

Research into the Influence of Rotation on the Internal Cooling of Turbine Blades

Dr D.A. Rowbury

Rolls-Royce plc, P.O. Box 3, Filton, Bristol, BS34 7QE, UK

Dr S. Parneix

Alstom Switzerland, Heat Transfer Group, CH-5405, Baden - Dattwil, Switzerland

Mr D. Chanteloup

Ecole Polytechnique Federale de Lausanne, DGM-LTT, 1015 Lausanne, Switzerland

Prof. A. Lees

Dept. of Mech. Engineering, University of Wales Swansea
Singleton Park, Swansea, SA2 8PP, Wales, UK

ABSTRACT

This paper provides an overview of the on-going work in the Brite-EuRam 'Internal Cooling of Turbine Blades' (ICTB) project. The project is a collaborative, pan-European research program that brings together partners from industry and academia. The project aims to build an extensive experimental database concerning the effects of rotation on the internal flow distribution and heat transfer levels of rotor blades. The database will be used to develop correlations for integration into company design codes, thereby improving design methodologies, and for CFD code validation and improvement.

Experiments are being conducted on a number of test facilities. Test rigs at Ecole Polytechnique Federale de Lausanne (EPFL) and Alstom UK allow static tests on realistic geometries, while testing on idealised geometries with rotation are carried out on facilities at the University of Wales Swansea (UWS) and Technische Universitat Darmstadt (TUD). Finally, an engine representative geometry is to be tested with rotation, matching all major non-dimensional parameters, in the new Rotating Heat Transfer Rig (RHTR) at R-R Bristol.

The program concentrates on mass and heat transfer measurements in the bend regions of a multi-pass system, with heat transfer data collected for all passages in the engine representative model. A number of CFD codes are then used to analyse a wide variety of the test cases.

The RHTR and the static test facilities provide full surface heat transfer coefficient data through the use of thermochromic liquid crystals. The EPFL rig also uses PIV to produce detailed velocity vector and turbulence measurements. The rotating rig at TUD employs the naphthalene sublimation technique to calculate mass transfer data. The UWS rig investigates the influence of rib angle, blockage ratio, pitch-to-height ratio, Reynolds number, and hub-to-tip ratio on heat transfer levels in square and rectangular passages through the application of a uniform wall heat flux.

This paper gives a brief description of each of the test facilities involved in the program. Results derived from the test rigs at EPFL and UWS are presented, along with predictions of the EPFL data using FLUENT. These CFD predictions have been carried out at Alstom Switzerland (Alstom CH).

INTRODUCTION

Internal cooling of turbine blades is essential for efficient turbine engine performance and dictates the life of the component. Engine specific thrust and efficiency benefit from blade cooling although the use of cooling air imposes cycle penalties and can reduce aerodynamic efficiency. Cooling research aims to develop and validate design methods to give maximum cooling effectiveness for minimum cooling flow. The design methods need to be reliable in order to reduce the risks associated with future projects, thereby helping to avoid in-service short falls and high maintenance costs. As a result of competitive pressures, all of this has to be achieved in the face of decreasing pre-production timescales and costs, while accommodating increasingly stringent environmental, safety, and reliability requirements. This Brite-EuRam project is a response to these challenges. Its prime

Report Documentation Page				Form Approved OMB No. 0704-0188	
Public reporting burden for the collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to a penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.					
1. REPORT DATE 00 MAR 2003		2. REPORT TYPE N/A		3. DATES COVERED -	
4. TITLE AND SUBTITLE Research into the Influence of Rotation on the Internal Cooling of Turbine Blades				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S)				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NATO Research and Technology Organisation BP 25, 7 Rue Ancelle, F-92201 Neuilly-Sue-Seine Cedex, France				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release, distribution unlimited					
13. SUPPLEMENTARY NOTES Also see ADM001490, presented at RTO Applied Vehicle Technology Panel (AVT) Symposium held in Leon, Norway on 7-11 May 2001, The original document contains color images.					
14. ABSTRACT					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 10	19a. NAME OF RESPONSIBLE PERSON
a. REPORT unclassified	b. ABSTRACT unclassified	c. THIS PAGE unclassified			

objective is to deliver an integrated research package for the development and validation of numerical methods for the design and off-design analysis of turbine blade cooling systems.

Current design methods have been almost exclusively derived from experiments performed using simplified cooling geometries without the influence of rotation, which is sufficient for nozzle guide vane designs but somewhat lacking for rotor designs. However, it is important to pursue both static and rotating experiments to establish the effects of rotation and to determine design rules that allow corrections to static experimental data. Furthermore, the experimental data can be used to validate CFD modelling, which has difficulty in predicting heat transfer levels in highly turbulent 3D flows.

It is a high priority to obtain design methods that incorporate the influence of rotation for relevant internal cooling geometries as rotation has been shown to significantly affect the heat transfer of simplified cooling geometries.

