FRONT PAGE

PROJECT FINAL REPORT

Grant Agreement number: 233799

Project acronym: ERICKA

Project title: Engine Representative Internal Cooling Knowledge and Applications

Funding Scheme: Collaborative Projects

Period covered: all periods from 01/07/2009 to 30/06/2014

Dr Cristian Orozco-Piñeiro, Rolls Royce:

Tel: +44 (0) 1332 70 (5)8733

Fax: +44 (0)1332 26 (5)1319

E-mail: Cristian.OrozcoPineiro@Rolls-Royce.com

Project website address: http://www.ericka.eu/

1 Final publishable summary report

1.1 Executive Summary

The Engine Representative Internal Cooling Knowledge and Applications (ERICKA) project has ended within its allotted time and has met its objectives in a successful manner within reason. On the one hand, the project has contributed to the improvement of the technologies and the understanding involved in the design of efficient blade internal cooling schemes. This represents a direct contribution to the reduction of fuel consumption through better engine efficiency, and the reduction of engine emissions. On the other, ERICKA has also indirectly contributed to the further development of the European aerospace industry through the establishment of new collaborative research links, acting as an ideal platform to educate students and to train staff, and by extensively disseminating its results. The complete benefits of ERICKA will become obvious in the next few years.

Within WP0, Rolls-Royce UK has led the project with the support of the ERICKA Project Office and appropriate approval from the Management and Steering Committee. Although the project needed a one year extension in order to complete all the work it had planned it managed to finalise the testing and CFD work it set out to do.

WP1 involved all of the SMEs who were tasked with optimising specific cooling technologies. The ERICKA project enabled the four SMEs involved to develop their simulation, design and optimisation software packages. New features have been integrated into the software developed by the partners and validated to robustly and efficiently parameterise, mesh, compute and optimise the different cooling configurations of the ERICKA project. Those software are now much more accurate than at the beginning of the project and the results of the ERICKA project demonstrated their efficiency to simulate and optimise the various internal cooling systems .All WP1 partners used a surrogate based optimisation strategy coupled to a genetic algorithm to efficiently deal with the computationally expensive simulations of the workflow. The objectives and constraints of the optimisations were defined for each case in agreement with the industrial partners. During the life of the project, this work package involved significant interaction between the WP1 leader and all of the industrial partners engaged in the experimental Work Packages – specifically WP2, WP3 and WP4.

Experimental workpackages 2, 3 and 4 addressed the evaluation and improvement of the cooling performance of a variety of turbine blade cooling systems, encompassing leading edge impingement systems and ribbed radial and U-bend internal cooling passages. Measurements of heat transfer and pressure loss in non-rotating and rotating test rigs were performed in order to investigate the influence of different conditions and geometrical configurations, together with the rotation effects on systems representative of current and future engines. All workpackages have produced a vast experimental database from which partners have benefited during the lifespan of the project and will continue to do so after the end of the project. Certainly, partners have benefited in a variety of manners, from the derivation of new heat transfer correlations of direct use for the design of internal cooling system of current and future engines, to a deeper understanding of the physical phenomena involved which directly affect the accuracy of the results obtained. This, in turn, has also lead to the improvement/development of to the experimental techniques adopted during the project.

Most of the partners involved in the project were also involved in WP5 which was dedicated to the evaluation of the performance of Computational Fluid Dynamics (CFD). Each of the 12 partners involved has

performed several computations, relying on a variety of both commercial and in-house developed CFD tools, and encompassing both standard and advanced turbulence models. WP5 has also gathered and disseminated the acquired lessons learnt within all partners. Beyond the obvious benefit associated with a deeper understanding obtained on the performance of CFD approaches considered here, and the identification of new challenges and therefore future lines of research, work conducted within ERICKA has exceeded the framework of WP5 itself, contributing to the definition of the CFD best practices adopted within WP1, and giving an insight into the physical phenomena involved within the configurations covered by the project. This has lead directly to improvements of the measurement techniques employed in the experimental workpackages 2, 3 and 4.

The final impact of ERICKA is expected to continue increasing well after the end of the project, as the vast experimental database created by the project continues to be exploited for the validation of CFD, and the improvements derived from improved CFD strategies increases the accuracy of the optimisation activities associated to the cooling system design of the current and future engines.

1.2 Project Context and objectives

The fuel efficiency of a gas turbine used for aircraft propulsion is dependent on the performance of many key engine components. One of the most important is the turbine whose efficiency has a large influence on the engine fuel consumption and hence its carbon dioxide emissions.

The high-pressure (HP) turbine stage must operate at high efficiency in the most hostile environment in the engine. The turbine aerofoils and all of the gas swept parts are subject to the engine's most aggressive heat loads as the working fluid supplied to this stage is at the peak cycle temperature and the work generation process in the turbine accelerates the flow to high speeds which results in enormous heat fluxes to component surfaces.

The design of turbine cooling systems remains one of the most challenging processes in engine development. Modern high-pressure turbine cooling systems invariably combine internal convection cooling with external film cooling in complex flow systems whose individual features interact in complex ways. The heat transfer and cooling processes active are at the limit of current understanding and engine designers rely heavily on empirical tools and engineering judgement to produce new designs. These designs are developed in the context of continuously increasing turbine entry temperatures as the latter leads to improvements in fuel efficiency, and increases in specific work.

Similarly, the stage efficiency of the low-pressure (LP) turbine also has a large impact on the performance of the engine. The cooling flows used in LP blades are subject to complex buoyancy and forced convection interactions which make LP cooling flow prediction and system design very demanding.

ERICKA intended to face the challenge of **reducing aeroengine carbon emissions by improving engine efficiency by better turbine cooling**. It is important to note that the technology from other aeroengine research and development programmes fed into the efficiency improvements of future engine. The goal of ERICKA is to directly contribute to reductions in aircraft engine fuel consumption with **a direct targeted contribution of 1% reduction in SFC relative to engines currently in service**. ERICKA aimed at providing a means of improving blade cooling technology reducing the cooling mass-flow, which is directly translated in improved component efficiency.

ERICKA’s approach was to focus the turbine blade cooling technologies to be advanced into three separate experimental activities that include tests in stationary and rotating facilities. The work covered supported, and was supported, by computational work performed using CFD analysis, and by design optimisation. The classes of cooling technologies covered in the three experimental activities thus set the context for most of ERICKA’s research. This blade technology focussed approach has been adopted to ensure that the results of ERICKA will be directly applicable to future gas turbines.

ERICKA’s partners incorporated most of the major European aeroengine manufacturers. The partners share the aim of dramatically advancing the technology used in the design of gas turbine cooling systems. The combination of experimental work with CFD studies and design optimisation activity is judged to be the strategy that would most likely to make a significant impact on blade cooling technology within a 4/5-year programme.

The experimental data gathered will:-

1. Enable the aeroengine companies to develop and validate their design methods using the results gathered from a combination of rotating and static experiments. In particular, the experimental Heat Transfer Coefficient (HTC) results from the rotating facilities are a major attraction to all of the industrial partners as the implementation of high resolution, heat transfer experiments under engine representative conditions is prohibitively expensive. The support of these experiments by the project represents the ideal way for Europe to capitalise on its test facility assets and its experimental expertise.
2. Provide data to optimise cooling designs. The data will be given to four SMEs engaged in the development of software for optimising complex thermal and fluid systems. ERICKA includes activity to enable these companies to develop their techniques in the context of turbine cooling. This code development activity includes extensive validation of their methods using the experimental data, and a set of optimisation challenges which will lead to dramatic improvements in turbine component performance.

The project aimed to evaluate experimentally new cooling geometries developed by the SME optimisers and to make the best use of the experimental data to be acquired from the rigs. Since the early part of the project involved code development and calibration of the SMEs optimising codes, this target was ambitious. The rotating rigs are major facilities and require considerable time to design, manufacture, instrument and commission models. For these reasons, the partners decided to evaluate the optimised geometries in static test facilities in each experimental workpackage. It should be noted that the understanding of the effect of rotation on the flow and heat transfer gained from the earlier rotating experiments and CFD would enable the performance of the new cooling geometries when rotating to be appraised.

The main objective of ERICKA was to r**educe CO_2 emissions by 1% compared to year 2000 reference engines**. The project would also significantly advance **blade cooling technology** which allowed the following supporting objectives to be included:

- The new heat transfer data and computer modelling strategies will enable turbines to be designed to cope with low NO_X combustors.
- The improved modelling and computer methods will reduce the time required to design the turbine by 20%. This will reduce engine time to market and engine cost significantly.

ERICKA’s detailed scientific and technological objectives are listed below: -

1. **Acquisition of engine representative Rotating HTC data**. The project included the measurement of world-class, high resolution heat transfer coefficient (HTC) data in two major **rotating test facilities** for the validation of the cooling system design methods of the partner aeroengine companies. Both the RHR and BATHIRE research rigs are costly to operate and are not routinely used for cooling system design or methods development. Earlier data from these and other rotating rigs has shown that rotation can significantly alter the internal HTC distributions, and hence cooling system performance. The methodologies commonly used in present blade designs use empirical factors derived from sparse test data to account for rotation.

   ERICKA will develop computational methods which will allow for the effect of rotation in a broad range of geometries specified in the WP descriptions below. This approach was used with success in the ICTB project for the class of cooling system with fixed aspect ratio, radial passages. The effect of rotation on internal HTC varies dependent on the cooling geometry and conditions but a working value of ±30% is realistic – see for example, Lees et al. (1999) which reports correlations for the effect of rotation resulting from ICTB. The better understanding and modelling resulting from ERICKA will enable the effect of rotation to be exploited to improve cooling and hence turbine efficiency. In addition, the improved accuracy of simulation will enable engine designs to be used which are inherently better. This is because the usual means of attending to a blade which is running hot in a particular location, following a shortcoming in the prediction methods, is to include extra rows of film cooling holes. The latter often add considerable loss to the turbine stage and reduce engine efficiency. An alternative fix is to the increase the local cooling flow which has the same adverse effect on efficiency.

2. **Development of optimising software for internal cooling**. Based on the measurements made in (1), ERICKA aims to develop the computer codes used by sector-leading SMEs to enable the latter's simulation strategies to be applied to the problem of finding optimum cooling passage designs for HP and LP turbines. There are currently no commercially available codes that can be used to optimise internal cooling passages for the range of cooling geometries to be investigated by ERICKA. The availability of affordable, increasingly powerful computers in the design office together with better software means that the cooling system design problem will eventually be tackled using computer optimisation strategies.

   ERICKA aims to accelerate this process in Europe by supporting the development of design software specifically for cooling system design. The optimising strategies typically integrate geometry management, meshing and CFD solvers with search routines. The adaptation of this strategy to cooling geometries will be a significant step forward in cooling system technology. Another more unusual approach will also be studied as the code used by one of the SMEs (partner 9, CFS) is a novel departure from the above.

3. **Creation of a database of detailed HTC data from a broad range of stationary internal cooling geometries**. ERICKA involves the application of the expertise of a set of leading European universities to the measurement of high resolution, HTC and flow data for static models of benchmark and optimised geometries. These results will supplement the data acquired by the two major rotating facilities. They will significantly improve the accuracy of computer simulations used in aeroengine design as the university tests cover a wider range of conditions and geometries than can be tested in the rotating rigs. It should be noted that the focus of the project is to investigate cooling ducts with Engine Representative geometries for which there are relative few reports in the literature. The project includes activity to develop the experimental methods which will improve European capability in the aerospace sector. The Consortium will share the experimental data to maximise the impact of this research on the European aerospace sector.

4. **Development of the CFD strategies used by the partners to predict internal flows**. ERICKA includes extensive CFD activity that will allow the engine companies to enhance their own predictive codes and computer modelling strategies. Importantly, all of the engine companies will take part in the CFD work-package. Their work will be complemented by studies performed by experts in flow and heat transfer simulation at partner 14, UPM, and partner 12, ONERA. The carefully planned coordination of activity here will ensure that key knowledge and expertise developed in the project will be shared across the Consortium
5. **Experimental evaluation of optimiser codes**. ERICKA includes three experimental activities which each focus on specific cooling technologies:
   - Impingement cooling suitable for a blade leading edge
   - Radial passages of varying aspect ratio
   - Radial passages combined with a u-bend

   ERICKA incorporates activity in which, at least, one optimised version of the specific cooling theme is produced and rig tested. The new designs will evolve using contributions from the engine companies, understanding of flow physics derived from both the university experiments and the CFD predictions, together with the results of automatic optimisation strategies introduced by the SMEs. This evaluation of the results of the optimisation programmes will ensure that the SME codes are used by the gas turbine community in Europe.
6. **Development of cooling system design methods suitable for future low emission and green fuel combustors**. The turbine designer of future low emission engine will be challenged to develop blade cooling systems that can cope with the more uniform combustor exit temperature profile produced by low NOx combustors. The departure from traditional combustor exit profiles means that design methods must be in place to cope with these new thermal boundary conditions. The ability to design for conditions relies on methodologies that are based on proper representation of the flow and thermal physics and not empirical interpretation of engine test data.

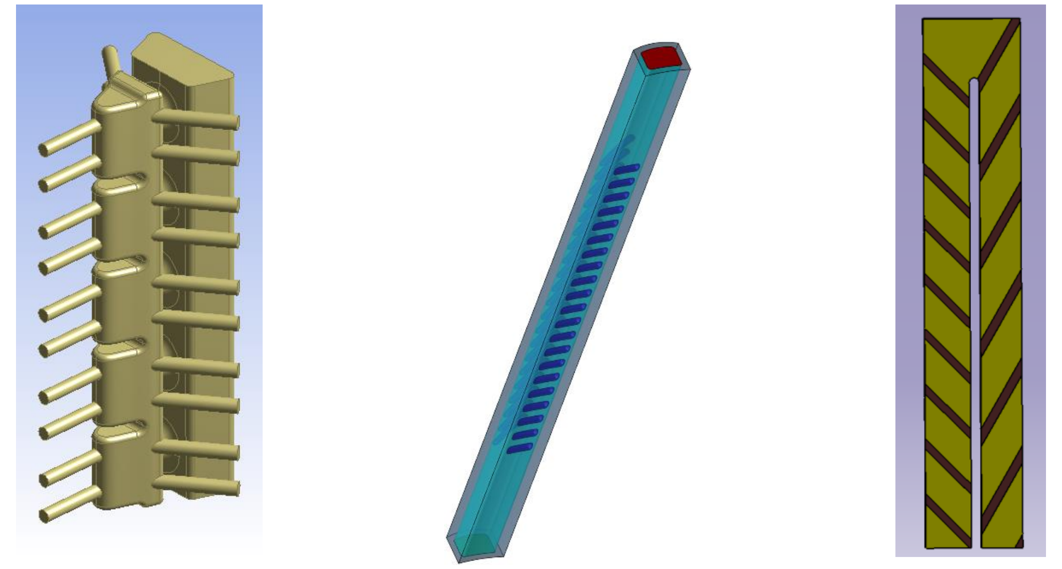
   ERICKA directly addresses this by a programme that combines design method development with key experiments in static and rotating rigs. It is also notable that the possible replacement of aviation fuel by green alternatives (liquid hydrogen, 2nd generation bio-fuels, etc.) could also lead to departures from the conventional combustor temperature profile. The advanced design methodologies developed in ERICKA will be well suited to these future design challenges.

7. Finally, the **dissemination and exploitation** objective will be for the industrial partners to implement the improved cooling systems in future aeroengines and disseminate in the supply chain and ensure that the simulation and optimiser software are fully industrialised. Communication to the European and global aerospace sector through technical publications of the results generated by the experiments and by the computer predictions will also be an important aspect of the ERICKA dissemination objectives to further support new research work in the area.

1.3 S&T results/foregrounds

WP1 – Optimisation of turbine cooling systems components

The objective of this work package was to implement and validate new integrated software strategies and optimisation methodologies, to improve the thermal behaviour of turbine blades by optimising internal cooling component. In this project three types of internal cooling systems depicted in Figure 1 were investigated. A matrix of cases is provided in Table 1.



(a) – Leading edge impingement cooling system

(b) – radial passage cooling system

(c) – U-bend radial passage cooling system

Figure 1: Cooling systems configurations optimised in WP1

WP	Design Owner	Test description	CENAERO	CFS	EnginSoft	NUMECA
2	AVIO	LE impingement			DONE	
2	ALSTOM	LE impingement		DONE		
3	RRUK	Mid-chord radial passage				DONE
3	ITP	Mid-chord radial passage				DONE
4	SNECMA	U-bend HP blade	DONE			DONE
4	MTU	U-bend LP blade	DONE			

Table 1: Matrix of Cases

The four SMEs involved in WP1 succeeded in designing a robust workflow composed of a parametric modelling step, a meshing step, a CFD computing step and a post-processing step. They all used a surrogate based optimisation strategy coupled to a genetic algorithm to efficiently deal with the computationally expensive simulations of the workflow. The objectives and constraints of the optimisations were defined for each case in agreement with the industrial partners. In most of the cases, the objectives were to maximise the heat transfer coefficient of the internal walls while minimising the pressure drop across the internal cooling system. These two objective are clearly in tension with each other and consequently a compromise solution had to be sought. To achieve this, two strategies were employed by the partners. Numeca and EnginSoft used a multi-objective optimisation approach, whereas CFS and Cenaero preferred searching a single objective while constraining the other objectives, which is less expensive in terms of optimisation effort. At the end, selecting a compromise inevitably involved making a choice on some form of weighting between the goals, whether it be before the optimisation in the case of single objective strategy or after for a multiple objectives strategy. The numerical setup used by each partner for the 3D Reynolds averaged Navier-Stokes (RANS) computations was adapted to the optimised configuration and to the simulation code which had been validated against experimental data prior running the optimisation. Numeca used a conjugate heat transfer (CHT) approach to account for the heat transfer in the solid also. All the optimisations resulted in improved internal cooling geometries able to better cool the blade without increase of the pressure drop. Besides all the partners gained experience in optimisation and modelling of heat transfer systems and had the chance to develop their in-house software and to validate it on such applications.

EnginSoft

The object of the optimisation campaign performed by EST in WP1 was the leading edge impingement concept developed by AVIO. Starting form an existing baseline configuration, EST created a full automatic procedure able to evaluate several different designs. Acting on 4 geometric input parameters (see Figure 2 and Figure 3 for reference), the objective of the optimisation, as agreed with AVIO, was to find the best possible solution in terms of heat transfer effectiveness, i.e. maximise the heat transfer with the same pressure loss respect to the baseline configuration.

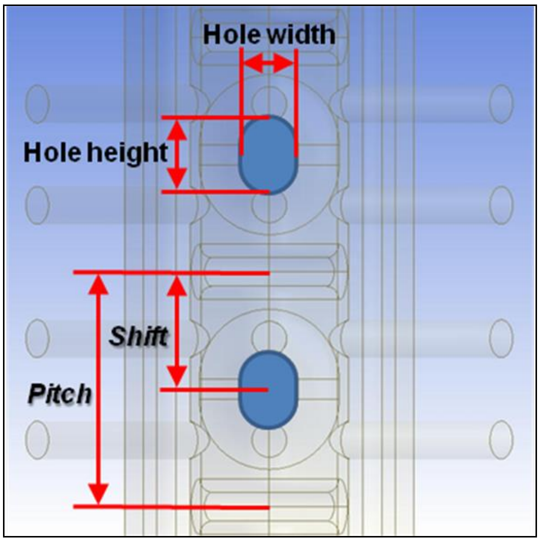


Figure 2: Geometric parameters (1)

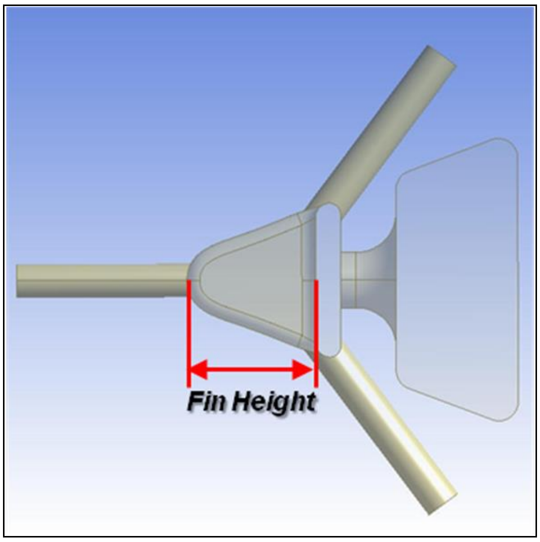


Figure 3: Geometric parameters (2)

Each design point was investigated through a CFD simulation. The numerical model was assembled by a sequence of simulation processes driven by ANSYS software:

- ANSYS DesignModeler ⇒ parametric definition of the geometry;
- ANSYS Meshing ⇒ unstructured mesh approach with high resolution of boundary layer;
- ANSYS CFX ⇒ CFD solver run and post-processing.

