondary Flow in a Turbine Cascade by Three-Dimensional Airfoil Design and Endwall Contouring. J. Turbomachinery, (Trans.ASME), 1999, 121: 191—199

[9] Harvey, N W, Rose, M G, Coupland, J, et al. Measurement and Calculation of Nozzle Guide Vane End Wall Heat Transfer. J. Turbomachinery, (Trans.ASME), 1999, 121: 184—190\

[10] Kang, M B, Kohli, A, Thole, K A. Heat Transfer and Flowfield Measurements in the Leading Edge Region of a Stator Vane Endwall. J. Turbomachinery, (Trans.ASME), 1999, 121: 558—568

[11] Kang, M B, Thole, K A. Flowfield Measurements in the Endwall Region of a Stator Vane. ASME Paper 99-GT-188, 1999

[12] Hwang, M, Pope, A, Shucktis, B. Advanced Seals for Engine Secondary Flowpath. Journal of Propulsion and Power, 1996, 12(4): 794—799

[13] Rhode, D, Johnson, J, Broussard, D, et al. Flow Visualization and Leakage Measurements of Stepped Labyrinth Seals, Part 1 & 2. ASME Paper No.

96-GT-136/137, 1996

[14] Waschka, W, Wittig, S, Kim, S, et al. Heat Transfer and Leakage in High-Speed Rotating Stepped Labyrinth Seals. AGARD-CP-527, 1992

[15] Hah, C. Secondary and Tip-Clearance Flows in Axial Turbines. VKI Lecture Series 1997-01, 1997

[16] Sjolander, S A. Secondary and Tip-Clearance Flows in Axial Turbines. VKI Lecture Series 1997-01, 1997

[17] Ameri, A A, Steinthorrson, E. Prediction of Unshrouded Rotor Blade Tip Heat Transfer. ASME paper 95-GT-142, 1995

[18] Chyu, M K, Moon, H K, Metzger, D E. Heat Transfer in the Tip Region of Grooved Turbine Blades. Journal of Turbomachinery, 1989, 111: 131−138

[19] Metzger, D E, Bunker, R S, Chyu, M K. Cavity Heat Transfer on a Transverse Grooved Wall in a Narrow Channel. Journal of Turbomachinery, 1989, 111: 73−79

[20] Ameri, A A, Steinthorrson, E, Rigby, D L. Effect of Squealer Tip on Rotor Heat Transfer and Efficiency. ASME paper 97-GT-128, 1997

[21] Liamis, N, Brisset, Ch, Lacorre, F, et al. CFD Analysis of Dual Spool Turbine Configuration. AIAA paper 99-2525, 1999

[22] Wolff, S, Homeier, L, Fottner, L. Experimental Investigation of Heat Transfer in Separated Flow on a Highly Loaded LP Turbine Cascade. Proceedings of the RTO/AVT Symposium and Specialists Meeting Heat Transfer and Cooling in Propulsion and Power Systems, Loen, Norway, 2001

[23] Martini, P, Schulz, A, Whithey, C F, et al. Experimental and Numerical Investigation on Trailing Edge Film Cooling Downstream of a Slot with Internal Rib Arrays. 5th European Turbomachinery Conference, Prague, 2003, Submitted for Publication

[24] Martini, P, Schulz, A, Wittig, A. Entwicklung eines Verfahrens zur Generierung vollständig parametrisierter, dreidimensionaler CFD-Rechengitter am Beispiel konvektiv-filmgekühlter Turbinenschaufelhinterkanten. DGLR Jahrestagung, Stuttgart, Germany, 2002

[25] Alfonso, A, De la Calzada, P. Steady/unsteady Numerical Simulation of Heat Transfer in Turbine Cascades with Separated Flows. 5th European Turbomachinery Conference, Prague, 2003, Submitted for Publication

[26] De la Calzada, P, Alonso, A. Numerical Investigation of Heat Transfer in Turbine Cascades with Separated Flows. ASME paper GT-2002-30225, 2002

(*continued from page* 113)

of insight into flow behaviour, not possible with experiment, was obtained. Areas of separation were identified in the turbine rotor. Overall, CFD has been shown to produce representative performance characteristics for the turbine.

Acknowledgements

The authors would like to acknowledge the financial support given by ESBI, Ireland and also by the Wave Energy Research Team, Department of Mechanical and Aeronautical Engineering, University of Limerick.

References

[1] Maeda, H, Sanathakumar, S, Setoguchi, T, et al. Performance of an Impulse Turbine with Fixed Guide Vanes for Wave Power Conversion. Journal of Renewable Energy, 1999, 17: 533−547

[2] Setoguchi, T, Santhakumar, S, Maeda, H, et al. A Review of Impulse Turbine for Wave Energy Conversion. Renewable Energy 2001, 23: 261−292

[3] Thakker, A, Sheahan, C, Frawley, P, et al. The Concurrent Engineering Approach to the Manufacture of Impulse Turbine Blades Using Rapid Prototyping Machine. Proceedings of the 17th Annual Conference of the Irish Manufacturing Committee (IMC-17). Galway, Ireland, 2000

[4] Thakker, A, Frawley, P, Sheik Bajeet, E. CA9: Analysis of a Stall Resistant Aerofoil in View of Wave Power Conversion. Proceedings of the Eleventh International Offshore and Polar Engineering Conference. Stavanger, Norway, 2001. 614−619

[5] Thakker, A, Frawley, P, Khaleeq, H B, et al. Experimental and CFD Analysis of 0.6 m Impulse Turbine with Fixed Guide Vanes. Proceedings of the Eleventh International Offshore and Polar Engineering Conference. Stavanger, Norway, 2001. 625−629 2001.