NOMENCLATURE

Symbols

D_h	hydraulic diameter	Re	Reynolds number
Nu	Nusselt number	Ro	Rotation number
Nu_0	Nusselt number with no rotation		

OVERVIEW OF THE 'ICTB' PROGRAM

The Brite-EuRam 'Internal Cooling of Turbine Blades' (ICTB) project incorporates three experiment-based tasks - 'Realistic Geometries without Rotation', 'Idealised Geometries with Rotation', and 'Engine Representative Geometry with Rotation and Correct Non-Dimensional Levels' – and one CFD-based task, where simulations of the experiments are conducted. By moving towards matching engine conditions in stages, we are able to develop correlations and validate CFD codes progressively, allowing a clearer understanding of the influence of the various factors involved.

The engine representative geometry is a ribbed multi-pass system based on a present day aero-engine blade design (see 'R-R Rotating Test Rig' section). The work carried out in the other tasks is then closely tied to experiments on this model by means of the so-called 'baseline configuration', a square-sectioned passage with 180° bend and staggered 45° ribs on top and bottom walls (see 'EPFL Static Test Rig' section). Consequently, the experimental programs at EPFL, TUD, and UWS involve testing the baseline configuration and variations on it. Structuring the test program in this way makes it possible to build up a consistent database for correlation development and also ensures that the CFD codes are progressively validated against geometrically similar models under increasingly complex conditions.

The ICTB project is due to be completed by January 31st 2002, although it is hoped that the expertise gained to date will be further applied to increasing understanding in this critical area.

TEST FACILITIES

EPFL Static Test Rig

The main aim of the EPFL experiments is to determine the influence that bend region features have on the flow field and heat transfer distribution upstream, downstream, and within the bend. As such, four test models have been manufactured - seen schematically in Figure 1 – to investigate how rib placement/inclination, the presence of film cooling holes and/or turning vane, and the bend tip angle affect the above. In addition, the experiments provide measurements of developed flow between ribs. It is worth noting that the EPFL data presented in the results section is for the baseline configuration.

The static test facility at EPFL is suitable for making detailed flow and heat transfer measurements on realistic geometries without rotation. PIV is employed to obtain information on the flow structure (i.e. velocity vectors), while the transient liquid crystal technique enables full surface heat transfer coefficients to be calculated.

On the rig, air flows from a compressor (mass flow rate = 2 kg/s, compression ratio = 8) to a settling chamber having a cross-sectional area ratio sixteen times larger, thereby allowing the flow to settle before entering the test section. In addition, the settling chamber is equipped with a combination of perforated plates, honeycombs and meshes in order to reduce unsteadiness and swirl in the flow. Further details of the EPFL rig can be obtained from [1].

For each of the EPFL test sections, the downstream and upstream legs have a cross section of 100 x 100 mm², with a corresponding hydraulic diameter, D_h , of 100 mm and a length of $20 D_h$.

As can be seen in Figure 1, configurations two and three have two series of holes along the centreline of the bottom wall for film cooling simulation. There are four holes of 7.2mm diameter in the bend region and twelve holes of 8.3mm diameter starting downstream of the first rib after the bend. The hole pitch is half of the rib pitch, leading to a pair of holes per rib module. The total 'coolant' mass flow was fixed at 50% of the inlet mass flow, adjusted by means of mass flow meters placed upstream and downstream of the test section.

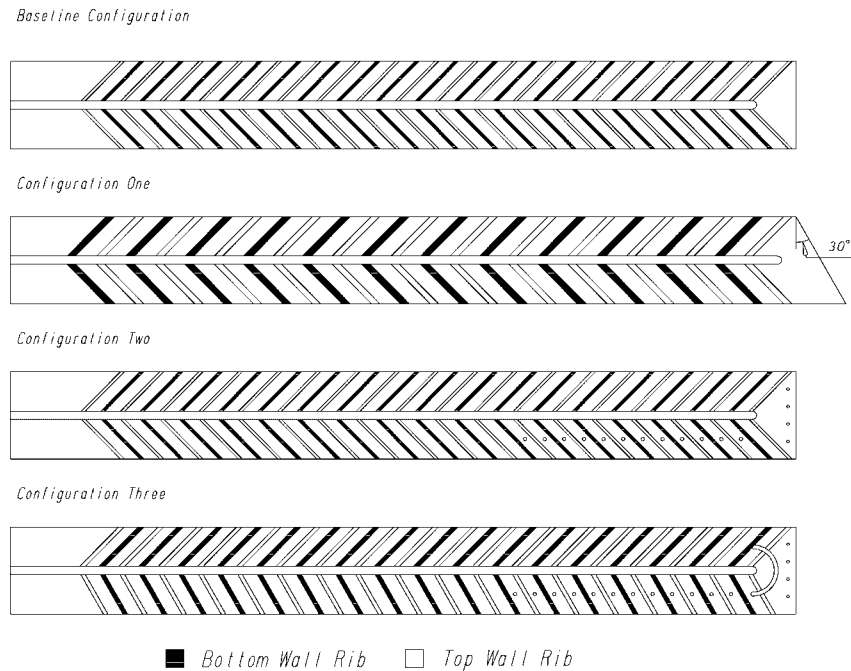


Figure 1: Schematic representation of the EPFL test configurations.

Alstom UK Static Test Rig

Within the ICTB project, the Alstom Transient Liquid Crystal Facility is used to evaluate heat transfer coefficients within idealised rotor leading and trailing edge passages.