All these processes were integrated in the ANSYS Workbench, a platform which offers a comprehensive view of the complete analysis procedure, a simple way to pass data from one application to the other, and a powerful tool to manage the parameter set.

In turn, the resulting ANSYS Workbench process is integrated into a workflow under the control of the optimisation platform modeFRONTIER™.

The computational costs of each analysis were such to make unsuitable a classical approach to optimisation. A hybrid Response Surface Methodology (RSM) technique based on Neural Network (NN) algorithm was used for this reason.

Main results and lessons learnt of this hybrid optimisation approach are summarised as follows:

- CFD Validation

Different approaches for both thermal boundary conditions and turbulence models were considered in order to find the best possible compromise between numerical accuracy and efforts. In particular, Shear Stress Transport with Curvature Correction was chosen as turbulence model, in addition to a Low-Reynolds approach.

- Physical principle and optimisation

The physical principle exploited in this kind of cooling system is convective heat transfer to impinging jets, aiming to achieve high localized values of heat transfer coefficient. The cooling air jet impacts the turbine's internal walls at leading edge and it is deflected in all directions parallel to them. Given the particular shape of these surfaces, the diverting stream generates two main counter rotating vortical structures, which intensity then affects the heat removal potential. The goal of the optimisation campaign it is to find the best possible recipe for the input parameters that maximises the strength of these vortices.

- Importance of the 4 input variables

Only two input variables, pitch and shift, have a significant impact on the optimisation. In particular, there are:

- an inverse correlation between pitch and mass flow rate at inlet.
- an inverse correlation between shift and heat transfer.
- a direct correlation between shift and mass flow rate through the impingement holes.

The other two input variables, hole width and fin height, can be considered negligible from a statistical point of view.

- Rotational effects

The effects of the fictitious forces (centrifugal and Coriolis) gives no appreciable enhancements on the heat transfer behavior of the leading edge cooling system. The main reasons are the relative low rotational speed and the impingement of the jets, which generates strong vortical structures in the leading edge cavity compartments.

- Results of the optimization

The optimized design of the AVIO leading edge impingement concept give an increment of mean HTC by 27% with a reduction in terms of mass flow rate estimated in -12%, as the pressure loss was considered as a design constraint.

CFS

Cambridge Flow Solutions also performed the optimisation of an impingement cooling system in this work package. The geometry was provided by ALSTOM. During the geometric parameterisation task, CFS was able to demonstrate complete control over the parametric features of an internal cooling geometry in order to fully explore the design space for an arbitrary number of parameters. This approach is based on the simple geometric manipulation, addition and subtraction of STL (surface triangulation) facets which represent particular features and the enclosing envelope of the geometry. The power of this approach lies in the complete absence of a traditional solid model CAD kernel which is normally required to drive the geometric variation. Also the CFS method does not require a perfectly manifold representation of the geometry – rather intersections can be arbitrary and non-topologically bound – this greatly reduces the processor overhead required for calculating valid geometric candidates. This approach is a major step in increasing the autonomy of a fully automatic optimiser process and reduces the time required to generate candidate geometry to a few seconds. CFS strategy for the automated mesh generation task was based on BOXERMesh. CFS has independently developed a proprietary mesh generation system, BOXERMesh, which is ideally suited to the automatic generation of hex-dominant hybrid meshes on parametric geometry. BOXERMesh was shown to be able to produce solvable meshes for 100% of design candidates across a range of different types of turbine internal cooling strategies (a total of 545 separate candidates). As BOXERMesh is a fully parallel application which can be fully controlled via a scripting language, the mesh generation process is fast, consistent and robust – three qualities which are absolutely vital to the successful operation of a fully automatic optimiser suite. This includes the vital step of inserting a viscous layer mesh of arbitrary thickness at specified surfaces. Besides CFS was able to show very good agreement between simulations using BOXERMesh and ANSYS CFX solver with published experimental data for impinging jets (ERCOFTAC case 25). CFS was also able to demonstrate an initial study of a full factorial Design of Experiment for the ALSTOM impingement cooling case containing 162 candidate geometries. Each candidate was successfully generated, meshed, solved and post-processed fully automatically with a total meshing time for all candidates of 21.5 hours and a total solve time of 27 days, generating 97.2 Gb of simulation data. The results showed a clear Pareto front payoff between cooling efficacy and total pressure loss, as expected. Eventually CFS undertook a full scale optimisation study on the ALST impingement cooling geometry, the baseline for which was tested by UNIFI. Based on the system created in T1.4, CFS was able to automatically generate, mesh, solve and post process all candidates without exception. This included several validation cases and a mesh sensitivity study. The optimisation process was conducted within the scope of the Minamo optimisation platform provided by the consortium partner Cenaero. The chosen strategy was an unconstrained single-objective surrogate-based optimisation over the design space, with the area-averaged Nusselt number on the target surface as the objective function, as agreed with ALSTOM. The actual optimisation consisted of close to 100 candidates. However the computational intensity and time required to generate each candidate point (around 11 hours) meant that the study had to be curtailed at a point where funding for resource ran out. Despite this limitation, CFS was able to successfully demonstrate the robustness, speed and power of their automatic optimisation system. The fluid dynamics and heat transfer phenomena involved meant that the problem which was to be solved required extreme care and attention to detail in the numerical simulation. This had an adverse effect on the number of candidates which could be finally included in the study.

NUMECA

The two radial cooling passages from ITP and RRUK have been optimised by NUMECA, along with a SNECMA U-Bend cooling passage. To optimise these three configurations, NUMECA first upgraded its in-house parametric blade modeler AutoBlade^{TM} with support for cooling features. The software can now handle realistic cooled blade channels including bends and ribs driven by a limited and dedicated set of simple geometrical parameters. The company also gained experience in the setup of Conjugate Heat Transfer (CHT) simulations from a sketched geometry to the post-processing by relying on its own tools for geometry creation, mesh generation, simulation and optimisation. In particular, the challenging optimisation problem of blade cooling requires local refinement and high quality viscous layers on geometries with highly variable geometric details that easily leads to failure of either the mesh process or the simulation itself. The use of the full hexahedral Hexpress^{TM} mesh generator allowed to achieve the quality of a structured mesh with the flexibility of the unstructured volume to surface method and increases the success rate in the optimisation process. NUMECA compared different turbulence models and highlighted the interest of CHT in the problem of cooling in its CFD solution FINE^{TM}/Open. In particular the SARC model has been successfully tested and the superiority of the anisotropic EARSM model has justified its use in the simulation of cooling under rotational effects. A fully scripted approach has been adopted and demonstrated in the optimisation suite FINE^{TM}/Design3D including a simple and efficient method to cope with a high amount of invalid samples. Working with accurate simulations, very large design spaces and multiple contradicting objectives also brought valuable experience to the company in high performance computers (clusters) and multi-objective optimisation where a right balance has to be found to take advantage of the available computational resources (up to 800 cores).

Three cases have been successfully optimised optimised using a methodology based on an artificial neuronal network for the construction of surrogate models coupled to a genetic algorithm. In agreement with the industrial partners (ITP, Rolls-Royce UK and Snecma), the main objectives are the maximisation of the mean Heat Transfer coefficient (HTC) over the pressure and suction sides and the minimisation of the pressure loss across the passage. Some geometrical constraints are also applied to avoid unrealistic geometries. All simulations involve CHT and rely on high quality meshes of solid and fluid domains containing between 5 and 25 million cells in total.

The enhanced design of RRUK simple ribbed radial channel leads to 10% increase of mean HTC on the suction and pressure sides together with a slight decrease of the pressure loss. Similarly, the optimised ITP case shows an increase of 19% of the mean HTC with no increase in pressure loss with modified rib arrays in a fixed passage cross section. The optimiser successfully manages the high failure rate of nearly 40% of this curved ribbed passage mainly due to some topological incompatibilities in the very large design space. Eventually, NUMECA was able to achieve a 12% increase in mean HTC on the ribbed sides of SNECMA two passages channel with U-bend. The design also decreases the HTC on the smooth sides with a similar pressure loss.

CENAERO

To continue with the optimisation of the U-Bend radial cooling passages, Cenaero was in charge of optimising a static low pressure turbine two-pass U-bend from MTU as well as a rotating high pressure turbine two-pass U-bend from Snecma. A fully automated surrogate-based design loop was successfully set up to optimise the cooling efficiency of these turbine blade internal passages. The design loop is managed by Minamo, Cenaero's in-house on-line optimisation platform. A genetic algorithm efficiently coupled to a radial basis functions network was exploited to perform both global optimisations. Besides, although the simulation chain was designed as robust as possible, it is inevitable that not all simulations provide reliable results. Thus, these optimisations took major advantage of Minamo's ability to handle simulation failures through the management of two separate surrogate models.

The simulation chain is composed of the following steps:

- CATIA v5 for a robust regeneration of the CAD model followed by the meshing of the surface with FMS.
- Simmetrix libraries for a rapid and robust generation of the unstructured mesh. Typical meshes for these optimisations contains between 5 and 40 million cells.
- ARGO, which is a massively parallel CFD solver developed at CENAERO, was used here for the steady state 3D Navier-Stokes computation. The code used with the Spalart-Allmaras turbulence model in its low-Reynolds-number formulation and has been validated with CFD and experimental results available from the partners and in the literature. This project enabled CENAERO to gain expertise in aerothermal simulations and demonstrates the ability of ARGO in this field. Post-processing was performed using ARGO's outputs and a set of Python scripts.

Each ribbed row has been parameterised independently, leading to a design space of 20 parameters for the MTU case, allowing the modification of the rib shape and arraying with a fixed passage shape. For the SNECMA case, three additional parameters allowed for the modification of the passage shape.

Both optimisations led to promising predicted gains demonstrating the possibility of drastically decreasing turbine blade metal temperature.

The MTU geometry was first optimised, setting the HTC as objective and the other cost functions (head loss, uniformity of the HTC field, intensity of the recirculation downstream of the bend) as constraints in agreement with MTU. This optimisation achieved encouraging HTC gains directly associated with head loss penalty still below the constraint upper bound. The optimised geometry presented in Figure 4 increases by 15% the amount of energy extracted from the hot blade. Based on these results, the optimisation specifications have been refined for the SNECMA configuration. In particular, heat transfer occurring at the ribs were taken into account and three parameters were added to modify the internal cooling passage shape. The high predicted gains obtained for the SNECMA case were correlated to the variation of internal cooling passage leg aspect ratios, leading to variations of the Reynolds Number. Therefore, an additional optimisation aimed at maximising the surface cooling performance coefficient based on a Nusselt number ratio has been carried out to handle varying Reynolds Number. In agreement with Snecma, the other cost functions such as the head loss and the intensity of the recirculation have been set as constraints. This optimisation generated an optimised geometry able to increase by 15% the amount of energy extracted from the hot blade while keeping the pressure losses equal to the baseline value.

Many parameters in both MTU static and SNECMA rotating optimisations have the same impact on the cost functions. For example, rib width and rib height both contribute to the increase of heat exchange between the coolant and the blade as well as the head loss. An analysis of variance (ANOVA) comparison between MTU and SNECMA cases indicates that the design parameters of one side of a given leg impacts HTC on the opposite surface for the MTU case. This was not identified for the SNECMA case because of a higher rib height to leg width ratio and the rotation of the domain which projects most of the flow on the pressure side in the inlet leg and on the suction side in the outlet leg. As a result, for the rotating case, parameters of each side of the passage legs interact less with the parameters of the opposite side than for the static case.

Eventually, CENAERO identified a set of recommendations when optimising internal cooling passages without performing a CHT:

- Heat transfer at the ribs must be accounted for.
- When the Reynolds Number varies for a given optimisation, optimising Nu/Nu(Re) may be a good alternative to HTC.
- Both optimisations showed that it is critical to parameterise each rib row individually.

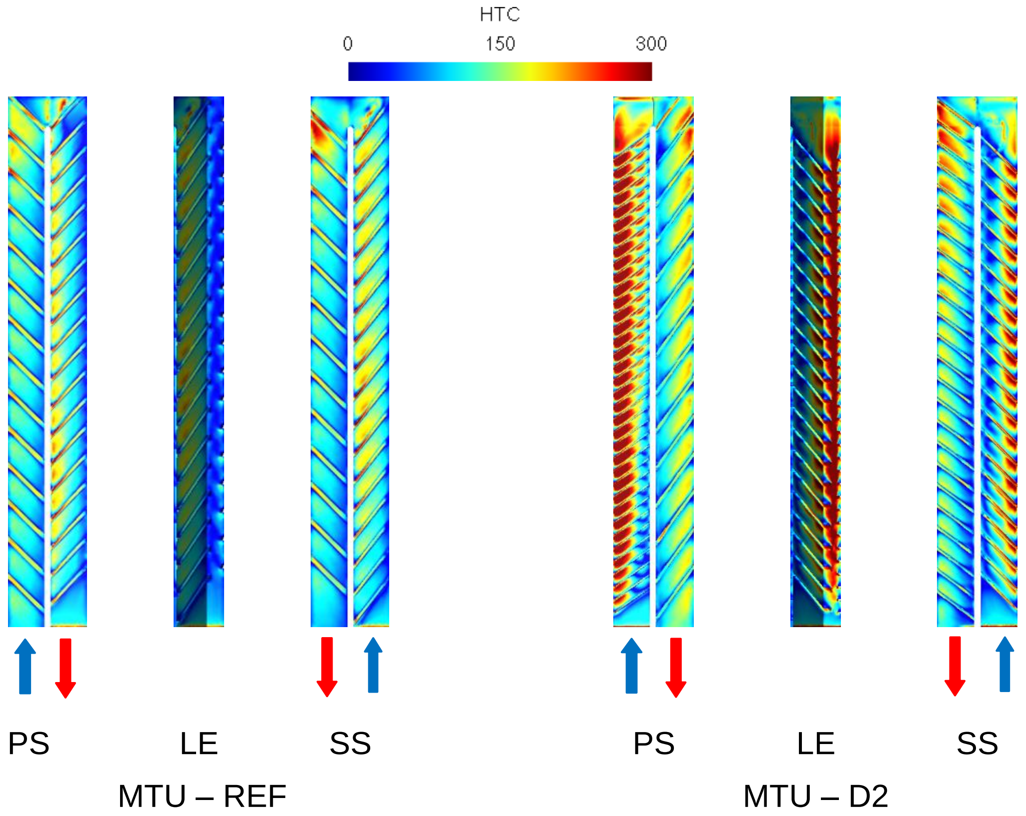


Figure 4: MTU baseline design (REF on the left) and one optimised design (D2 on the right)

WP2 – Lead edge impingement engine geometry (RHR)

WP2 focused on the evaluation and improvement of the cooling performance of impingement systems applied to the leading edge of turbine blades, see Figure 5. Measurements of heat transfer in non-rotating and rotating test rigs were performed in order to investigate the influence of different geometrical configurations and the rotation effects on impingement systems representative of current and future engines. These high quality results are of direct use to the manufacturers in engine design; for the validation of CFD activity (performed in WP5) and for validation of the optimisation code development (WP1).

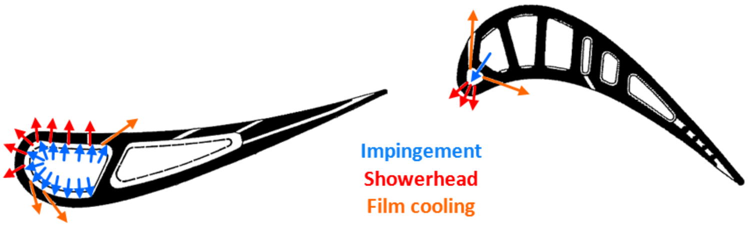


Figure 5: Representative cooling schemes for leading edge region of turbine aerofoils

WP2 addressed the lack of engine representative data for rotating impingement cooling for state of the art and future gas turbines. The heat transfer data was produced in both static and rotating test rigs. These combined experiments, together with computational studies in WP1 and WP5, are optimizing the turbine blade design of leading edge passages.
During the first 18 months of ERICKA, the test geometries for the Rotating Heat transfer Rig (RHR) and the Static Heat transfer Rig (SHR) were specified and designed using in-house codes to optimize the RHR and the SHR test sections. Technical drawings of one RHR test section were developed in close collaboration with RRUK. In addition, the positions of the pressure and temperature measurements for the test sections were specified and the test matrixes for the RHR and SHR have been agreed.

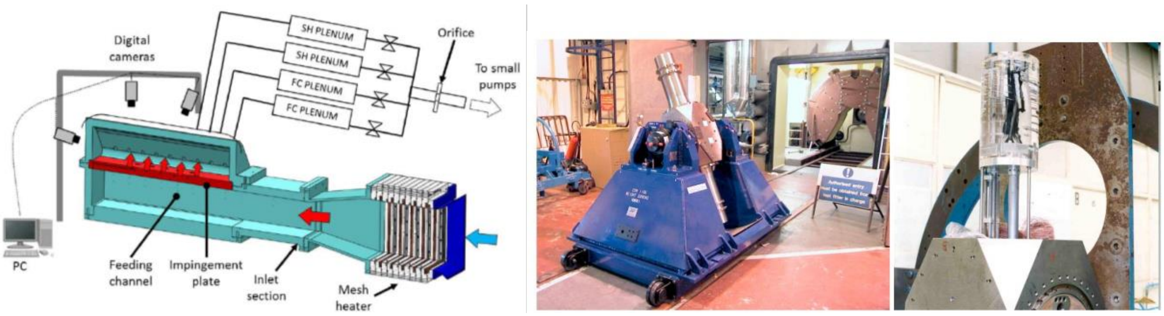


Figure 6: Static heat transfer rig (left) and rotating heat transfer rig (right)

SHR Tests (baseline geometries)
In the case of the baseline SHR, for ALSTOM twelve test sections (2 outer geometries and 12 different impingement plates) were designed and for Avio three test sections (with additional cross flow variants). The SHR model specifications were communicated to the University of Florence (UNIFI) who completed the 3D CAD models and the manufacturing of the models.
The SHR baseline experimental investigations have been performed on the static geometries to measure the internal Nusselt number in the ALSTOM and Avio leading edge cavities. Tests have been performed for varying jet Reynolds numbers (ALSTOM and Avio) and cross flow conditions (Avio) and in total 89 tests have

been performed. For these entire tests UNIFI reported local and area averaged Nusselt data for the wide variety of geometrical and flow parameter configurations of industrial interest. This resulted in a huge database, which can be used directly in component design process.

All investigated geometry parameter variations (e.g. leading edge shape variation or impingement hole shape variation) and flow parameter variations (e.g. jet Reynolds number or plenum cross flow) are at engine representative conditions and have been reported with availability to all partners. However, only some illustrative results are shown here.

One main result for the ALSTOM SHR geometries revealed a strong influence of the heat transfer values on the tangential pitch, as shown in Figure 7. The heat transfer for the case with the higher tangential pitch of $S_x/d=5$ significantly increases compared to the reference case with $S_x/d=4$.
Another relevant result for the ALSTOM baseline SHR experiments is that the radial pitch has no strong effect on the average heat transfer values, see Figure 8. However, some local differences can be observed since the relative position between impingent and extraction holes shifts.

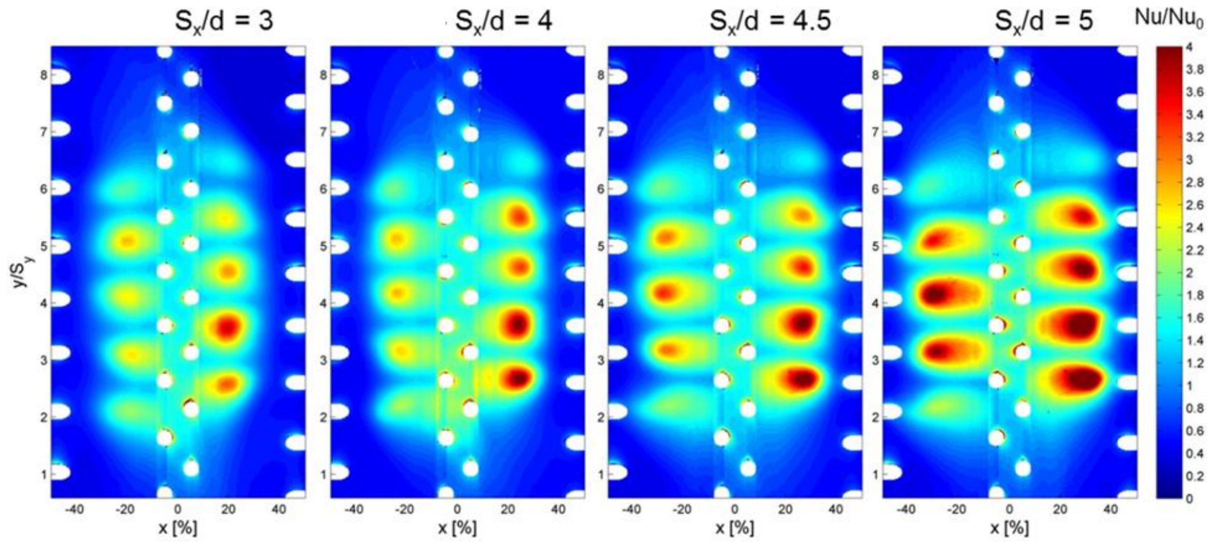


Figure 7: SHR – ALSTOM – tangential pitch (S_x) variation with fixed radial pitch ($S_y/d = 3$)

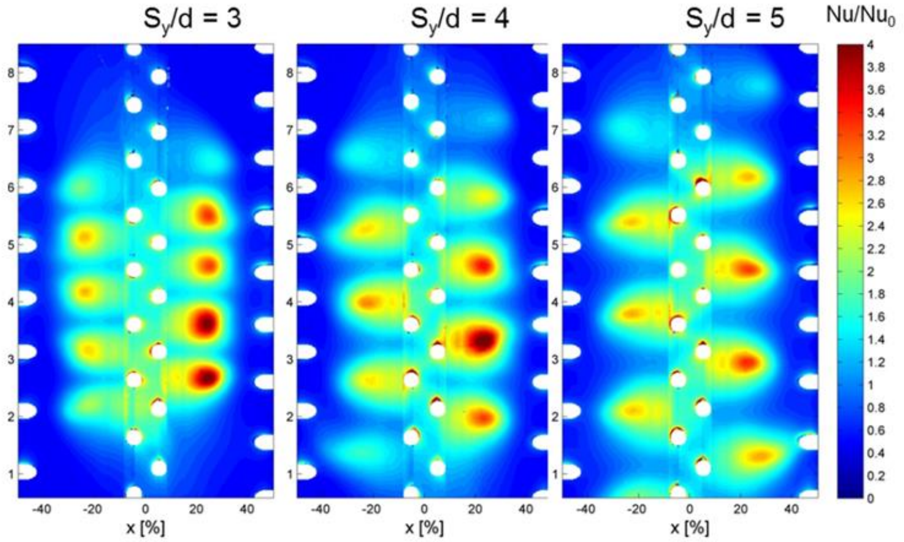


Figure 8: SHR – ALSTOM – radial pitch (S_y) variation with fixed tangential pitch ($S_x/d = 4$)

For the AVIO baseline geometry, results revealed that the Nusselt peaks are located in four lobes, with two peaks on the lateral walls, the leading edge opening and impingement jet spreading, and two peaks at showerhead extraction holes. The Nusselt contours show are asymmetric with higher Nusselt peaks on pressure side and higher Nusselt on the film cooling extraction on the suction side In addition, results showed that as the external cross flow decreases the heat transfer near the shower head extraction increases together, with a slight shift of the jet stagnation region (black and red line versus green in Figure 9).

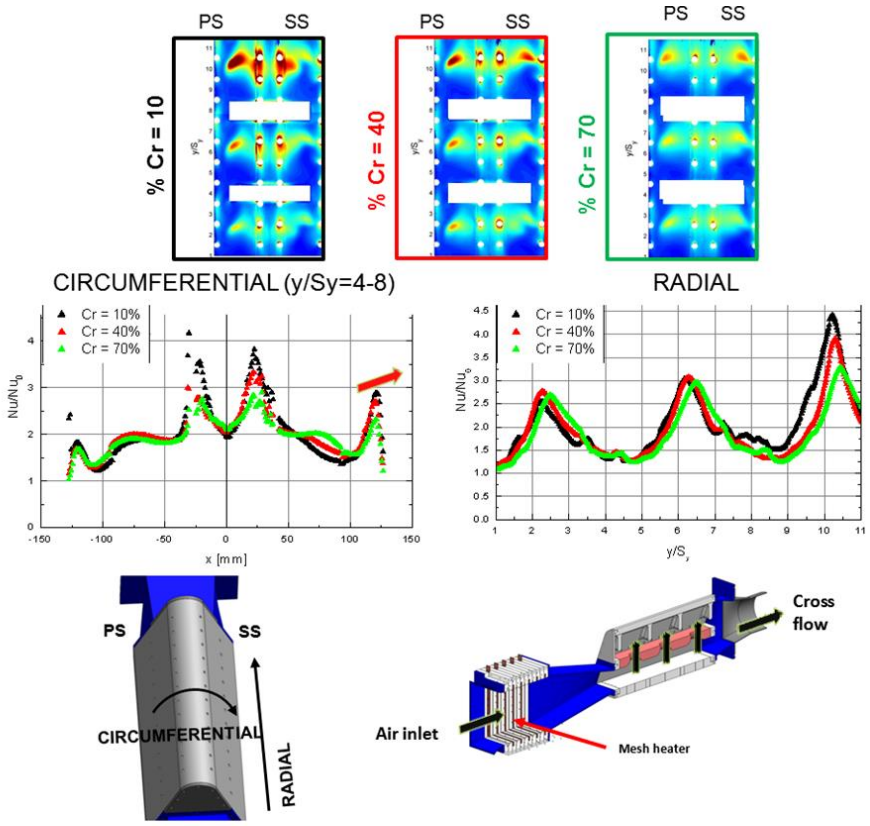


Figure 9: SHR – AVIO – cross flow variation

SHR Tests (optimized geometries)

Based on the findings of the SHR baseline geometries the optimized geometries have been generated for Task 2.3.Thanks to the modular approach of the test article, different setups were agreed with UNIFI to explore not only one but two different optimized geometries for each ALSTOM and Avio. According to experimental results shown on Avio C3 baseline, a shift of the jet holes was proposed. Moreover, to further explore possible optimization, a shift in fin location was experimentally investigated. For ALSTOM’s optimized geometries the configuration changes in impingement hole position (tangential and radial pitch), jet hole angle as well as the hole shape (from circular to race-track, constant area and converging area), according to WP2 experimental and WP5 numerical results.

The results in Figure 10 for the optimized ALSTOM geometries show that jets inclination together with racetrack shape (G105 vs. G090) leads to a more uniform Nusselt number distribution. However, the area average heat transfer is not affected. A converging hole shape (G100c) leads to an increase of both peak Nusselt values and to an extension of the high heat transfer region up to the showerhead holes.

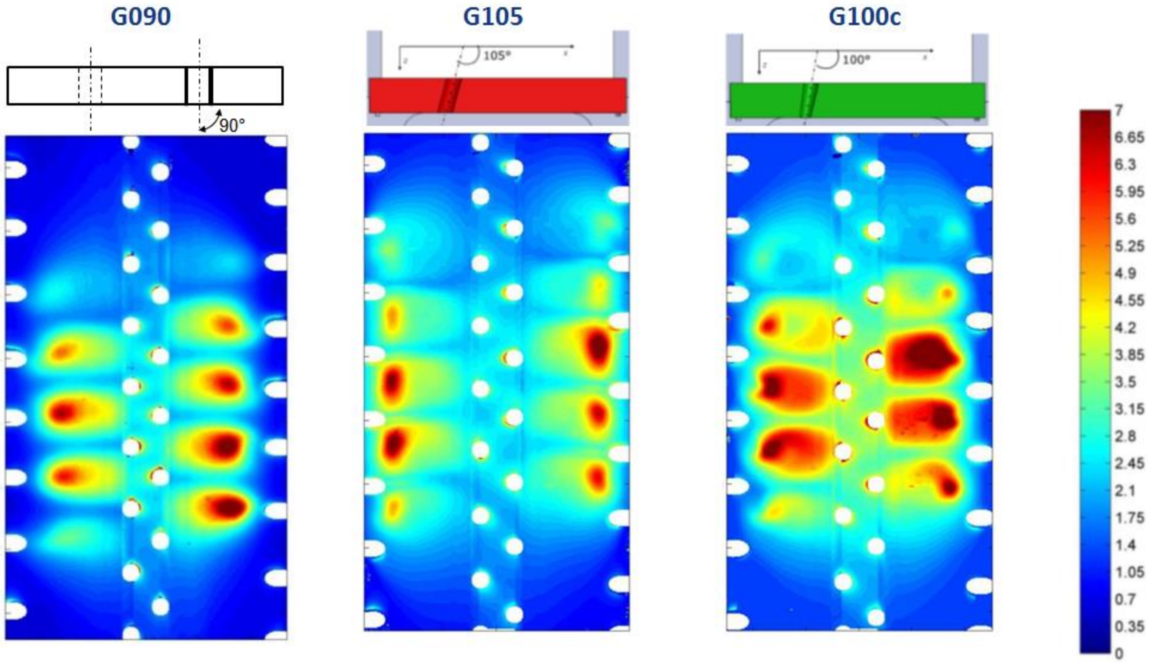


Figure 10: SHR – ALSTOM – optimised geometries with fixed radial pitch (S_y/d=3) and tangential pitch (S_x/d = 4) and varying hole shape (circular, race-track, conical race-track) and inclination.

The results in Figure 11 for the optimized Avio geometries show for the fin shift geometry that no area with concentrated Nusselt number enhancement can be highlighted, this means that the jet stagnation region is mainly located on the fins. However, it is interesting to note that downstream the fins there is a uniform increase in the Nusselt number, resulting in a lower thermal gradient along the radial and tangential direction of the airfoil. Moreover, it is important to note that all the reported results do not consider the heat removed by forced convection from the fins that might be not a negligible contribution, especially in the fin shift case where part of the jet stagnation region is located on the fin surface. For the second optimized geometry, with a hole shift it can be observed that the cross flow condition in the feeding channel leads to an offset between the center of the cold bridge holes and the jet stagnation region, as already seen in the baseline results, but in the present case the location of the stagnation region is located exactly in the center of the impingement module. This evidence affects the stagnation region shape that appears almost circular with higher values in the central curved region.

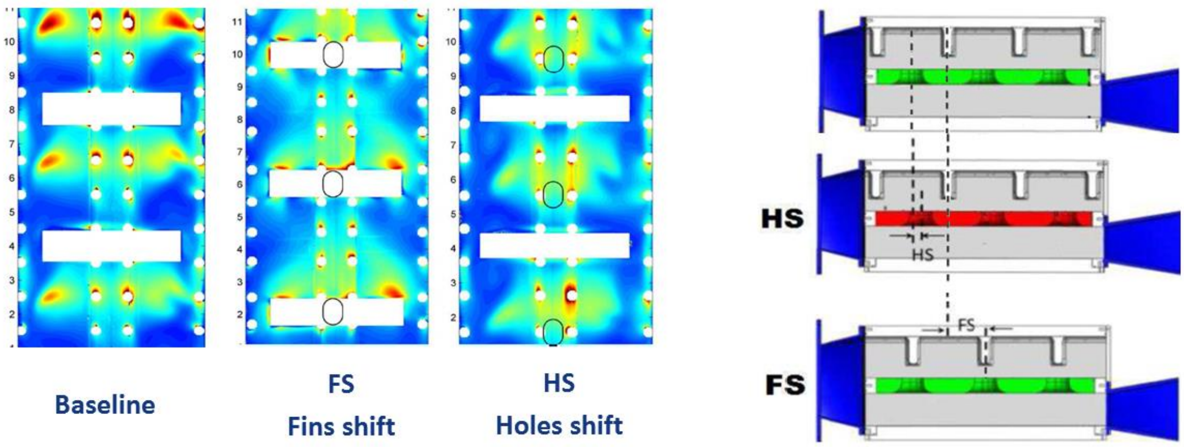


Figure 11: SHR – Avio – optimised geometries with fine and hole shift

RHR Tests

The RHR test campaign was done in Task 2.4 and in total 17 test points for each geometry, both Avio and ALSTOM, were performed. For the RHR tests the two effects of I) jet Reynolds number variation at different Rotation numbers as well as II) Rotation number variation at different jet Reynolds number could be investigated. The investigated Reynolds numbers and Rotation numbers are representative for engine applications. The RHR results are reported by RRUK in terms of local 2D heat transfer coefficient maps, of which ALSTOM and UNIFI (for the Avio geometry) performed area and spanwise averaged post-processing as well as a comparison of local heat transfer data for different test conditions. Furthermore the rotating data is compared to non-rotating data and is available in the reports for all partners.

For the ALSTOM RHR geometry it could be found that for low rotation speed the heat transfer was not affected by rotation, see Figure 12. Please note that for Figure 12 and Figure 13 the heat transfer is normalized to exclude the Reynolds number effect. However, for higher rotation speeds the heat transfer on the pressure side increases, whereas for the suction side the heat transfer slightly decreases, as shown in Figure 12.

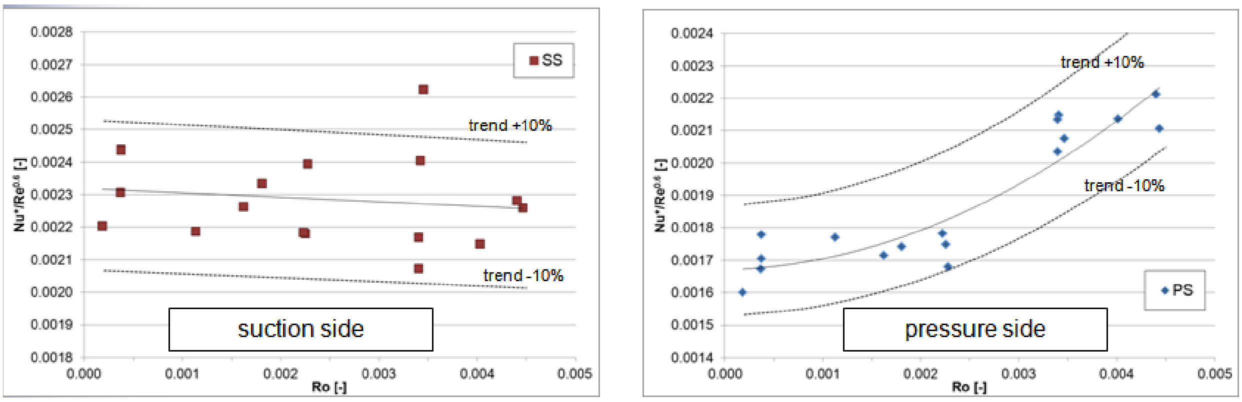


Figure 12: RHR – ALSTOM – effect of rotation on area averaged heat transfer

For the Avio RHR geometry in general, the agreement between SHR and RHR in terms of average Nu values, span-wise averaged results and distributions is quite good, and the effects of rotation are very slight, see Figure 13. For the PS an increasing Rotation number results in a slight decrease of Nusselt number, whereas for the SS an increasing the Rotation number results in a slight increase of Nusselt number.

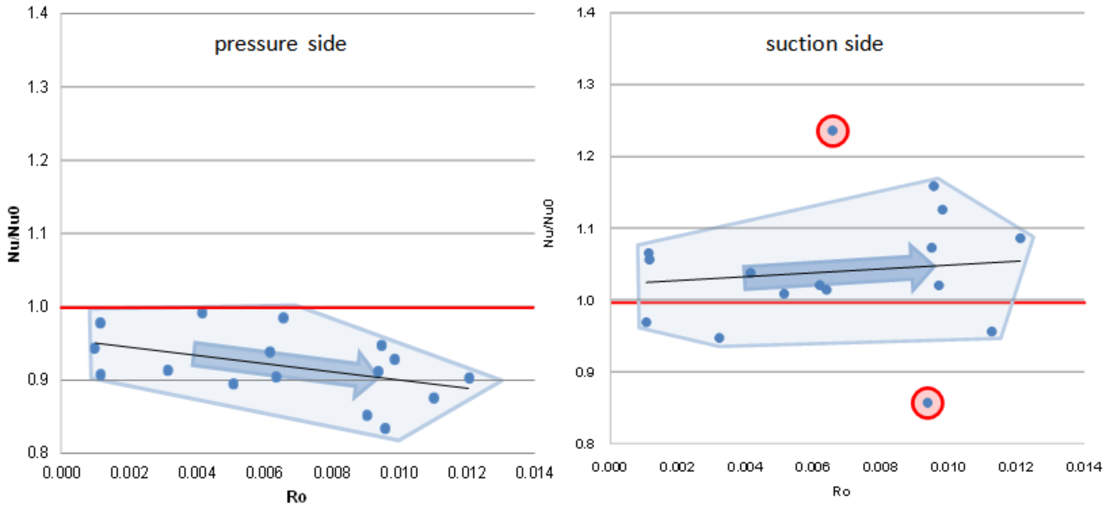


Figure 13: RHR – AVIO – effect of rotation on area averaged heat transfer

WP3 – Radial Passage engine geometry

WP3 was conceived to generate data on heat transfer and flow losses for engine representative radially aligned passages. Two main experimentation threads were followed:

1. Static tests: These have the advantage of well-defined boundary conditions, good instrumentation coverage and access. Such tests lend themselves to optimisation of the instrumentation and of the rapid modification of cooling channels for parametric studies of the effect of specific features. Additionally, static facilities experience less restrictions in terms of size and are therefore more amenable for measurements at high resolution.
2. Rotating tests: These experiments provide a close view of the physics of a cooling flow in a rotating component. The complications involved in obtaining engine equivalent conditions and engineering on board instrumentation result in compromises in terms of the scope of measurements and place limits on the extent of parametric studies.

Taking into consideration the above, the work undertaken in WP3 was coordinated such that there were points of similarity between all the static and rotating cooling geometries that can be bridged using CFD. Furthermore, the measurements on the rotating test article can be strengthened by the tests on the (similar) static channels which benefit from an enhanced instrumentation cover.

One other objective was to interact with optimisation specialists in order to generate a process whereby testing, CFD and optimisation can be used to explore more effectively the design space. In practice OEMs already have such processes in place and it was not the intention to replace them with the outcome of the work supported by WP3. Instead, the intention was to expose OEMs to relevant heat transfer information such that their methods could be honed against such data.