The system uses a 1-D conduction assumption through a semi-infinite solid such that lateral conduction is assumed negligible for short duration experiments. The thick wall case is also assumed such that external conditions do not affect the internal temperature variations. It is a static facility capable of meeting typical rotor blade Reynolds number, Re , flows at engine operating conditions. Test sections are typically 5x or 10x scale models of typical blade cooling passages, manufactured from optically clear PMMA ('Perspex').

The transient technique necessitates a sudden introduction of the flow medium at a different temperature to that of the model. In this case, the flow system involves a simple open loop air conduit. Filtered air is drawn from atmospheric conditions using a 5.5kW blower, heated using a 2kW in line heater, and metered through a sharp edged orifice plate and lagged ducting. Flow initially expels to atmosphere until stable thermal and pressure conditions are achieved at which point the flow is then diverted into the test section using a pair of fast acting pneumatically operated flow gates. A schematic representation of the facility is provided in Figure 2.

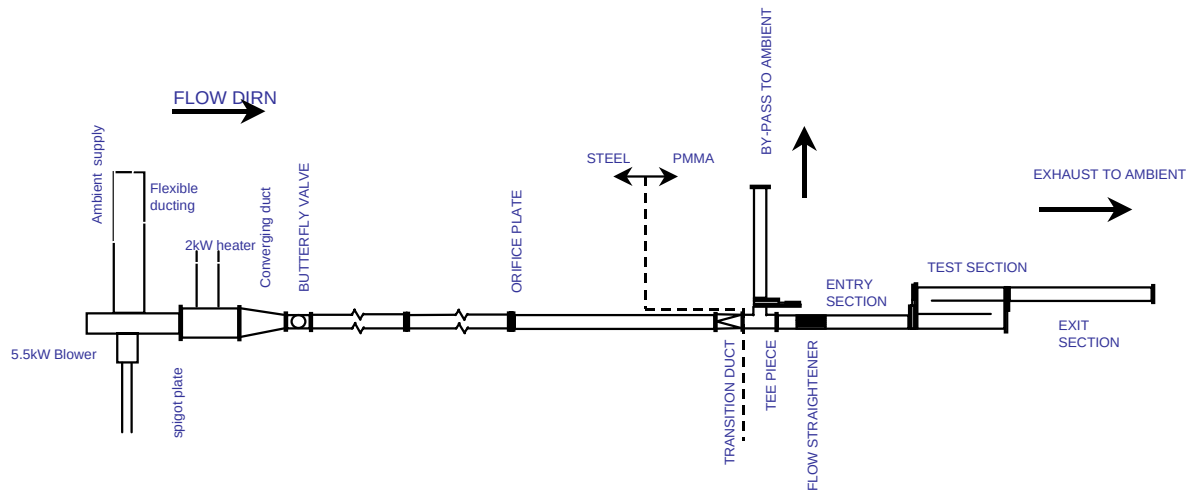


Figure 2: *Cross-section through Alstom UK static test facility.*

Full PAL resolution (762x560pixels) images are recorded transiently at 25fps using a video capture and editing workstation such that sequenced images may be relayed in digital format and stored for post processing. Instrumentation includes a series of streamwise type 'K' thermocouples through the test section for bulk flow temperature measurement and static pressure tapings for determination of leg pressure ratios. Temperatures are logged transiently and concurrently, allowing the calculation of heat transfer coefficient values over the surface when used with the image information. Pressure measurements are made at steady state and are used in the determination of channel friction factor values for estimation of the overall channel thermal performance.

Currently the rig is being validated using a square triple pass test section using a series of geometric cases (ribbed 30°, 60°, 90° vs. unribbed), while varying Reynolds number, rib pitch-to-height ratio, and rib height-to-hydraulic diameter ratio. The leading and trailing edge models will be tested under a similar range of conditions, along with investigations of the bend region.

UWS Rotating Test Rig

The UWS facility is being used to investigate the influence of Coriolis and centripetal buoyancy forces on the heat transfer of idealised single-pass geometries in rotating ducts. There are eight geometries in all, based around the baseline configuration and the Alstom UK leading and trailing edge models. The experiments investigate the influence of rib angle, blockage ratio, pitch-to-height ratio, Re , Rotation number (Ro , sometimes referred to as the Coriolis number, the inverse of the Rossby number), and hub-to-tip ratio through the application of a uniform wall heat flux. Each model is tested at 5 Rotation numbers, about 5 Reynolds numbers, and 3 Buoyancy numbers. The data collected is used to develop empirical correlations for implementation into company design codes.

Figure 3 shows the basic constructional details of the experimental apparatus. The rotor assembly consists of a main drive shaft (1) connected to a secondary shaft (2) by means of a flexible coupling (3). Each shaft is supported on two bearings, (4) and (5) respectively. Two webs (6), mounted on the main drive shaft, support the heated test sections, which model the flow geometry of a particular turbine blade cooling channel.

The right hand side of the main drive shaft (as indicated in Figure 3) has a centrally located blind hole to permit air to enter the shaft. The air then flows through the test section before returning to a centrally located blind bore on the left-hand side of the main drive shaft. The secondary shaft is hollow and is connected to the main drive shaft at the flexible coupling by means of a flexible bellows. In this way the test air can enter the rotor, pass through the test section and finally exit from the rotor at the left-hand side of the secondary shaft.