The initial discussions held with partners in the work package and indeed the project helped to clear a number of misconceptions amongst the different partners. This outcome on its own was very positive for many of the partners not formerly exposed to the cooling of turbine components as it triggered a challenge of a mind-set as much as of the set of capabilities. The question of the shapes of the ribbing gave rise to a number of lively discussions with many of the CFD practitioners and optimisation specialists. Initially, many of the analytical practitioners joining the project believed that they would not be able to use rounded ribs and would have to confine themselves to sharp edged ribs. This conflict was resolved rapidly as all practitioners challenged their preconceptions and the capabilities of their methods to adopt the use of rounded ribs.

The choice of static test channels for WP3 was largely driven by Rolls-Royce UK (see Figure 14 and Figure 15). The choices made resulted from reviews of the open literature, a judgment on the likely needs for current and new designs and the desire to provide a realistic channel for the project. Clearly, the open nature of the project would have curtailed the freedom to explore specific designs (or operating conditions) but it is arguable that in many respects a more fundamental study can bring about more durable gains. With that in mind, the cooling geometries investigated by UUOXF and IMPPAN were large aspect ratio channels with filleted corners and ribs. The static test sections were constructed in a modular form to permit, where possible, the re-use of the same components for different tests.

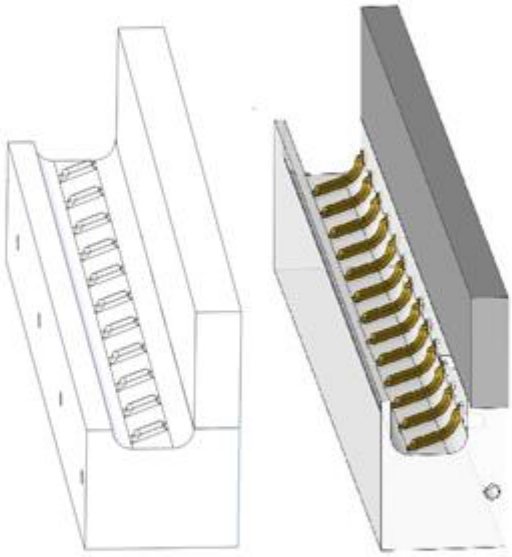


Figure 14: UUOXF Module for baseline and corner wrapped ribs

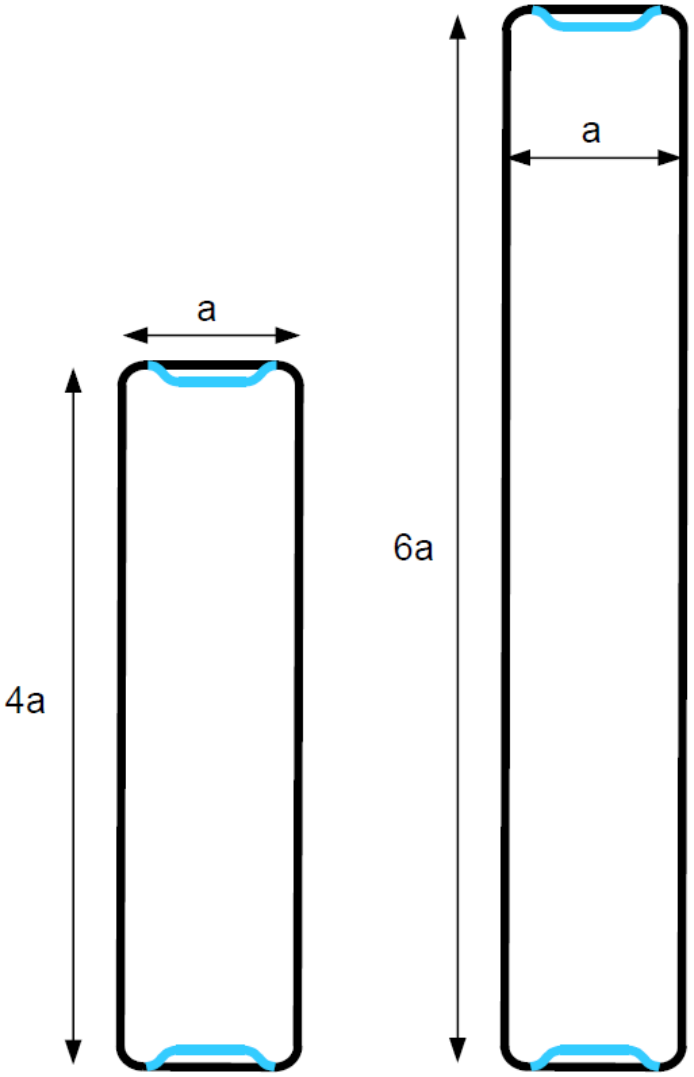


Figure 15: Schematic of cross section of channels tested at IMPPAN

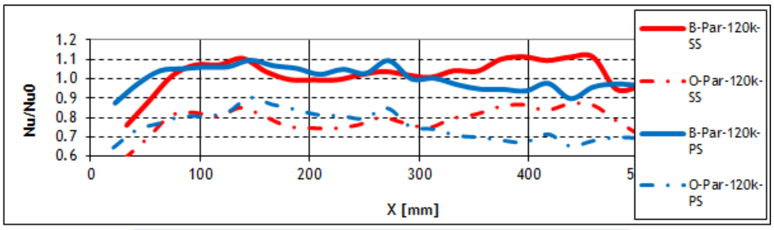
The measurement techniques used in WP3 relied on the use of thermochromic liquid crystals, thermocouples and pressure transducers. IMPPAN also reported the use of pitot tubes and hot wire anemometry to qualify the details of the fluid stream in the cooling channels. Thermochromic liquid crystals show a relatively abrupt change of phase as they reach a specific temperature; this temperature can be obtained consistently thus making the ‘liquid crystals’ very accurate temperature markers. The change of phase is accompanied by a change in the intensity of reflected light; this change is repeated reliably as the ‘liquid crystal’ changes phase.

Both UOXF and IMPPAN made use of externally mounted video cameras which could view three sides of the test section at any time through the use of mirrors. This minimised the number of repeat tests (as only an extra test would be required to take readings from the 4th unseen surface. UUOXF (Oxford University) have a long track record on the use of thermochromic liquid crystals, having been pioneers in their use. Yet the cooling geometries investigated provided a significant challenge to their capabilities. IMPPAN on the other hand took part in the project with a view to gaining experience on the use of thermochromic liquid crystals.

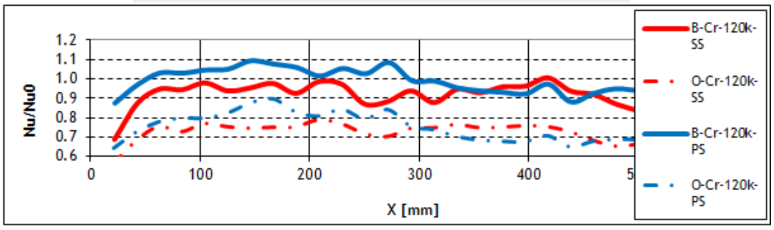
From the selection of cooling geometries of interest, UOXF and IMPPAN launched their test campaigns on baseline geometries. UOXF initially measured heat transfer from cooling channels with aspect ratios of 1:2, 1:3 and 1:4. The work package leader encouraged IMPPAN to take a first look at one of the cooling geometries investigated by UOXF (the 1:4 aspect ratio channel). This was deemed to provide a convenient back to back benchmark for both sets of investigators. As hinted above, measurements are taken both from the ribbed (external) walls and the side (internal) walls; the information is equally important to a designer and even though the cross section appears symmetrical, the relative inclination of the ribs with respect to the direction of the flow in the channel renders the arrangement asymmetrical.

One of the initial outcomes of WP3 was the specification of flow conditions suitable for testing as well as the modelling / optimisation exercise. It is relatively straightforward to select a range of interest as this is normally done by choosing a Reynolds number, the aspect ratio and the ribbing arrangement; however the choice of geometrical scale determines the power and instrumentation requirements as well as the accuracy of the measurements.

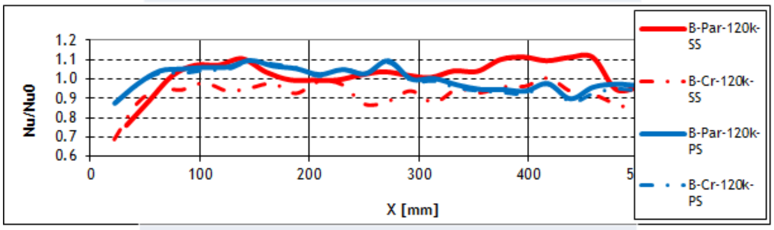
The measurements obtained by UOXF and IMPPAN are certainly novel in terms of data available in the literature for channels of the shape investigated, which satisfies one of the main objectives of the project. Data available in the literature tends to come from arrangements relying on the measurement of heat loss from patches of heaters attached to the channel surfaces; such an arrangement provides no detail beyond any physical splits between the heaters. UOXF developed a 3D mapping technique for measured heat transfer coefficients which increased the applicability of the TLC technique to complex geometries; this was very successful and exceeded the original scope of the work. IMP PAN also took the initiative of investigating the relative alignment of ribs on the opposite faces (Figure 16). Normally, the turbulators in a ribbed HP component cooling channel are aligned in the sense that they are inclined at the same attitude with respect to the direction of the flow aiming to avoid ‘rifling’ of the passage; in addition the ribs on opposite walls are staggered with respect to each other. IMPPAN designed their test channels such that the relative alignment could be changed simply by pivoting one of the panels making up one of the ribbed wall. Thus, the ribs on opposite walls formed a criss-crossing pattern.



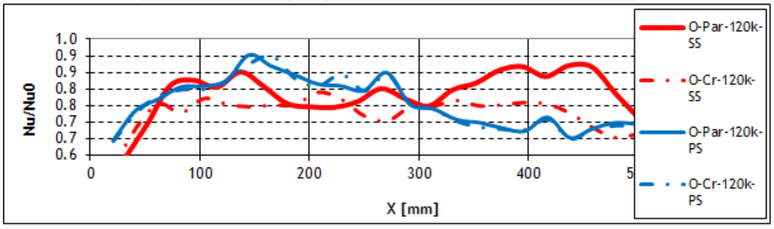
Baseline (1:4) v Optimised (1:6) – Parallel Ribs



Baseline (1:4) v Optimised (1:6) – Crossed Ribs



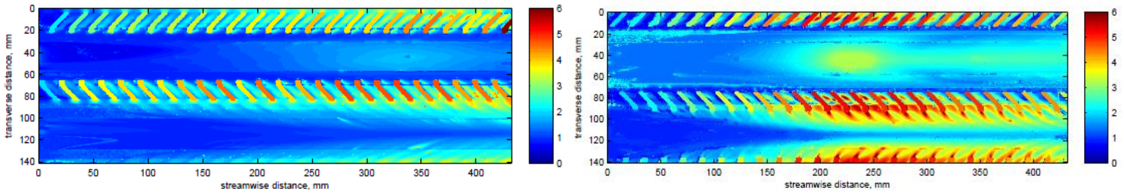
Baseline (1:4) – Parallel v Crossed Ribs



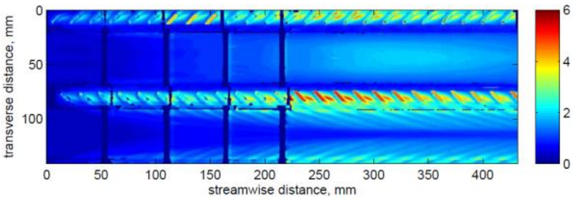
Optimised (1:6) – Parallel v Crossed Ribs

Figure 16: Task 3.3 - Results for the various arrangements tested- Ribbed walls

The conclusion of both test campaigns on the baseline geometries resulted in the creation of a comprehensive set of results of detailed heat transfer coefficient from end (ribbed) and side walls on large aspect ratio channels (see **Figure 17**). As it stands, the information could be used directly for design calculations; alternatively, a more useful outcome, as test cases for CFD methodology. In addition to matching distributions of the heat transfer coefficient there are pressure gradients and maps as well as turbulent intensity distributions to replicate.



‘Trailing edge wrapped ribs’ ‘Leading edge wrapped ribs’



‘Nominal ribs’

Figure 17: Task 3.4 - Aspect ratio 1:3 - Effect of wrapping on the distribution of the normalised Nusselt number

The original plans for the Work Package intended timing the conclusion of the test campaigns on the baseline geometries (Task 3.2 and early Task 3.3) to the outcome of modelling and optimisation analysis in order to inform the design of further optimised cooling channels. This did not happen as planned due to significant difficulties in making progress with the optimisation analysis. The extent of the progress did not reflect the effort by the partners; in fact it highlighted a key difficulty with the ‘objectives’ of the optimisation. To mitigate the impact of the delay in the optimisation work, the work package leader took the decision to select, on the basis of judgement and experience, a set of geometries that could be regarded as ‘optimised’ for testing in the second half of Task 3.3 and Task 3.4. Thus for Task 3.3, the work undertaken at IMPPAN, a 1:6 aspect ratio channel was chosen while for Task 3.4 (at UOXF) the effect of ‘wrapping’ ribs around the corner of the channel was selected as the optimisation parameter to be investigated. The outcome of the tests on the ‘optimised’ cooling geometries has been to provide a clear view of the impact of increasing aspect ratio as well as the details of the ribs in the channel Figure 18). Comparison of the data with that obtained for the baseline geometries does expose significant differences that could be either exploited or mitigated in a cooling design as required.

The Rotating Heat Transfer Rig housed two sets of cooling geometries for work undertaken under WP3 (See Figure 18). One set consisted of three intermediate pressure turbine blade cooling passages of interest to ITP and one three passage serpentine suggested by Rolls-Royce UK. The three pass serpentine consisted of one outward flowing passage with an aspect ratio of 1:4, a radially inflowing passage also with an aspect ratio of 1:4 and a last radially outflowing passage with an aspect ratio of 1:3. Although more extreme aspect ratios are possible, these would have come at the cost of a reduced target surface since the cross sectional area has to be maintained.

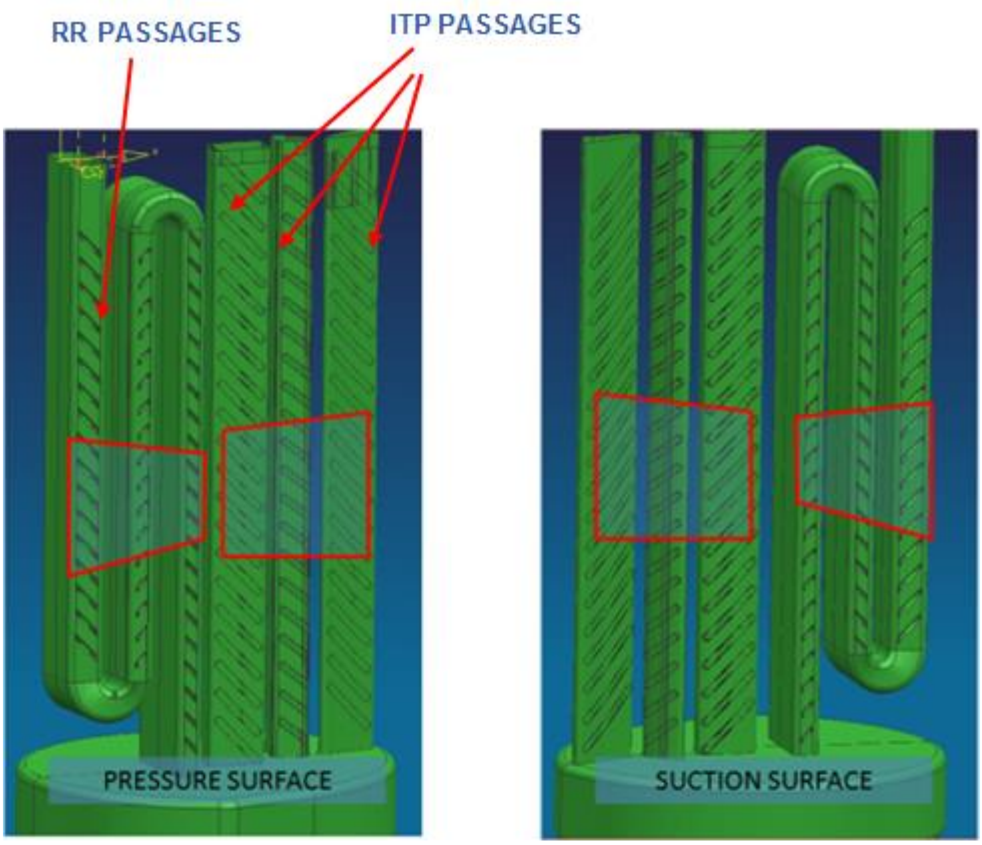


Figure 18: 'Core' passages inside rotating test article.

Work on the rotating heat transfer test campaign started early in the project although it was hampered by significant delays. The main delay arose from the very late signature of the Consortium Agreement; which happened at the beginning of PR2. This in turn delayed the start of the ‘design and make’ phase of the rotating test articles until M15.

The results obtained from the rotating rig can be regarded as little less than striking. In some respects the values of the average heat transfer coefficient measured follow trends with rotation that might have been expected from information available in the literature and yet in others it departs significantly (and consistently) from them (see for example **Figure 19**). The results obtained also contain a significant amount of detailed information directly comparable with the results from the static facilities. In particular, the patterns of heat transfer coefficient obtained in the inter-rib areas should provide a challenge to the capabilities of the CFD methodology to replicate them both in the static and rotating facilities. The benefits do not end there; the on board cameras were angled in order to capture a view of the side faces which, as pointed above, are important in determining heat transfer in the cooling channels.

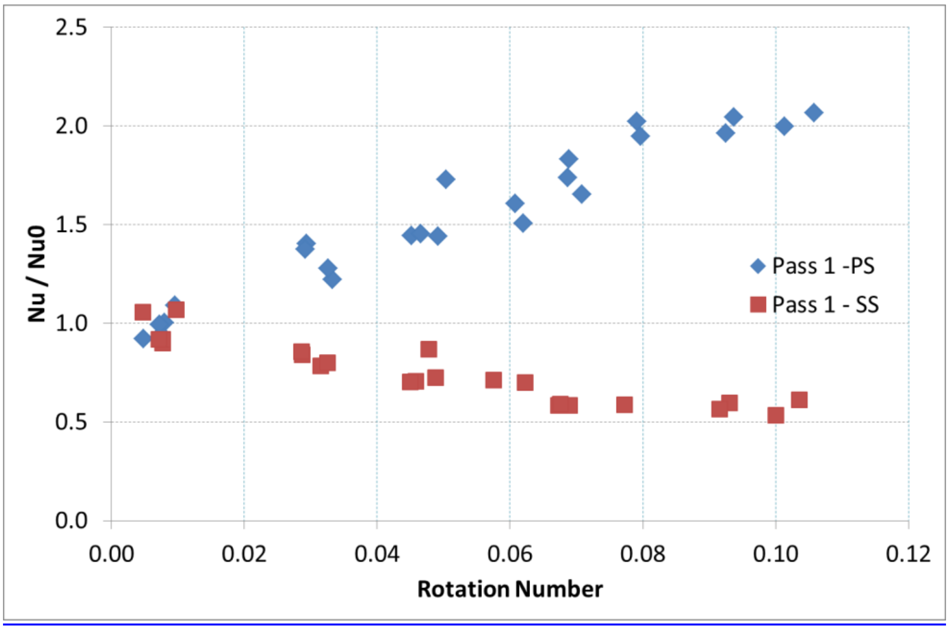


Figure 19: Task 3.5 - RR Serpentine Pass 1 (AR 1:4) - Correlation of area averaged Nusselt number on ribbed faces with Rotation Number

WP4 –U-bend and radial passage study

The main objective of WP4 was the generation of an experimental database for radial and U bend ribbed internal cooling systems for both high pressure and low pressure turbine configurations, in both static and rotating conditions. The tests were performed by USTU and ONERA using the static test rig at USTU and the BATHIRE facility at ONERA, both cases in close collaboration with MTU and Snecma. Results from the experimental campaign can be directly used for the design of such systems, or for the accurate calibration of CFD codes.

WP4 was divided into three tasks.