Rotary magnetic seals (11) fitted at the inlet and exit stations of the rotor permit air access to/from the rotor, which is driven by a controlled electric motor using a toothed belt drive (8).

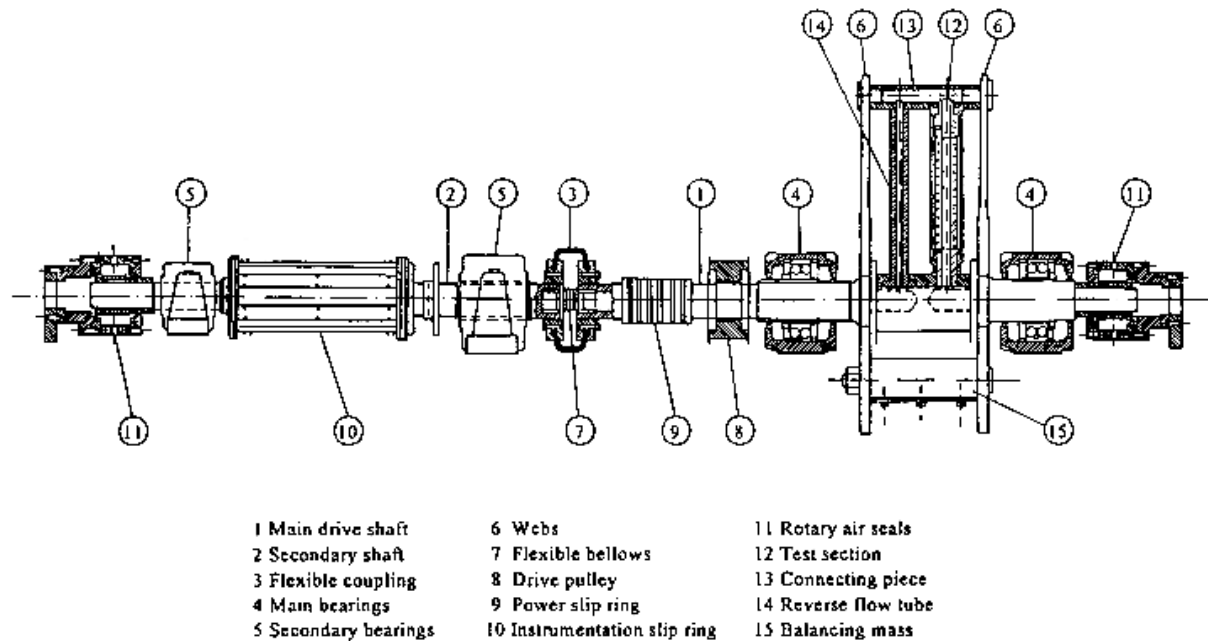


Figure 3: Cross-section through UWS rotating test facility.

An electrical power slip ring assembly (9) permits a controlled voltage to be supplied to the electrical heater fitted to the test sections. A silver instrumentation slip ring assembly (10) permits electrical signals from thermocouples attached to the test section surface to be taken from the rotor to a data acquisition system for storage and data processing.

Assorted test sections (12) may be fitted on to the rotor assembly with the connecting channel (13) acting as a structural member as well as a part of the overall flow circuit.

The test section itself is square in cross-section having sides of 14 mm. The section is made from four pieces of tufnol which are bolted together to form a tube having the required internal cross sectional shape. The leading and trailing edges of the test section are electrically heated using foil heaters with a thin sheet of copper bonded over the heaters to act as thermal smoothers. The under side of one of these copper sheets is fitted with thermocouples on the axial centre line. This side of the channel could be studied as either a leading or trailing edge by reversal of the drive motor.

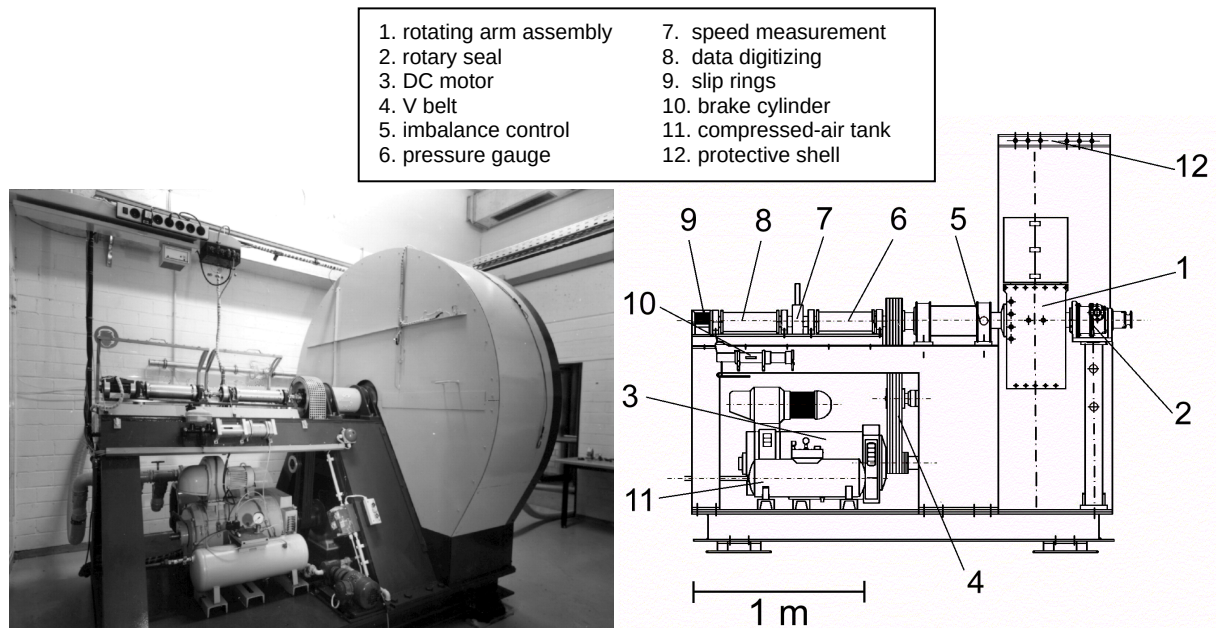
A tufnol bush fitted to the exit end of the heated test section minimises axial heat loss. An outer tufnol end piece supports the assembled heater section in an encapsulating sleeve made from aluminium to give a self contained test module which can be bolted on to the main rotor facility. The actively heated length of the test section is 183 mm.