Task 4.1:

The task 4.1 consisted in the BATHIRE tests with high pressure turbine geometry. The objective of Task 4.1 was to generate an experimental database for the high pressure turbine geometry defined by Snecma. Tests were performed by ONERA using the BATHIRE facility.

First, Snecma defined the aero design for radial and U-bend passages to be tested on the BATHIRE rig. Snecma and ONERA specified test conditions and boundaries conditions for the experiments. Three configurations were designed and tested at ONERA: a smooth configuration, a configuration with 14 ribs only in the channels and a configuration with 16 ribs in channels and bend.

The ribs are rectangular and the ribs flow angle is 45°. Only the stainless steel plate is heated and ribbed to allow for optical access during infrared measurements. The mock up was manufactured and instrumented on the BATHIRE rig. The heated wall is fixed on the skeleton with no contact with the framework in order to prevent backward thermal losses. In front of the measurement plate is the sapphire window divided into three parts in the mock up length. This sapphire window offers an optical access for Infrared measurements.

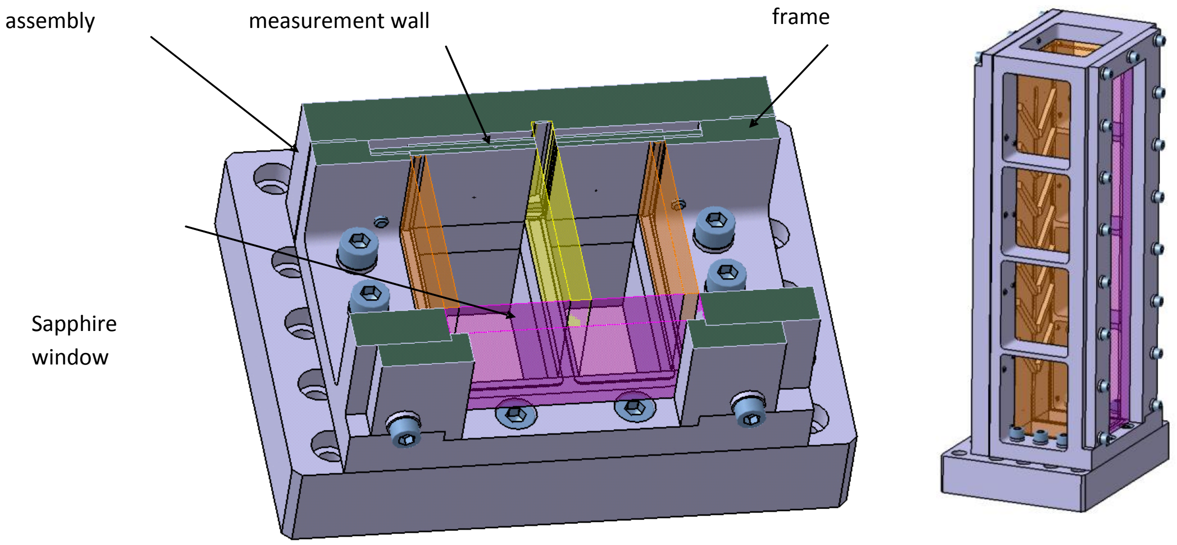


Figure 20 : HP mock up

ONERA performed an experimental campaign on the BATHIRE rig following tests conditions specified by SNECMA. For each configuration, three Reynolds numbers based on the inlet channel passage were tested. The impacts of rotation number were also investigated. For each test case three heating power values were applied. This is required to post process the data.

The experimental results were post processed by ONERA to obtain the heat transfer coefficient repartition on the heated wall based on the temperature maps. The thermal losses within the experimental mock up were difficult to estimate. Thus, ONERA performed additional tests to assess the heat losses: Infrared measurements of the back side of the back side of the mock up were achieved in static conditions. Several post-processing techniques were used to try to get the most reliable results possible. The post processing techniques are the ones following:

1. Estimation of the losses with the Dittus Boelter correlation on the smooth channel with thermocouples readings
2. Estimation of the losses with the backside temperature measurements
    a. Computation of the wall temperature
    b. Computation of the heating film temperature
3. Relative comparison of the cases
    a. Influence of heating power
    b. Influence of rotation
    c. Influence of the ribs

The heat transfer coefficients maps and Nu/Nu0 maps for each tests condition are available. The following figure shows Nu/Nu0 distribution for the three configurations in rotating conditions resulting from the first post processing method.

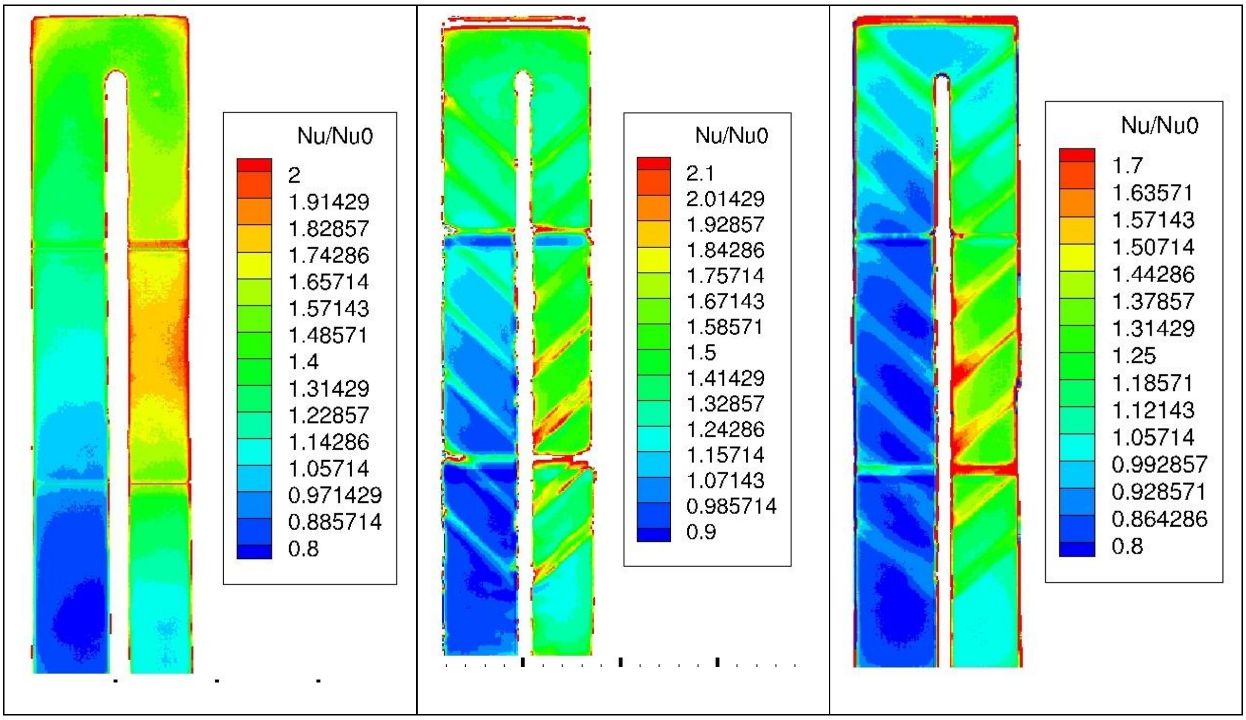


Figure 21: Nu/Nu0 for Re = 25000; Rotation pressure side Ro = +0.22

The heat transfer coefficient range is limited on the experiments. Therefore, in the third method, the effects of heating power, rotation number and number of ribs on the global Nusselt number were studied. For example, the studies showed that the additions of ribs in the channels had a decreasing beneficial effect on the heat transfer with the rotation.

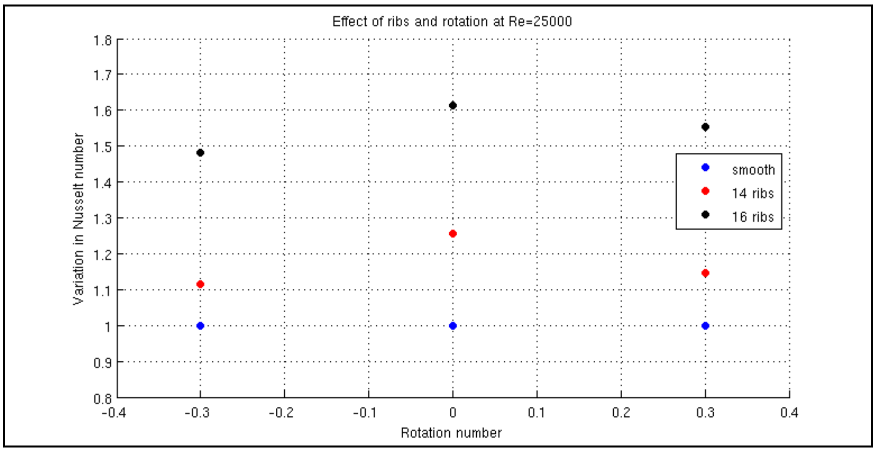


Figure 22: Average Nusselt number ratios variation with rotation number

Thanks to these additional studies some trends were identified and can be compared to CFD results.

***Task 4.2**:*

The task 4.2 consisted in the BATHIRE tests with low pressure turbine geometry. The objective of the Task 4.2 was to generate an experimental database for the low pressure turbine geometry defined by MTU. Tests were performed by ONERA using the BATHIRE facility.

The specification of the LPT test geometry was performed in close collaboration between ONERA and MTU. Thereby ONERA provided the geometric constraints as well as the range of operating conditions for the BATHIRE facility. MTU defined the aero design for radial and U-bend passages to be tested on the BATHIRE rig as well as tests conditions for experiments. To ensure an optimum optical access for infrared thermography measurements only one face is ribbed and heated. Therefore, MTU performed several numerical studies to investigate the impact of having only one wall ribbed versus two faces on the flow and heat transfer. The CFD simulations showed that removing ribs on one wall has a significant impact on the heat transfer level and distribution.

Following the rig constraints the basic test geometry with ribs on the suction side only and adjusted rib spacing was defined in M4.02 together with the required operating conditions. One configuration was designed and tested at ONERA. The mock up was manufactured and instrumented on the BATHIRE rig. The heated wall is fixed on the skeleton with no contact with the framework in order to prevent backward thermal losses. In front of the measurement plate the sapphire window offers an optical access for Infrared measurements.

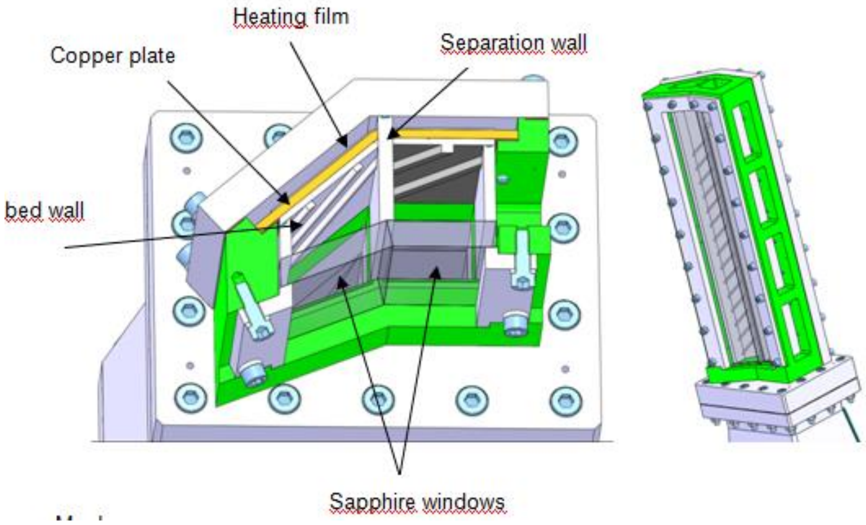
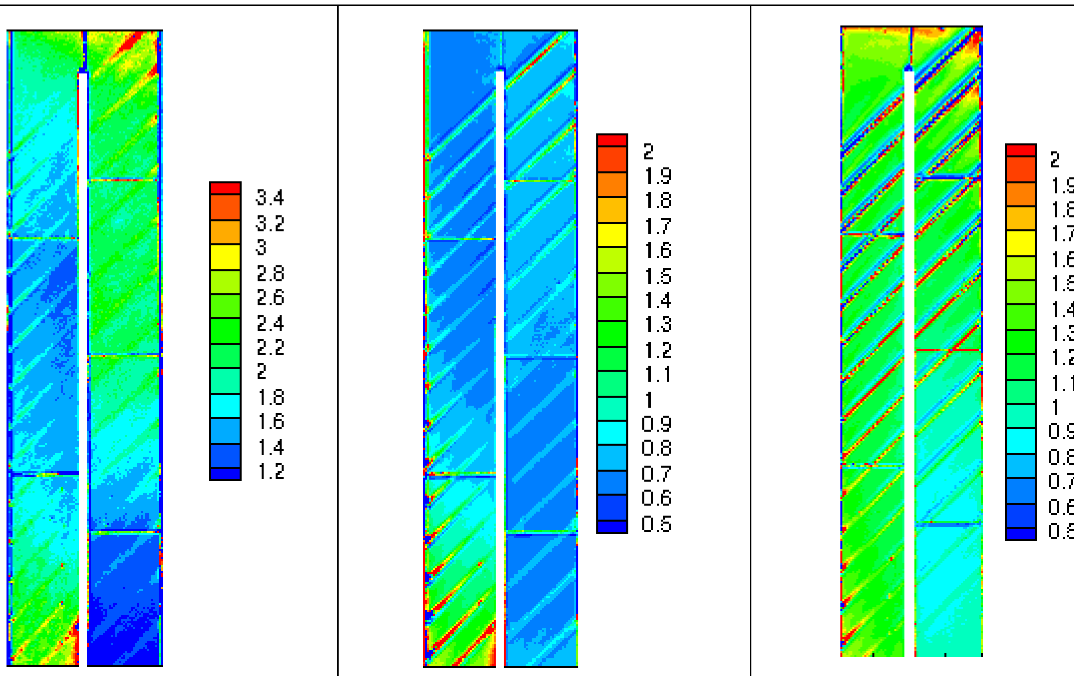


Figure 23: Mock up

ONERA performed an experimental campaign on the BATHIRE rig following tests conditions specified by MTU. Four Reynold numbers based on the inlet channel passage were tested (10000, 20000, 40000 and 60000). The influence of rotation was also investigated. In order to post process the experimental data for each Reynolds numbers three value of the heating power were set.

The experimental results were post processed by ONERA to obtain the heat transfer coefficient on the heated wall based on the temperature maps. Some difficulties were encountered to assess the heat losses. Finally estimation of the losses was achieved thanks to the Dittus Boelter correlation at the entrance of the channel.

The following figure shows Nu/Nu0 distribution deduced from the temperature maps for illustrative results.



Static

Rotation forward 380 rpm

Rotation backward 380 rpm

Figure 24: Nu/Nu0 for Re = 40000

The results obtained from the measurements and illustrated in this report through the case Re=40000 point out some difficulties in post processing the results. The heat transfer coefficient range is limited on the experiments. Thus additional post-processing investigations were done to identify trends in the results, in particular the effect of rotation on the heat transfer was studied.

Task 4.3:

The objective of the task 4.3 was to generate high quality local heat transfer database for engine representative turbine blade internal cooling configurations using transient thermochromic liquid crystal method to be usable for optimization purposes. MTU and USTU defined a starting geometry to be tested on the static rig. The geometry is a simplified low pressure turbine cooling passages configuration. It includes an U-bend and two radial passages with ribs on the suction and pressure sides. The first passage represents a leading edge cooling passage with radial outward flow. It has a trapezoidal shape whereas the second passage has a rectangular cross section.

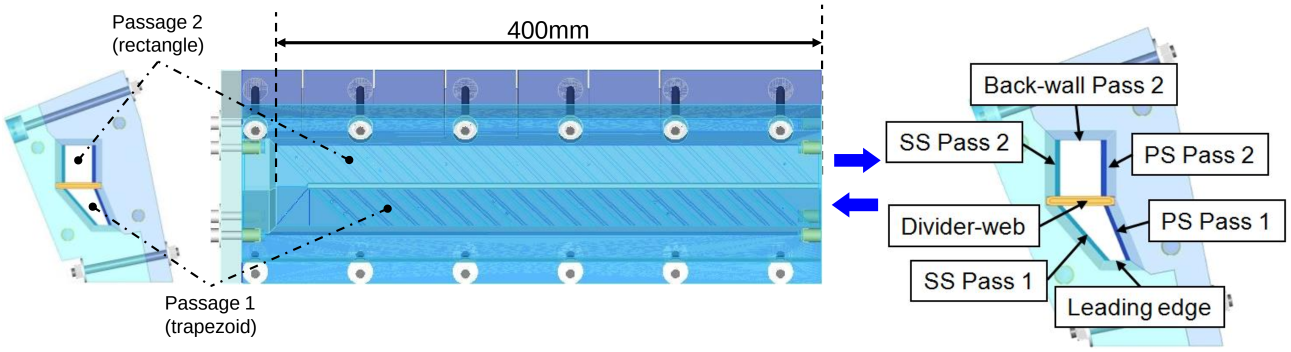


Figure 25: Passage cross section and model dimension

Based on the CAD model a Perspex model was manufactured to allow for optical measurements using the transient Thermochromic-Liquid-Crystal (TLC)-technique. The Perspex-model consists of four main parts: a ribbed suction sidewall, a ribbed pressure sidewall, a divider web and a tip-wall. The inner channels were coated with TLCs and black backing paint. After assembling the model is instrumented with 20 thermocouples and 13 pressure measurement taps. The leading edge, the pressure side and the suction side are observed via a single RGB video camera (either directly or via mirrors). Detailed local heat transfer measurements in the relevant parameter ranges were performed using the transient Thermochromic-Liquid-Crystal (TLC)-technique. Tests were conducted for five Reynolds numbers.

MTU performed CFD simulations on the starting geometry. Based on the experience with the baseline geometry a wide range of rib configurations was investigated numerically. The influences of the rib angle, rib pitch, rib height and rib width on the heat transfer and friction factor were studied. More than 50 configurations have been investigated.

Based on the starting geometry data and CFD analysis the first optimised geometry was defined. Besides, several improvements to the test rig were implemented for the first optimised geometry. Compared to the reference baseline geometry the ribs configuration of the first optimised geometry was altered: the rib angle was changed from 45 degree to 60 degree and the rib height to width ratio was increased. All other parameters such as the channel cross-section or length remain unchanged. USTU performed heat transfer and pressure loss measurements on the first optimised geometry for five Reynolds numbers.

Then, MTU specified the third optimised geometry to be tested on the static rig at USTU. This last geometry is based on the optimised geometry defined by CENAERO in WP1. MTU performed several CFD investigations concerning the effect of some rib parameters on the heat transfer distribution. Three parameters were changed from the initial CENAERO geometry. The aligned ribs became staggered, then the angular ribs became normal and finally the high ribs became smaller on the SS. The design and manufacturing of the third geometry are achieved. Heat transfer experiments are completed. USTU is performing pressure loss measurements and the data evaluation is ongoing.

For the three tested geometries, results of heat transfer measurements are presented in three different ways: local Nu/Nu0 distribution, segment averaged Nu/Nu0, line averaged Nu/Nu0 (the reference Nu0 is deduced from the Dittus-Boelter correlation) as given in the figures below. USTU and MTU defined data evaluation, analysis and reporting procedures to allow for direct comparisons of the experiments with numerical simulations as needed for numerical optimization processes.

The high quality local Nu/Nu0 distributions for the starting and first optimised geometry are illustrated on the following figures:

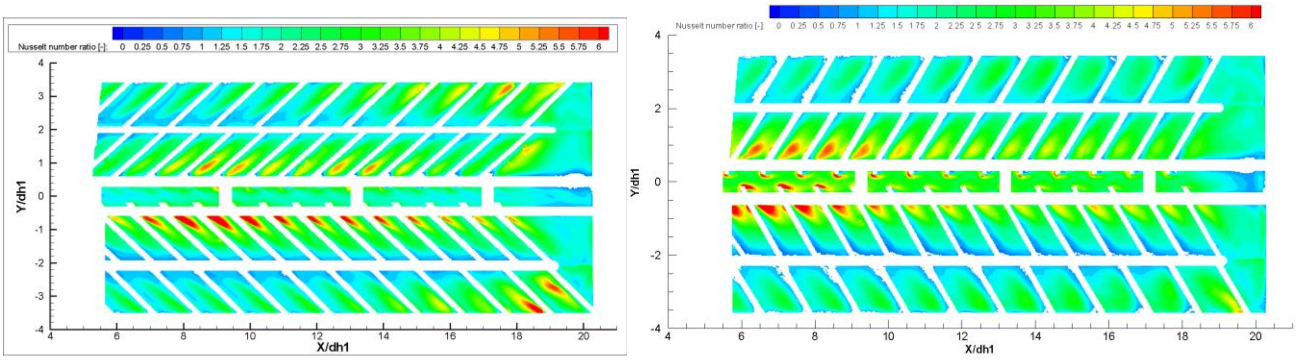


Figure 26: Local Nu/Nu0 distribution for starting (left) and first optimized geometry (right)

The optimisation of the rib configuration clearly enhances the heat transfer on the leading edge.

In the following presentation of the results, the suction and pressure sides are divided in inter ribs segments. The segment averaged Nu/Nu0 is plotted and can directly be compared to CFD results.

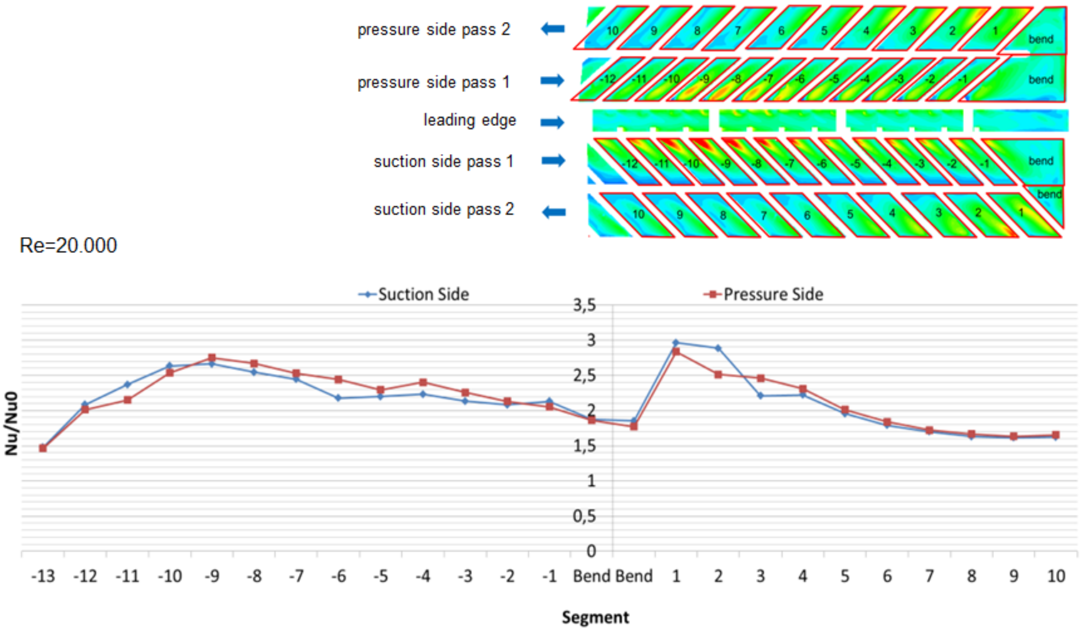


Figure 27: Data evaluation for segment-averaged heat transfer information; starting geometry

The last form of data reduction is presented by line averaged Nu/Nu$_0$-values that are plotted against s/dh1 (normalized stream wise distance). The influences of the local rib arrangements on the local heat transfer distributions can well be observed.

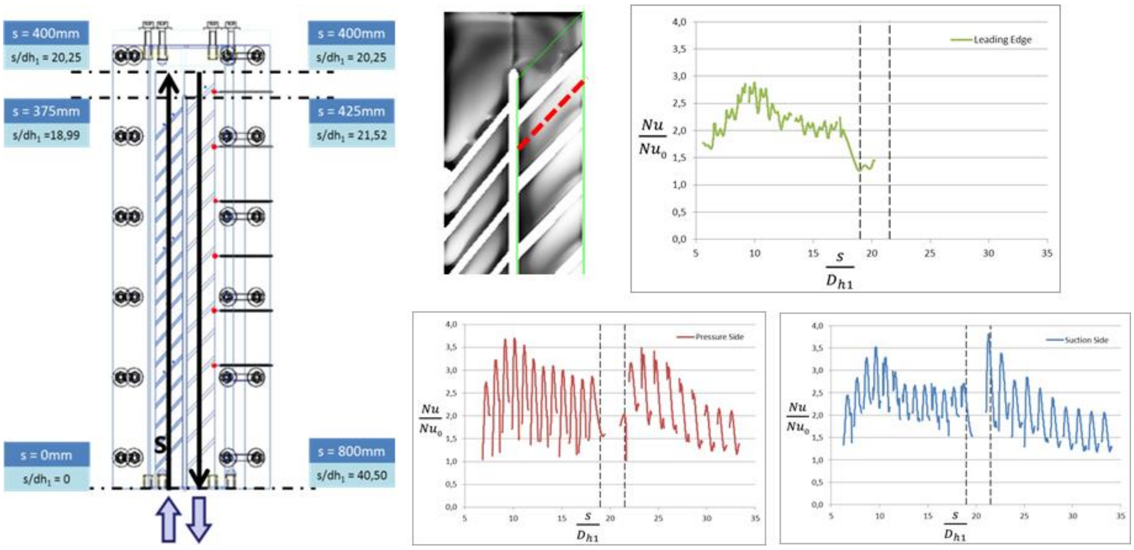


Figure 28: Data evaluation for line-averaged heat transfer information for the individual model sides

A concise database of the starting and optimised geometries was achieved. The database contains spatially-resolved, area-averaged and line-averaged heat transfer data as well as pressure loss measurements for the five investigated Reynolds numbers. The results of the starting geometry obtained through the cooperation between MTU and USTU were published in the joint technical paper: “Investigations of heat transfer and pressure loss in an engine-similar two-pass internal blade cooling configuration” and was presented at the 10th European Conference on Turbomachinery in Lappeenranta, Finland.

WP5 – CFD Calculations

A large numerical activity was planned to complete the experimental database produced within the ERICKA project. The main goal was to provide practical recommendations for developing future cooling systems in an industrial context. Most of the partners involved in the project are also involved in WP5 which is dedicated to Computational Fluid Dynamics (CFD). Each of these 12 partners has performed several computations relying on a variety of both commercial and in-house developed CFD tools. The idea is to obtain a large overview of the performance of the CFD codes commonly used within the European turbomachinery industry and research institutions. From this very unique information, relevant practices can be extracted to fulfil the WP5 objectives.

The WP5 outline follows the project logic. The three main tasks of the work-package were devoted to the configurations studied in the three corresponding experimental work-packages. A complementary task was added to extract the best practices emerging from the whole set of computations.

ERICKA uses 7 rigs that lead to the definition of 17 configurations. For each of these configurations several cases can be operated by varying the geometry, the Reynolds number, the rotation rate or the thermal boundary conditions. In addition to the set of configurations, it has been decided to test codes on an existing experimental case. The results were obtained after a test campaign done on ONERA’s MERCI rig, a rotating internal cooling U bend channel. These contain velocity measurements useful to discriminate the codes or the used models, especially for turbulence. In fact, one of the drawbacks of the configurations studied in the ERICKA project is that they do not include any velocity measurements. It has consequently been assumed that codes will be able to reproduce the aerodynamic fields and that the main difficulties of simulating such configurations lie in the estimation of the heat transfers for the large scale observations.

In addition to the undertaken simulations on the tested configurations, preliminary computing was done to prepare the geometry and operating conditions of the configurations tested. The list of all cases analysed is summarised in Table 2

	Case	**Rig**	**Conditions**	**Partner contribution**
WP2 : Impingement	ALST geometry	RHR	rotating	ALSTOM, ONERA, ITP/UPM
	AVIO geometry	RHR	rotating	EST, ONERA
	ALST geometry	UNIFI	static	ALSTOM, ONERA
	AVIO geometry	UNIFI	static	RRD, EST, ONERA
WP3 : Radial Ducts	MC passage 1:4 aspect ratio	UOXF	static	MTU, NUMECA, UOXF.DF/RRUK
	radial passage	IMP PAN	static	NUMECA, UOXF.DF/RRUK
	RR triple pass	RHR	static	UOXF.DF/RRUK
		RHR	rotating HPB conds	UOXF.DF/RRUK
		RHR	rotating LPB conds	UOXF.DF/RRUK
	ITP pass. (LE,Mid-Chord,TE)	RHR	static	ITP/UPM, UOXF.DF/RRUK
		RHR	rotating	ITP/UPM, UOXF.DF/RRUK
WP4 : Bends	HPT geometry	BATHIRE	static	MTU, SN, CENAERO, NUMECA, ONERA, ITP/UPM
		BATHIRE	rotating	MTU, SN, CENAERO, NUMECA, ONERA, ITP/UPM
	LPT geometry	BATHIRE	static	ONERA
		BATHIRE	rotating	ONERA
		USTU	static	MTU, SN, CENAERO, ONERA, ITP/UPM
External	MERCI test case	ONERA	static: smooth and ribbed	MTU, CENAERO, NUMECA, ONERA, ITP/UPM
		ONERA	rotating: smooth and ribbed	MTU, CENAERO, NUMECA, ONERA, ITP/UPM
WPs support	WP2	RHR	rotating	EST, ONERA
	WP3	RHR	rotating	NUMECA, UOXF.DF/RRUK
	WP4	BATHIRE/USTU	rotating/static	MTU

Table 2: CFD cases processed within ERICKA

All the experimental configurations have been simulated by at least one partner. Some configurations have been covered by more than one, which has allowed for the cross-comparison between results provided by different solvers and modelling strategies especially on:

- WP2: ALSTOM Leading Edge operated at UNIFI in static conditions
- WP3: MC passage operated at UUOXF in static conditions for which the results obtained were similar with the models used
- WP4: LP Ubend operated at USTU in static conditions, for which the results were close with the partners involved
- WP4: HP Ubend operated at ONERA in static and rotating conditions, which shown very similar results, but could not be efficiently compared to experiments
- The MERCI test case

A template was defined to help the different partners synthetize each numerical study performed. It has three main categories: mesh, solver and results, which allow having a quick look on the main characteristics of the computations (Table 3). This document is provided along D5.5 on the website as "WP5_CFD_template_and_summary.xls"

MESH	**SOLVER**	**RESULTS**
Reference to geometry (ERICKA report)	**Reference to test conditions (ERICKA report)**	Mesh sensitivity conducted? If so, what are main conclusions?
Domain descritpion (intake/outlet regions considered). Add reference to illustrative images.	Solver software (incl. version number)	Turbulence models influence
Meshing software (incl. version number)	Compressible/uncompressible solver	HTC definition (compare with experimental HTC, add reference to experimental report)
structured/unstructured mesh	steady state/unsteady/transient	Main Conclusions
Element types (tetra, hexa, etc.)	Time discretization (if required)	
Number prism layers (if applicable)	Numerical schemes	
Prism layer expansion ratio (if applicable)	Boundary conditions	
Prism layer height (total or first element) (if applicable)	Turbulence model, including any modifications of default closure coefficients.	
Local mesh refiment controls	y+ values	
Further comments on achieved mesh quality	Specific Low-Re modelisations (if applicable) or use of wall function	
Final mesh size/s	Convergence criteria (if applicable)	
Reference to illustrative mesh image		

Table 3: Numerical studies template

Main results

Several categories are identified to summarise the results obtained from the large number of computations done during the project. They comply with the template from Table 3.

Meshing

Meshes play a key role in the accuracy reached by the computations. The meshes have to be defined in accordance with the solver capabilities. Indeed, unstructured compressible Navier-Stokes solvers dealing with generalized polyhedral elements do not require the same mesh than a structured incompressible Euler solver. Given the context of the ERICKA project that focuses on cooling technologies for turbomachines, most of the computations have been performed using unstructured compressible Navier-Stokes solvers.

Several technical solutions are available for preparing meshes dedicated to unstructured solvers. The majority of the computations have been carried out using meshes composed of tetrahedrons and prisms layers for near wall refinements. The latter are necessary to catch the boundary layers that develop along the walls, which drive the heat transfer through the viscous friction forces. However, such refinement needs to fit the main flows properties in order to increase the accuracy of numerical schemes. Using prisms layers on the wall enable the meshes to have faces orthogonally aligned with the main gradients of the flows, especially in the wall normal direction. This practice is common and well spread in the turbomachinery industry.

These general recommendations do not guarantee the quality of the mesh yet. The mesh size has to be adjusted to catch the flow gradients regarding the order of the used spatial schemes. It appears that most of the meshes with targeted refinement (ribs, jets and U-bends) yield meshes with around 10M cells. Some which are dedicated to geometrical optimisations or unsteady simulations are however larger, reaching 20M cells to allow for full volume parameterisation and to catch small flow effects. 10 to 20 prisms layers are used on the walls and prove to be sufficient to obtain y+ values around unity for the wall cells but more layers may be indicated in some cases presenting strong near wall gradients.

All partners have used meshes composed of tetrahedrons and prisms layers (Figure 29), except for NUMECA who have used trimmed meshes combined with prisms layers as depicted in Figure 30. One of the advantages of this technique is to propose meshes with a structured-like core. Even if solvers are to treat unstructured meshes, the use of structured meshes increases the spatial schemes performances. However, this kind of solution can be inappropriate or hardly feasible when configurations exhibit complex geometries. More investigation of the potential of structured meshes can be further studied for industrial applications.

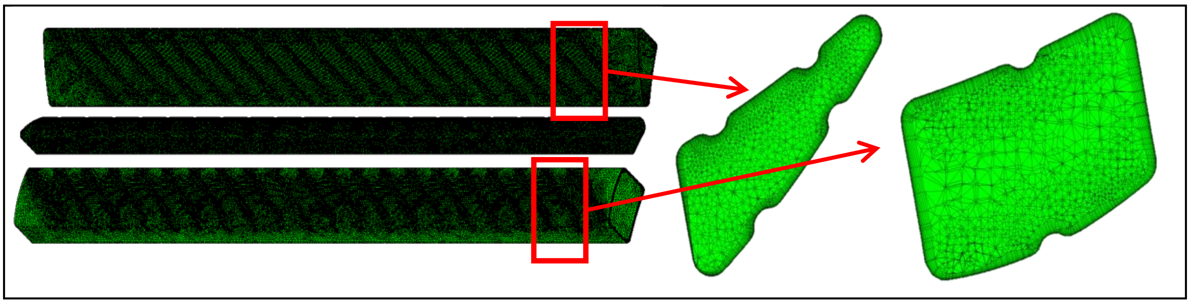


Figure 29 : example of mesh composed of tetrahedrons and prisms layers (work by UPM)

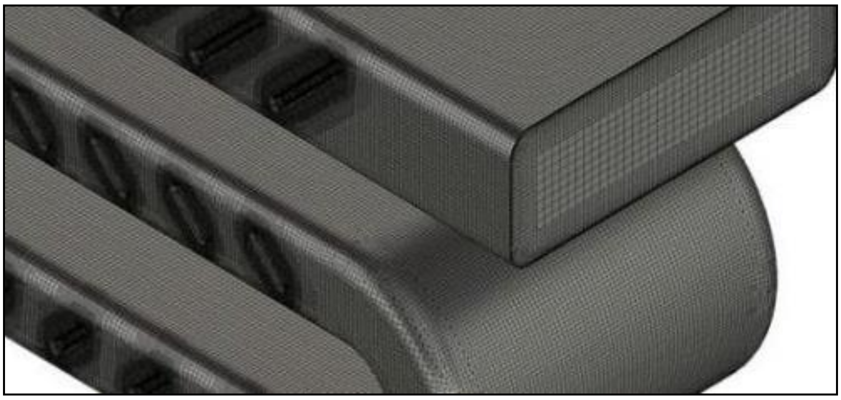


Figure 30: example of trimmed mesh including prisms layers (work by NUMECA)

Solvers and turbulence modelling

As explained above, partners have essentially used unstructured compressible Navier-Stokes codes. Some are commercial codes as Fluent, CFX, Starccm+ and some are in-house codes generally developed by labs. One of the interests of the present work is to propose an evaluation of all these codes which have different backgrounds and serve different goals. Indeed, in-house codes offer more advanced models, but are sometimes not mature enough for an industrial environment, whereas commercial codes propose robust and fast methods to answer their client requests.

The turbulence models have a predominant role among the numerical parameters that determine the quality of the produced results. Such models are expected to have a great influence on heat transfer. It was thus important to evaluate their impact on the configurations explored within ERICKA. Simulations were mainly based on a steady RANS formulation as one of the objectives of WP5 is to produce recommendations for industrial numerical simulations. Only few computations have been performed using unsteady modelling and are used as demonstrators but are less affordable for practical applications. Those are still of higher interest to get more detailed flowfields, more accurate results and allow for debugging and calibrating turbulence models.

Almost all the simulations were run using standard turbulence models:

- the Spalart Allmaras model
- the k-ε (realizable) turbulence model
- the k- ω SST and the standard k- ω Wilcox models
- the EARSM Wallin-Johansson model

Except from the EARSM model, all the used turbulence models are based on the Boussinesq hypothesis. This implies that the anisotropy of turbulence that may exists in the present cooling configurations due to rotation or curvature effects has not been taken into account. Depending on the level of precision that is expected for the simulations, one can consider the Boussinesq hypothesis acceptable or not.

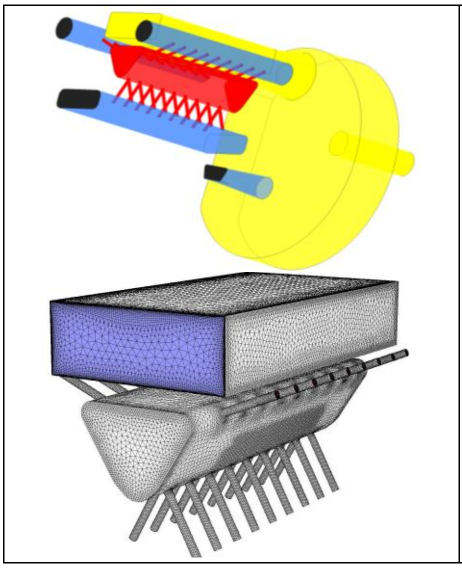
Coupled to the modelling of the Reynolds stress tensor, the turbulence models involve a modelling of the turbulence heat flux. Most of the time, the latter comes down to the use of the Fourier law (Gradient Diffusion Hypothesis) where the turbulent conductivity is obtained from the eddy viscosity and a constant turbulent Prandtl number. To go beyond, ONERA has conducted several studies with different levels of modelling for the turbulent heat flux. The Gradient Diffusion Hypothesis is dropped for the benefit of more complex modelling involving the Reynolds Stress Tensor to assess the accuracy and operability of these models. These promising models are not yet sufficiently mature to be commonly used as they depend on the resolution of the Reynolds Stress Tensor *per se*. As long as the latter is modelled using the Boussinesq hypothesis, it is not possible to rely on the heat fluxes produced by such models as it cannot change the solution. Two similar simulations but with different heat flux models has shown discrepancies. The aerodynamic field are almost identical, implying that the observed differences come from the turbulent heat flux model. As the difference in obtained values in small, very accurate experiments or highly complex numerical simulations would be able to validate one or another.