The heated surfaces may be fitted with a variety of copper rib configurations by bonding the appropriate shapes directly to the surfaces. Thus the design philosophy permits a variety of rib configurations to be studied with the same basic test module.

TUD Rotating Test Rig

Experiments conducted at TUD use the naphthalene sublimation technique to investigate the influence of rotation on local heat transfer for a number of idealised blade cooling passage geometries. The technique measures mass transfer, but analogy between heat and mass transfer means that the results can be treated as heat transfer results for the purposes of CFD code validation and correlation development.

The rig provides the capability of measuring temperatures and pressures in the rotating frame of reference. A photograph of the facility can be seen in Figure 4, with a schematic layout presented in Figure 5. The theoretical maximum rotational speed is 1500 rpm, depending on model weight and balance.

**Figure 4:** *The TUD rotating test facility.***Figure 5:** *Schematic of TUD rotating facility.*

The key element of the test rig is the rotating arm assembly containing the model. It is fixed to a shaft driven by a 30kW DC-motor using four v-belts. Ambient air is drawn through the model by a 22kW vacuum blower which is located downstream of the apparatus. The coolant passes through an orifice meter before entering the rotating system via a rotary seal. After leaving the rotor, the mass flow is measured by a second flow meter. The leakage flow rate can therefore be determined. The pressure and temperature data measured in the rotating model is digitised before being transferred to a PC via slip rings. A more detailed description of the TUD rig can be obtained from [2].

Three two-pass test configurations will be tested at TUD, investigating the influence that Re , Ro , rib inclination, rib shape, channel inclination relative to the axis of rotation, and the presence of a turning vane have on channel heat transfer coefficient. Each configuration will be tested at Reynolds numbers of 25000, 50000, and 70000. Tests will be conducted at $Ro = 0, 0.05, 0.10$, and 0.20 for $Re = 25000$, $Ro = 0, 0.05$, and 0.10 at $Re = 50000$, and $Ro = 0$ and 0.05 at $Re = 70000$. Consequently, there are nine Re - Ro combinations for each configuration.

R-R Rotating Test Rig

This investigation will be carried out by Rolls-Royce using a recently developed rig that allows clear perspex models of full engine geometries to be spun at rotational speeds up to 3,000 rpm at a mean radius of ~ 0.75 m, thereby matching all the major non-dimensional parameters. The Brite-EuRam model is based on a current engine blade design, but at 2.75 times engine scale. Cross-sections illustrating the cooling geometry tested are presented in Figure 6. It can be seen that this consists of a ribbed, single-pass leading edge passage feeding pressure and suction surface film cooling rows, and a triple-pass for which the first two passages are ribbed and the third pass contains a pedestal bank. The first and third passages of the multi-pass feed pressure surface films. Dust hole feeds are also modelled.

Video cameras are incorporated into the model, viewing the internal surfaces of the cooling geometry. By coating the surface with a thin layer of liquid crystal, it is possible to acquire the time at which a precise temperature is reached at all points on the surface. As such, the transient liquid crystal technique is used to obtain detailed heat transfer data. During each transient test, which lasts up to 40 seconds, measurements of temperatures and pressures are captured and passed to a logging computer through an optical telemetry system. Analysis of the data has been automated using digital image processing which allows heat transfer coefficients to be calculated for each pixel of an image.

Naturally, all industrial partners are keen to validate their design codes against this data!

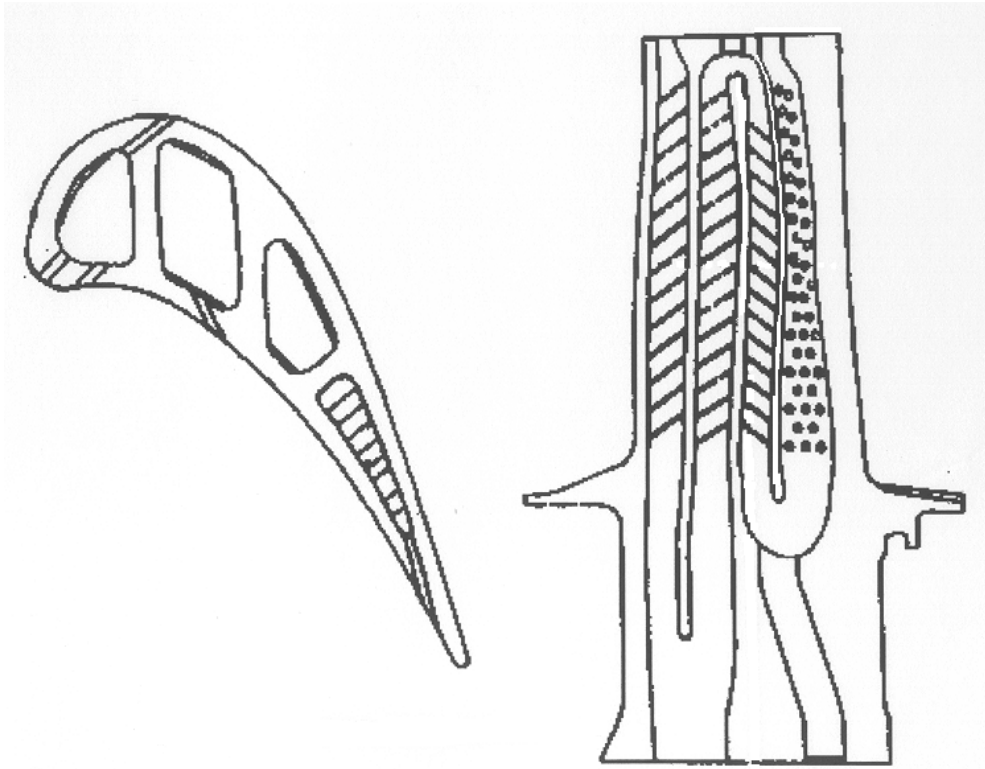


Figure 6: Cross-section through the 'engine representative' blade cooling geometry.

CFD MODELLING

CFD predictions are made using a number of codes, namely FLUENT, Mathilda (MSD), and TASCflow. FLUENT is being used by four of the partners (Alstom CH, Alstom UK, R-R Deu., and Fiat Avio), Mathilda by one (Snecma), and TASCflow by three (Alstom UK, MTU and TUD). Each partner is using a different selection of experimental results in order to validate their CFD code, although all but TUD include a comparison to the EPFL baseline case.

RESULTS

A variety of results are included in this section. Experimental data from EPFL and UWS and predictions of the EPFL data - carried out at Alstom Switzerland using FLUENT - are presented. As such, PIV measurements without rotation, along with FLUENT CFD predictions, and an illustration of the accuracy achievable with correlations developed at UWS are provided.

Static Test Data and Comparison to CFD Predictions

Figure 7 presents a comparison between PIV measurements made at EPFL and CFD predictions run at Alstom CH using FLUENT. The results pertain to fully developed flow in the leg upstream of the bend, presenting data in planes perpendicular to the ribs (plots on the left of Figure 7) and perpendicular to the channel axis (plots on the right of Figure 7). For the former the flow is from right to left (upstream on the right), while for the latter we are looking from downstream, so the channel outer wall is to the right and the divider wall the left.

The former illustrates the re-circulation present behind the ribs, while the latter highlights the secondary flow pattern present within the channel. The angling (at 45°) of the ribs to the flow results in two three-dimensional vortices being formed. In effect, the flow is driven along the ribs to the channel outer wall, then along the wall until it encounters the counter-rotating vortex. This drives it across the channel to impinge on the divider wall. The secondary flow leads to enhanced mixing, and hence heat transfer, in the channel, although there is a region of low pressure associated with it, so an increase in the overall pressure drop might be expected.