As proved by the obtained results, the k-w and EARSM gave close results with the accuracy required. So far, isotropic turbulence models are accurate enough to look at turbine blade heat transfer at a large scale and especially to catch trends and variation behaviours in physics which would facilitate optimisation processes.

Heat transfer calculations

In order to illustrate the variety of the produced results and the relevancy of the computations, three examples are provided below. The first one comes from a WP2 configuration dedicated to leading edge impingement techniques.

Figure 31 shows the geometry and a view of the mesh used for the computations performed by ALSTOM. The geometry is particularly complex with many holes and tubes. Recovering the pressure drops on such complex geometries is complicated for Navier-Stokes solvers as they are directly linked to the prediction of the wall friction coefficients. The mesh quality and the use of prisms layers is a key point for such configurations.



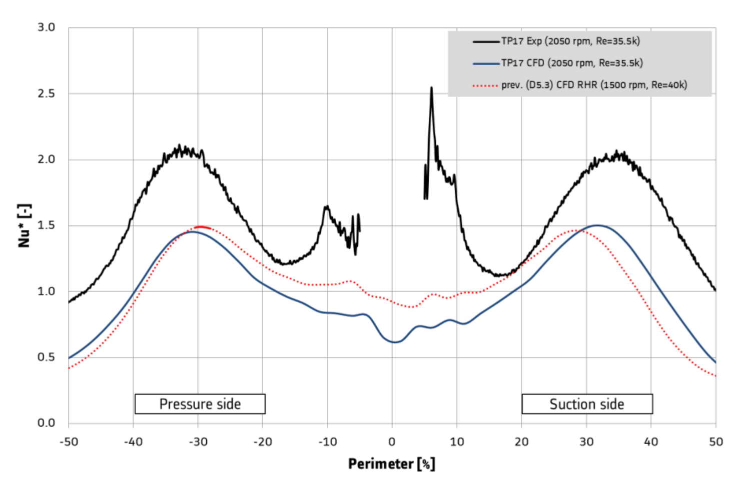
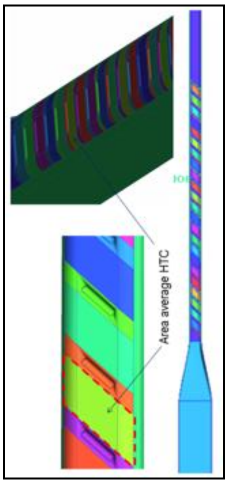


Figure 31: geometry and mesh of the ALSTOM leading edge configuration tested at the RHTR

Figure 32: comparison between experiments and computations for ALSTOM RHTR configuration (work by ALSTOM and RRUK)

In spite of these difficulties, reasonable results have been obtained that correctly discribe the distribution of the heat transfer on the leading edge surface as seen on Figure 32. The position of the maximums are well captured. In addition, the influence of rotation proved to be well captured by the simulations, but is not shown on the figure. However, the main concern with the simulations still resides in the quatification of the error the CFD models show, which, in this very example, reaches 35%. More investigation should be done on turbulence models and heat diffusion models to try to get pin point accurate heat transfer results.

The second example comes from a work by MTU on a configuration tested at UOxF. It corresponds to a radial passage of turbine cooling radial channel that was tested on static conditions (Figure 33).



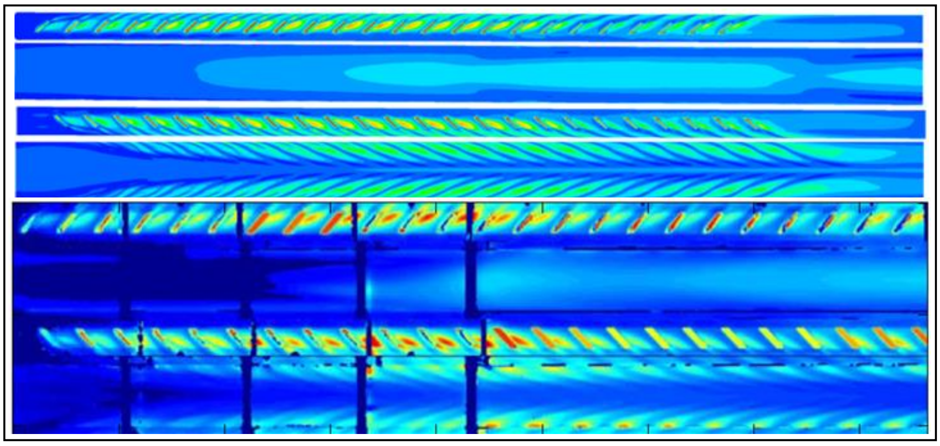


Figure 33: geometry of the UUOXF configuration

Figure 34: Nusselt numbers contours for numerical (top) and experimental (bottom) results

Figure 34 illustrates the good overall agreement obtained between the experimental and typical standard RANS numerical results in terms of heat transfers. One can note that the presence of ribs on the top and bottom part of the channel increases the heat transfer and guide the flow. The trace of these ribs is clearly visible on the lateral side of the channel in both the experimental and numerical Nusselt distributions.

Once again, averaged heat transfer evolutions and distributions are well predicted by simulations (slope of curves and position of min/max). The influence of the main parameters is captured which indicates the trends to be followed. Figure 35 shows the influence of the aspect ratio of the channel on the evolution of the averaged Nusselt number along the axis. If there are some noticeable differences between the absolute values of Nusselt numbers, the trends are reproduced by the computations. Such results prove the applicability of CFD in a conception phase of turbine blades cooling systems and complex optimisation processes. However, there is still a need of more refined simulations, using unsteady methods or complex models to get accurate values, scale the common RANS results obtained and ensure the design done.

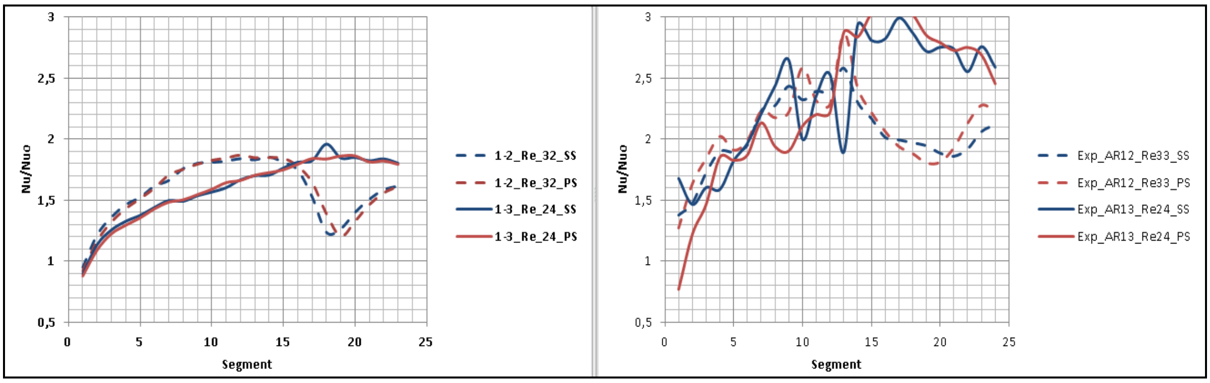
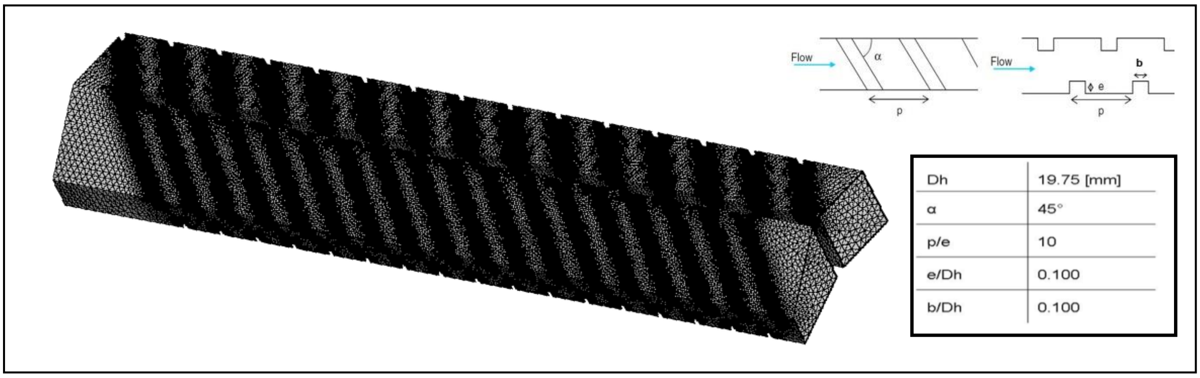


Figure 35: influence of the aspect ratio on the heat transfer (work by MTU)

The last example is linked to WP4. As explained above, a special attention was paid on the possible use of advanced turbulence models for evaluating the heat transfers. For this reason, ONERA studied the influence of several modern turbulent models both for the Reynolds stress tensor and for the turbulent heat fluxes.

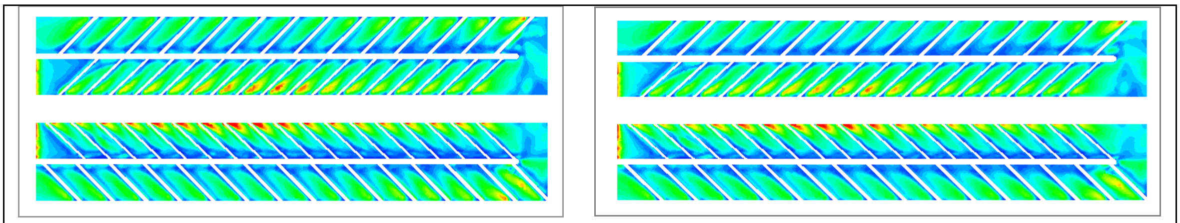
The chosen configuration is a Ubend ribbed channel that was tested at USTU, as shown in Figure 36. Very accurate thermal measurements were extracted from the experimental campaign enabling the evaluation of turbulence models.



Dh	19.75 [mm]
α	45°
p/e	10
e/Dh	0.100
b/Dh	0.100

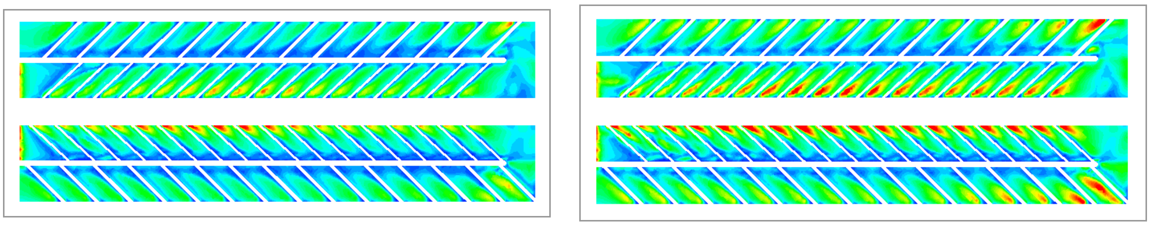
Figure 36: geometry and mesh of the Ubend ribbed configuration tested at USTU

Several Reynolds numbers were examined and for an illustrative comparison, experimental results and numerical simulations are compared on Figure 37.



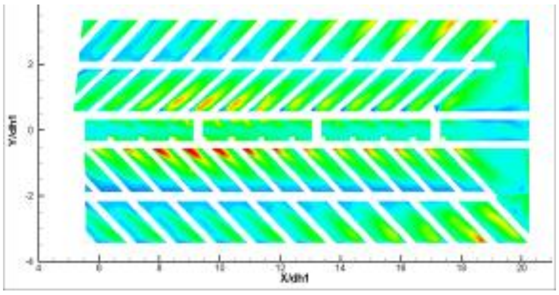
k- ω SST

EARSM

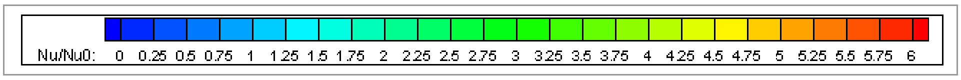


EARSM DH

EARSM GGDH



Measurements



Colour scale

Figure 37: numerical and experimental results

Generally speaking, a satisfactory overall agreement is obtained for all the tested turbulence models. However, there are some discrepancies between numerical results that prove the influence of modelling on the heat transfers. These results are a first step for enhancing industrial methodologies with advanced modelling. Additional work is still required to make these models usable in an industrial context.

In order to reach the objective of WP5, recommendations were extracted from the whole set of computations. These recommendations are expressed as “best practices” found out from the experience gained along the project.

1.4 Potential Impact

The Engine Representative Internal Cooling Knowledge and Applications (ERICKA) project has ended within its allotted time and has met its objectives in a successful manner within reason. On the one hand, the project has contributed to the improvement of the technologies and the understanding involved in the design of efficient blade internal cooling schemes. This represents a direct contribution to the reduction of fuel consumption through better engine efficiency, and the reduction of engine emissions. On the other hand, ERICKA has also indirectly contributed to the further development of the European aerospace industry through the establishment of new collaborative research links, acting as an ideal platform to educate students and to train staff, and by extensively disseminating its results. The complete benefits of ERICKA will become obvious in the next few years.

The technology and knowledge developed within the ERICKA workframe directly addresses one of the six activities of the ACARE SRA2 “Greening of Air transport” and the corresponding impact from the work programme under the same heading.

The biggest impact of ERICKA was to reduce fuel consumption through the improvement of the engine efficiency obtained by an improved turbine cooling technology. The consequence on fuel consumption and corresponding CO2 emission expected to be a reduction of 1% in SFC on top of other technologies developed or under development in other R&D programmes. In particular the impact is achieved not only for existing state-of-the-art engine architectures but also for future engines including innovative low NOx combustors

ERICKA consequently represents a major contribution to the first expected impact listed in the workprogramme in AAT.2008.1.1.3 Propulsion : “To reduce fuel consumption and hence CO2 emissions by 50% per passenger-kilometre”. The contribution of the engine to this 50% reduction corresponds to about 20%, these 20% CO2 emission reduction being equivalent to a 20% fuel consumption reduction. This reduction will be achieved by increasing the turbine efficiency according to two alternative choices: i) using a combustor running hotter than existing ones, ii) reducing the required amount of cooling air

ERICKA also addressed two additional expected impacts listed in the workprogramme in AAT.2008.1.1.3 Propulsion, i.e. the reduction of both i) NOX emissions by 80% in landing and take-off and fuel burnt in cruise down to 5 g/kg, and ii) the unburnt hydrocarbons and CO emissions by 50% according to ICAO standards. Certainly, the technology developed in ERICKA is of direct used to reduce NOx emissions by contributing to the technology which will enable future aeroengines to be fitted with lean burn combustors. This will be achieved through the reduction of the amount of cooling used in the turbine, enabling more compressor air to be used in the burner stage of the combustor which has the effect of reducing peak temperatures and hence NOx emissions. This also means that the size of the combustor could be increased which leads to improvements in combustion efficiency with attendant reductions of CO and unburnt hydrocarbons.

ERICKA also addresses a major concern of the air transport regarding gaseous emissions as expressed in the EC White paper ‘European Transport policy for 2010’. Engines using the technology developed in ERICKA will contribute to the reduction of greenhouse gas emissions that is also an action of ICAO.

Detailed Impact by Workpackage

WP1 – Optimisation of turbine cooling systems components

The ERICKA project enabled the four SMEs of WP1 to develop their simulation, design and optimisation software packages. New features have been integrated into the software developed by the partners and validated to robustly and efficiently parameterise, mesh, compute and optimise the different cooling configurations of the ERICKA project. These software are now much more accurate than at the beginning of the project and the results of the ERICKA project demonstrated their efficiency to simulate and optimise the various internal cooling systems. For example, NUMECA upgraded its in-house parametric blade modeller AutoBlade^{TM} with support for cooling features. The software can now handle realistic cooled blade channels including bends and ribs driven by a limited and dedicated set of simple geometrical parameters. NUMECA also highlighted the interest of using CHT in its CFD solution FINE^{TM}/Open. CENAERO demonstrated the efficiency and robustness of its massively parallel CFD code ARGO on static and rotating U-bend computations and CFS showed the robustness and rapidity of BOXERMesh, their proprietary fully-parallel and automated mesh generator. The optimisation platforms modeFRONTIER™ developed by EnginSoft, Minamo developed by CENAERO and FINE^{TM}/Design3D developed by NUMECA proved that they were able to cope with noisy, non-linear and expensive to compute cost functions through the use of surrogate models efficiently coupled to single or multi-objective evolutionary algorithms. The majority of the new features implemented during the project are already industrialised, thus giving the SMEs an immediate opportunity to support the industries towards the objective of “halve time to market”.

The optimisation workflows designed during the ERICKA project have been tested on several optimisations with different configurations. They proved to be flexible and robust while allowing the design space to be large enough to simulate innovative geometries. These workflows are valuable for designers to get an improved accuracy of the design process so that time consuming development and design re-works are minimised. The usage of such workflows will also accelerate the design process thanks to the use of fast codes and computer automation. Besides, dedicated optimisation strategies have been tailored to find innovative designs in the framework of the ERICKA project. These optimisation methodologies are thus available to be used by the SMEs to perform optimisation studies on engines under development. The expected impact is to reduce the design time of new engines not only in the aerospace industry but also in the energy and automotive industries.

By working together, SMEs of WP1 shared their knowledge, skills and experience in optimisation and numerical simulation of cooling systems. Moreover a valuable expertise has been gained by the SMEs in their collaborative work with industries and universities involved in the project. Indeed, SMEs have acquired and strengthened new competencies in parametric modelling, meshing, heat transfer computation and particularly in optimisation of innovative cooling system components. Thanks to this new expertise, the four SMEs will be able to reinforce their services for the aerospace, energy and automotive industries. They will also be able to further develop their software packages and corresponding specific expertise for a large dissemination of the project results that will profit to the European aerospace industry relying on these highly skilled SMEs, thus enhancing the existing blade internal cooling capabilities in Europe.

Results have already been disseminated throughout the course of the project in several scientific and industrial technical reviews and conference presentations such as the ASME Turbo Expo conference or the ERCOFTAC-Belgium yearly seminar. Final results will continue to be disseminated in such scientific reviews, journal papers and conferences. Results of the project have also been communicated in the SME’s internal newsletters and internal meetings.

The ERICKA project has been an ideal platform to train the SMEs’ staff to internal cooling systems since several researchers joined the project and had to be sensitised to these issues. Moreover, the optimised geometries and the associated optimisation methodologies will be transferred to the engineering departments of the industrial partners of the project through workshops to prepare the deployment and the exploitation of these results. These workshops fit into a process of continuous learning for the engineers of the industrial partners which will profit to the European industry development in this sector. SMEs will also interact on the outputs of the ERICKA project with their other partners and customers in the aeronautic, energy and automotive sectors. Eventually, it must be highlighted that the ERICKA framework was multicultural and well-diversified. The environment was therefore conducive to the development of international relationships thereby strengthening the European research community.

WP2 – Lead edge impingement engine geometry (RHR)

The biggest aimed impact of ERICKA is to reduce the fuel consumption through an improvement of the engine efficiency obtained by an improved turbine cooling technology. The high quality data gained during WP2 benefited in a further understand of the complex physics for leading edge impingement flows. Since the leading edge of an aerofoil has a high thermal load and is usually heavily cooled, it offers a great potential of cooling air reduction. However, the cooling system in blade leading edges is bound by tight physical constraints; hence it is challenging and requires new improved cooling technologies which were investigated in WP2.

Further validation of the heat transfer for this region can be transferred to engine level and lead to a reduction of required cooling air or allow for a hotter combustion exit temperature; both resulting in a higher efficiency of the engine. In collaboration with WP5 the CFD calculations could be validated with experimental data and lead to a better understanding of CFD predictions. It could be observed that CFD is able to predict most of the trends of the experiments correctly, even if the absolute level of the heat transfer was not matching. This gives further confidence to use CFD based optimization in the design process, which then would lead to a reduced design time.