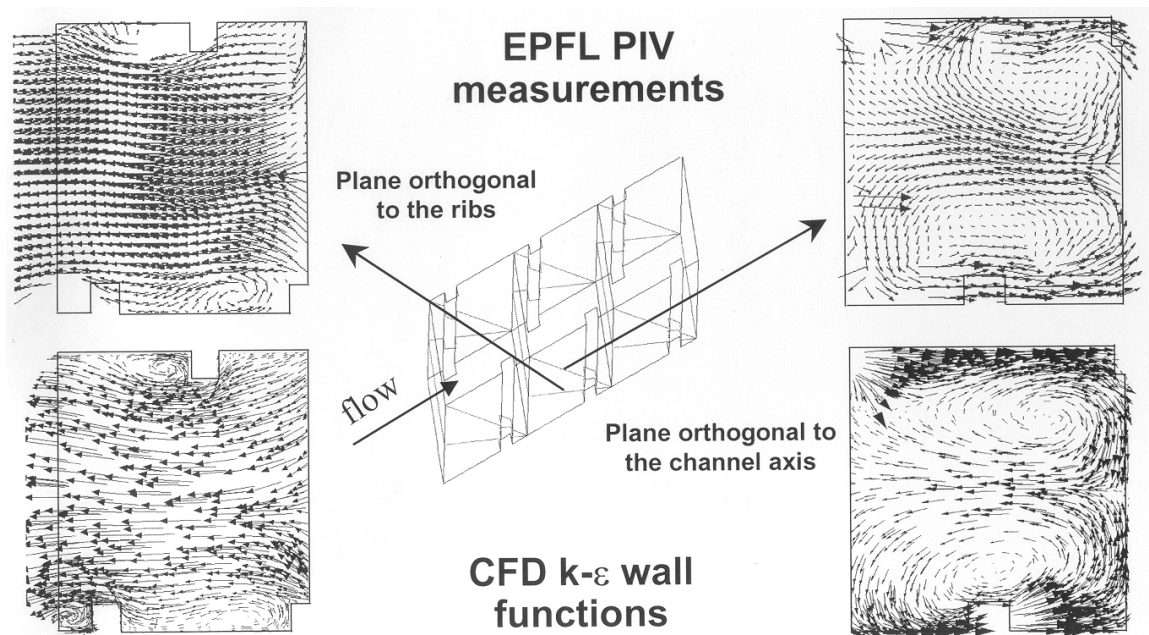


Figure 7: Comparison between EPFL measured flow vectors and CFD predictions for fully developed flow upstream of the bend (planes perpendicular to channel and to ribs)

It can be seen that the agreement between predictions and measurements of the flow structure is very good. The blank regions within the 'PIV measurements' fields represent areas where insufficient optical access could be gained for valid data collection.

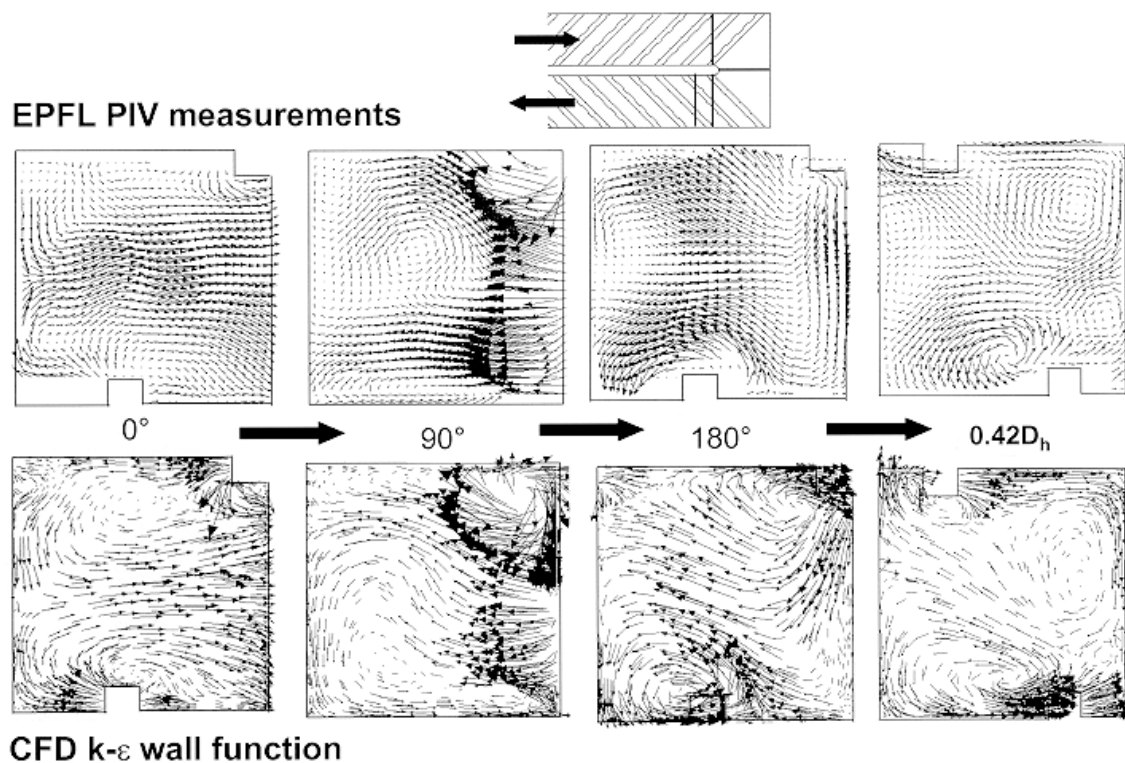


Figure 8: Comparison between EPFL measured flow vectors and CFD predictions (planes perpendicular to flow at inlet, 90°, exit, and $0.42 D_h$ downstream of the bend)

A comparison between the measured and predicted flow field for the EPFL baseline configuration, in planes located around the bend region, are presented in Figure 8. As shown in the sketch, the four planes are located at 0° (inlet), 90° , and 180° (exit) around the bend, and $0.42D_h$ downstream of the exit. In each case the channel outer wall is on the left, with the divider wall on the right - note that this is opposite to the data presented in Figure 7.

Not surprisingly, the flow field resembles the fully developed flow field (Figure 7) at inlet to the bend. At 90° the secondary vortex motion, which moves flow at mid-channel height toward the channel divider, is opposed by the streamwise acceleration around the bend. These opposing forces create extremely large vortex stretching and gradients, which can be modelled by the $k-\varepsilon$ model. The general flow pattern is in agreement, although the CFD predicts one large vortex in the outer portion of the bend, while the measurements show this to be smaller with a second recirculating region adjacent to the bottom wall. This is an indication that the CFD is slow in predicting how the cross-duct velocity component splits the large vortex seen in the CFD into two smaller vortices. At 180° there is a large asymmetry present due to the vortex contortion that has occurred in the bend. Finally, $0.42D_h$ downstream, the experiment reveals four vortex structures as the upper-wall vortex dissipates and the fully developed pattern begins to reform.

Generally there is good agreement between the predictions and the experimental data.

Additional data from the EPFL rig is reported in [1].

Test Data with Rotation

The data collected at UWS is being used to develop empirical correlations that account for the influence of rotation on heat transfer. Consequently, the relative Nusselt number, Nu/Nu_0 , is correlated as a function of Coriolis and Buoyancy numbers, ready for implementation into company design codes.

The data presented in Figure 9 illustrates the accuracy of these correlations. The data is for 45° staggered ribs (i.e. as with the EPFL baseline case, above) at the mid-rib location. It can be seen that the Nu/Nu_0 data is correlated within $\pm 15\%$. The accuracy of correlations at the rib location is equally as good.

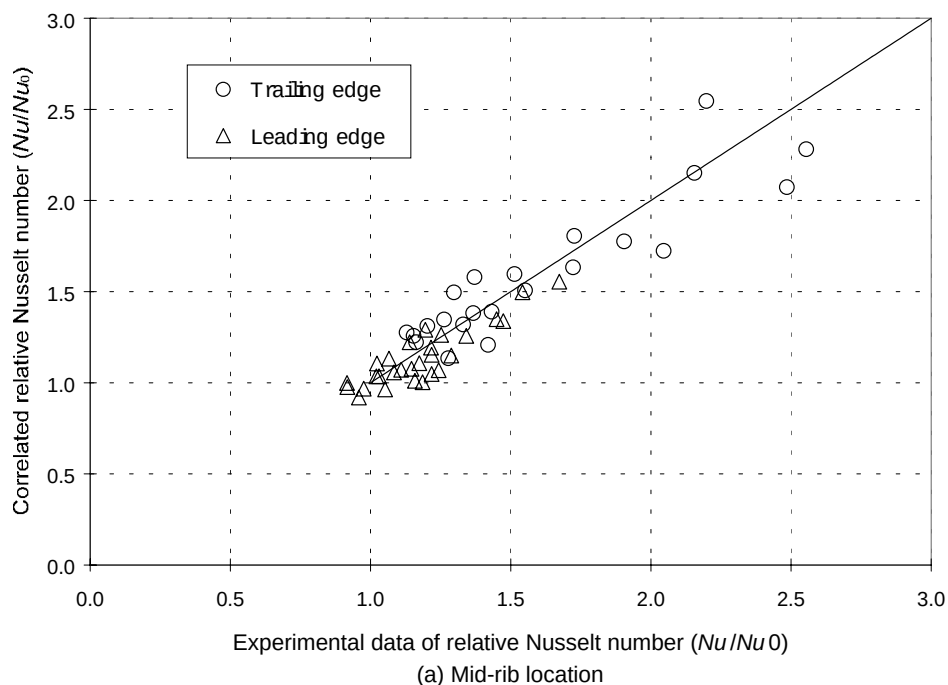


Figure 9: Comparison between measured and correlated relative Nusselt number values.

CONCLUSIONS

As stated earlier, the prime objective of this Brite-EuRam project is to deliver an integrated research package for the development and validation of numerical methods for the design and off-design analysis of turbine blade cooling systems.

Experiments on realistic two-pass geometries without rotation are underway at EPFL. This data has been used to validate and improve CFD codes, leading to the formulation of an approved methodology for code utilisation under static conditions. Testing of idealised geometries with rotation at TUD and UWS have also been used for code validation purposes. Following validation against this information, data for the engine representative geometry with the correct non-dimensional levels will be available for the final stage of code validation. It is worth noting that modelling of the engine configuration has already led to improvements in CAD and girding issues. Furthermore, data from the static and rotating rigs is being used to develop correlations for implementation into the partner companies' design suites.

The experience and expertise gained during the ICTB project places the consortium in an excellent position to do further work in this critical area. Additional partners are keen to get involved with the research, where it is hoped that the database of test cases can be extended in order to further increase understanding and, ultimately, design code accuracy.

ACKNOWLEDGEMENTS

This work has been carried out with the support of the European Commission through the Brite-EuRam 'Internal Cooling of Turbine Blades' research program. The authors would also like to thank the multitude of personnel at each of the establishments involved who have helped with this research.

REFERENCES

- [1] Chanteloup, D., and Bölcs, A. (2001), *PIV Investigation of the Flow Characteristics in 2-leg Internal Coolant Passages of Gas Turbine Airfoils*, Submitted to the Fourth European Conference on Turbomachinery Fluid Dynamics and Thermodynamics, 20th-23rd March 2001, Florence, Italy.
- [2] Rathjen, L., Hennecke, D.K., Elfert, M., Bock, S., and Henrich, E. (1999), *Investigation of Fluid Flow, Heat Transfer and Pressure Loss in a Rotating Multi-Pass Coolant Channel with an Engine-Near Geometry*, Proceedings of the Fourteenth International Symposium on Air Breathing Engines, 5th-10th September 1999, Florence, Italy, Paper No. 99-7201.