In WP2 a huge high quality database of heat transfer measurements is obtained in both static and rotating test rigs at UNIFI and respective at RRUK. These results have been shown in periodic ERICKA reviews and were in addition published at peer-reviewed technical conferences (ETC and ASME). The optimized geometries tested at UNIFI, which showed a significant improvement in the blade cooling design, are of direct use for the industrial partners and can be transferred to real engine designs. In addition to the experimental data a huge database of test geometries, comparison of experimental to CFD results (in collaboration with WP5) and reports are produced and available to all partners. Also the work between the different partners within WP2 involved a strong relationship between academic partners, University of Florence, and industrial partners, namely Avio, RRUK and ALSTOM, which results in a better understand for the academic partner about industrial needs. Hence, ERICKA leads to a stronger relationship between academia and the industrial partners which will encourage academia to engage further in solving problems with a direct impact on the environment and the economy.

At the University of Florence one PhD thesis was fully dedicated to the ERICKA project. At ALSTOM multiple students have been hired for a diploma thesis or an internship and have been involved in the project work. Thus, the ERICKA project enabled young people to be involved in an EU project and gain a first practical experience for their future work life as engineers.

For Avio the work done in WP2 helped to validate the engine design with experiments and CFD and will allow an integration of the tested configurations into an engine design with lower cooling air consumption. The University of Florence gained further experience in designing and testing complex model with the transient TLC method and in addition gained a further understanding of the design needs of industrial partners. The transient rotating TLC measurements done at RRUK were done for an engine representative leading edge impingement configuration and for such a configuration no experiment data has been available before ERICKA. Therefore, the produced data is novel and helps to establish and validate more complex cooling designs for blade leading edges. ALSTOM will use the results gained in WP2 to further validate their engine design and improve the in-house correlations for rotating heat transfer, where part of the tested design features are already industrialized. The collaboration with WP5 also allows ALSTOM a revisit of the CFD strategy and confirmed the used best practices. In collaboration with WP1 further experience for optimizer strategy has been gained and all partner gained an increase of intellectual capital in the area, which is getting more and more important.

WP3 – Radial Passage engine geometry

Work package 3 has produced a large amount of heat transfer and flow data covering 4 aspect ratios from 1:2 to 1:6, three ribbing strategies (baseline, wrapped and cross-aligned), a number of Reynolds number values up to 110000 as well as static and rotating conditions. The data obtained have been reported at two levels of detail; detailed data, concentrating on the features of the channel, can be used for the benchmarking of CFD calculations. On the other hand data can be averaged over large sections of the passages to provide design information that had not been previously available. The impact on the partners is different: UUOXF have been able to improve their thermochromic liquid crystal (TLC) technique further and in particular have constructed a very useful analytical tool to take account of the 3D nature of geometries on which TLC measurements are made; furthermore, their studies on driving gas temperature have contributed much to the understanding of the nature of heat transfer in channels with such severe aspect ratios. IMPPAN have made a promising start on the use of TLC as a heat transfer measurement technique and have contributed greatly to the package through their hydrodynamic measurements using pitot tubes and hot wire anemometers. RRUK will benefit from the reporting of heat transfer and flow data for radial cooling channels which, though stylized, are often found in turbine components.

In summary, the various partners contributing to WP3 joined up with a varied set of interests and priorities. Designers in OEMs rely heavily on heat transfer and pressure drop correlations obtained from the analysis of data gained from tests on idealised cooling channels carried out over restricted ranges of conditions. Computational fluid dynamics (CFD) provides a useful tool to explore the design space around a particular design but it often lacks appropriate benchmarking against test data obtained at engine equivalent conditions. Academic institutions excel at the search for understanding of the dynamics of flow and heat transfer; this understanding can then be used to construct a framework on which more efficient flow cooling systems can be designed. The challenge involved in capturing more and more detail of the physics of the flow drives forward advances in measurement and computational techniques. The exposure of the OEMs and the academic institutions to a problem that straddles their respective requirements therefore enhances the knowledge base used by the designers while deepening the understanding of the underlying mechanisms of flow. The intellectual capital gained by the academic institutions is in some respects priceless as it would have been perfectly possible to spend similar amounts of money without a significant advance.

In terms of the wider impact of the project so far, there are a number of tangible and intangible outcomes. Leaving aside the recorded gains in knowledge of the subject matter, one important impact is the increase of the intellectual capital available to partner and suppliers thus:

Type of stakeholder	Description of intellectual capital gain
Industrial Partner	Technical Review meetings prior to the start of the project encouraged a critical review and internal dissemination of the existing background knowledge on the topic. Critical evaluation of existing capabilities both in terms of testing and analysis. The preliminary analysis triggered improvements of in-house analytical

	capabilities Contribution to the design and make process enhanced internal knowledge and understanding of design and manufacture constraints. Investment into the infrastructure of the facilities: Beyond the monetary investment, the main gain is the increased capability, knowledge and confidence on the part of the staff. Enhancement of the level of understanding of physical mechanisms; this will enhance the contribution of the staff involved to other projects.
Academic Partner	Many of the gains listed above will also hold for the academic partners. Additionally, an academic institution has access to and a requirement to provide undergraduates with projects. Thus, the institutions are able to explore the subject of the project in a manner that significantly improves their educational provision and encourages undergraduates to take post-graduate life. Alternatively, the institution has been able to benchmark acquired capability through comparison and discussion with other partners.
Supplier	Our supplier consolidated their established capabilities for design and manufacture. Experience on the specialised subject has been passed to a new generation of staff.

Another impact observed was the level of engagement of other members of staff with a less direct involvement in the project. It may be stated that however well recompensed a member of staff is, interest in a task is a particularly effective means to enhance engagement of a member of staff with their peers.

Type of stakeholder	**Description of improvement in level of engagement**
Industrial Partner	In the case of RRUK, the presence of the rotating rig amongst the engine stands generated much interest among the fitters and other supporting engineering staff. Many of those involved expressed pride to contribute to the assembly of a unique facility and the conduction of a safe and successful test programme. On a couple of occasions even the site security staff would approach the test facility to ask questions about the tests under way.
Academic Partner	Academic institutions would be expected to exhibit high levels of engagement. Involvement in an interesting project will maintain that.
Supplier	The staff at the supplier welcomed the level of interest shown by the

	industrial and academic partners.

Concerning the contribution to local communities, this was accomplished through the supplier engaged. The main supplier used by RRUK (and UOX) in WP3 was the firm involved in the detailed design and manufacture of the Perspex rotating test articles. The firm concerned are based in a deprived area of Manchester (NW England) and the chief design engineer explained to the Leader of WP3 that most of the cost incurred in the manufacture of the test articles was the detailed and painstaking manual work that the test articles required. Thus, the test requirement were translated into a significant amount of work for a local craftsman; in contrast, the cost of the materials and the machining of the test channels were a relatively small part of the total cost. The converse of that statement is that a relatively simple test article would not trigger many local benefits.

WP4 –U-bend and radial passage study

To improve the efficiency and performance of turbines a detailed understanding of heat transfer distribution and level in blade internal cooling passages is needed. WP4 focussed on blade internal cooling systems including U-bend and radial passages. This workpackage gathers the research aerospace institute ONERA, the University of Stuttgard with expertise in experimental methods and the industrial partners MTU and Snecma. A gender balance in the workforce was achieved in WP4. Tests were performed by USTU and ONERA using the static test rig at USTU and the BATHIRE facility at ONERA in close collaboration with MTU and Snecma. Experimental database for an high pressure and low pressure turbine configurations in both static and rotating conditions were achieved.

USTU generated high quality local heat transfer database for engine representative turbine blade internal cooling configurations using transient thermochromic liquid crystal method. Based on the lessons learnt during the ERICKA project several improvements to the test rig were implemented: the connection between the model and the air supply system was improved to avoid deflection of the flow and a bypass valve unit which reduces the heat losses in the supply passage was added. Thus the quality of the experimental device and accuracy of the measurements was increased during the project. Besides, USTU, in close cooperation with the partner MTU, defined during several bi-lateral meetings data evaluation, analysis and reporting procedures to allow direct comparisons of the experiments with numerical simulations as needed for numerical optimization processes and for WP5. A concise database of the starting and optimised geometries was achieved. The database contains spatially-resolved, area-averaged and line-averaged heat transfer data as well as pressure loss measurements for the five investigated Reynolds numbers. The results of the starting geometry obtained through the cooperation between MTU and USTU were published in the joint technical paper: “Investigations of heat transfer and pressure loss in an engine-similar two-pass internal blade cooling configuration” and was presented at the 10th European Conference on Turbomachinery in Lappeenranta, Finland.

ONERA used the rotating BATHIRE facility to generate experimental databases for an high pressure and low pressure turbine configurations in both static and rotating conditions. Unlike other ERICKA rigs BATHIRE facility is not using liquid crystal method but infrared measurements. ONERA encountered problems to post process the experimental results and assess the heat losses on the back side of the mock up. Thus several post-processing techniques were used to try to improve the initial results. Thanks exchange of ideas between partners during both phone calls and experimental workshops ONERA has benefited from working with other partners within ERICKA. The drawbacks of the rig have been identified and could be corrected to improve accuracy of the experimental data, so ERICKA will keep delivering benefits after the life of the project.

The experimental databases generated by WP4 were directly used to achieve calibration of the CFD codes to predict the flow and heat transfer in internal cooling systems including U-bend and radial passages. These databases were shared between the ERICKA partners. Some best CFD practises were identified thank comparisons between the experimental data and the numerical results obtained by different partners using various software.

Finally, the ERICKA project offered the opportunity to work with different partners (academia, institute of research and industrial partners) and therefore to share experience, skills and knowledge in blade' internal cooling systems topic. The project also involved working with student hired for an internship. This project was thus rewarding on both scientific and societal aspects.

WP5 – CFD Calculations

The computations emphasize the overall good behaviour of standard turbulence models when fine meshes are used. Although advance turbulence models resolving the Reynolds Stress Tensor can bring very interesting results, just as advanced turbulent heat flux models do, the usual practices provide reasonable results in terms of heat transfer. This important point highlights the maturity of current practices in Europe. The applicability however can be limited to preliminary design phases or optimisation processes.

The objective of WP5 is directly linked to the industrial need. In this respect, this work-package is one of the main outcomes of the project in terms of impact for industry. Everything was thought in order to serve this idea and to bring to the European community of Turbomachinery a large range of numerical activities from which best practices can be extracted.

First, WP5 proves that current practices relying on standard RANS turbulence models provide reasonable results. More interestingly, trends exhibited by experimental campaigns are generally well captured by these numerical procedures. Thus, the effects of the modification of the mass flow rate (Reynolds number), the channel aspect ratio, the internal geometry and the rotation rate were predicted in a satisfactory manner by the used codes (increase or decrease of thermal performance). One can expect that industry will take advantage of this situation and will increase its confidence in CFD, with yet improvable aspects to enhance the numerical accuracy of the design phases.

Second, there are two different perspectives of enhancement for the numeric. These two are respectively the use of advanced turbulence RANS modelling and the use of unsteady methods. They can be followed separately and simultaneously as they do not share the same characteristic time. The first one that relies on a finer modelling of the Reynolds Stress Tensor can be short-term solution. Models are almost operational but they require some more modifications especially for near wall treatments. This option would provide the ability to fully understand the behaviour of the fluid flow and not only on the surfaces. Indeed, some small changes and differences obtained in surface heat transfer can have a cause of greater magnitude within the entire volume of simulation. Furthermore, the use of highly complex geometries will require finer solving, by the use of finer meshes or more refined modelling. The second one is more prospective in the sense that it is not mature enough to be usually operated in an industrial context. Efforts have to be made by the scientific community to complete the existing models and bring fast, robust and intelligible models to the turbomachinery community. This more long-term activity has to be supported by industry but has a great potential.

Third, a best practices document was made to help the industrial community get directives to speed up the design of turbine blades. The use of standard RANS models with the specifications gathered allows quick preliminary design and optimisation process from known results. The accuracy of the results proved to be dependent on the geometries tested. RANS models provide accurate results for the geometries from WP4, however, for WP2 and some of WP3, these models catch the trends, but the support of experimental databases or complex simulations is necessary to obtain final accurate values.

Fourth, a significant amount of knowledge comes out of all the confrontations with experimental results. A large effort was focused on reproducing the experimental conditions within the CFD work. Indeed, a fair amount of simulations performed in ERICKA use transient methods, which needs some adaptation to simulate with stationary methods.

Fifth, partners have published a large amount of scientific results within the scientific and industrial community. Detailed papers are available within conference proceedings and journals, dealing with the progress that could be worked on with the validation of CFD codes, modelling techniques, precision of simulations and the understanding of complex fluid dynamics.

A fair amount of knowledge was produced within ERICKA, in all the aspects of the CFD work. These processes allow for providing a state of the art best practices for the industrial partners with the current CFD codes, the hardware available, and the knowledge acquired.

The obvious benefits of deeper understanding the performance of the CFD approaches considered here led to the identification of new challenges and future lines of research. The work conducted within ERICKA has exceeded the framework of WP5 itself. Indeed, it has been contributing to the definition of the CFD best practices adopted within WP1 and has been giving an insight into the physical phenomena involved within the configurations covered by the project. As a direct result, the measurement techniques used in the experimental workpackages 2, 3 and 4 were improved.

OTHER IMPACTS

Education and Training

In the life of the ERICKA project at least six students have achieved their PhDs and Master's degrees through ERICKA funding and some of these have now been employed in companies working directly on ERICKA. In addition new collaboration links have been established such as the partnership between the University of Stuttgart and Oxford University and in a twist of fate, one PhD candidates external examiner was a professor from another Institution who is also participating in ERICKA. Finally, the links between industry and academia have been strengthened through prolonged discussions and collaborative work between researchers.

Main Dissemination Activities: Inside the Consortium

Information has been disseminated inside the consortium on a regular basis. At the start of the project ARTTIC set up an internal website to disseminate information. All partners have access to this website and have made extensive use of it. The website is structured in such a way that the technical output is collected and stored under its relevant WP while another separate section holds the contractual documents all partners need for the smooth running of the project such as the GAM, budget or any amendments. All meetings are also recorded with a copy of the minutes and slides available throughout the life of the project. Each WP leader has been asked to present a technical update of their workshop every two months during the management meetings. This has allowed the management committee to also function as a technical steering group, anticipating risks and discussing any issue before It became a problem.

ERICKA has also made extensive use of internal workshops to disseminate information within the consortium. At the beginning of the project it was agreed that the consortium would meet every nine months, alternating the periodic reviews with an internal technical review. While this format proved to be effective at presenting results, it proved to ineffective to initiate discussions therefore in the middle of the second period it was decided to extend the meeting by an additional day and devote that time to workshop sessions. These were loosely structured around themes from the experimental and CFD Work packages and partners were encouraged to present specific problems or results such as, uncertainty and error estimation, experimental techniques etc. to each other in a detailed manner. This gave way to many varied discussions and contributed to partners agreeing on concepts which facilitated discussions in the next meetings. This in turn has resulted in a deeper understanding of the problems involved from a wide variety of perspectives, from which all industrial partners, SMEs and academia and research institutes have large benefited. The workshops also allowed for professional and personal relationships to develop and enabled new working collaborations between parties who previously had not done so, such as a now established collaboration between the University of Stuttgart and the University of Oxford.

Main dissemination activities: Outside the Consortium

When ERICKA started the Management Committee was very aware of the need to disseminate the ERICKA results to the wider world in spite of a very tight dissemination budget. MTU, the partner in charge of the dissemination Work Package prepared together with ARTTIC a dissemination plan and the corresponding procedures to put it into place. This plan was then presented and approved at the first technical meeting and has been in use ever since. The basis of the plan was to enable all partners to disseminate technical results as they became available and to target any major event where ERICKA could be presented as a project. This resulted in the participation of most partners to major conferences such ASME Turbo Expo or European Turbomachinery Conference (ETC) where a total of 14 papers were presented.

The ERICKA Management Committee also prepared a set presentation and poster which were approved by the Steering Committee and could be used by any partner to present the project to the general public. Thus ERICKA participated in such diverse forums as Aerodays 2011 or the KIAI-TECC (FP7 Funded Projects) public workshop in 2012 with the presentation and poster respectively.

In addition, ERICKA also set up a public website (www.ericka.eu) which describes the scope of the project in detail, gives information on the individual partners who make up the consortium and list the latest news to come from the project. This website is kept up to date and populated with any information which has been approved by the Steering Committee to be disseminated. In a google search the website appears on the first google page when searching for “Ericka” and has had hits from such varied places as Brazil, Canada or India.

2 Use and dissemination of foreground

2.1 Section A (public)

TEMPLATE A1: LIST OF SCIENTIFIC (PEER REVIEWED) PUBLICATIONS, STARTING WITH THE MOST IMPORTANT ONES

NO	Title	Author(s)	Title of the periodical or the series	Volume Issue	Publisher	Place of publication	Date of Publication	Relevant pages	Permanent identifiers (if available)	NO.
1	Heat Transfer Measurements in a Leading Edge Geometry With Racetrack Holes and Film Cooling Extraction	L. Andrei, C. Carcasci, R. Da Soghe, B. Facchini, F. Maiuolo, L. Tarchi, S. Zecchi	**Journal of Turbomachinery, (Transaction of ASME paper** GT2012-69581)	**Vol. 135, 031020, DOI: 10.1115/1.4007527 (Transaction of ASME paper GT2012-69581**)	ASME	USA	June 2012	pp1-10	**DOI: 10.1115/ 1.4007527**	
2	Experimental investigation on the heat transfer in a turbine airfoil leading edge region: Effects of the wedge angle and jet impingement geometries	F. Maiuolo, B. Facchini, L. Tarchi, and N. Ohlendorf	**European Turbomachinery Conference**,2013	ETC130-2013	ETC	EU	March 2013	Pp 1-10		
3	Experimental investigation on the heat transfer of a leading edge cooling system: Effects of jet-to-jet spacing and showerhead extraction	F. Maiuolo, B. Facchini, L. Tarchi, and N. Ohlendorf	IGTI-ASME 2013	GT2013-94759	ASME	USA	June 2013	Pp 1-10		
4	Numerical analysis of heat transfer in a leading edge geometry with racetrack holes and film cooling extraction	Luca Andrei, Antonio Andreini, Riccardo Da Soghe, Bruno Facchini and	IGTI-ASME 2013	**ASME paper** GT2013-94673	ASME	USA	June 2013	Pp 1-10		

		Stefano Zecchi								
5	Experimental Analysis of Gas Turbine Airfoil Leading Edge Cooling Systems	Francesco Maiuolo	PhD Thesis		DIEF-UNIFI	ITALY	April 2013	Pp 1-200		
6	Experimental investigation on the heat transfer of a leading edge cooling system with optimized racetrack inclined holes	C. Carcasci, B. Facchini, L. Tarchi, and N. Ohlendorf	IGTI-ASME 2014	**ASME paper** GT2014-26219	ASME	USA	June 2014	Pp 1-10		
7	Numerical analysis of Heat transfer in a leading edge geometry with racetrack holes and film cooling extraction	Antonio Andreini, Riccardo Da Soghe, Bruno Facchini, Luca Andrei, Stefano Zecchi	Turbo expo 2013	N/A	ASME	N/A	2013	N/A	GT2013-94673	no
8	Heat Transfer Measurements in a leading edge geometry with racetrack holes and film cooling extraction	Francesco Maiuolo, Carlo Carcasci, Riccardo Da Soghe, Bruno Facchini, Luca Andrei, Lorenzo Tarchi, Stefano Zecchi	Journal of Turbomachinery	Vol 135	ASME	N/A	MAY 2013	N/A	N/A	no
9	Advanced Evaluation of Transient Heat Transfer Experiments Using Thermochromic Liquid Crystals	R.Poser	Proc. IMechE, Part A: J. Power and Energy	Vol. 221	IMechE	England and Wales	2007	pp. 703-801		no
10	Analysis of a transient heat transfer experiment in a two pass internal coolant passage	D.Chanteloup	International Journal Heat and Mass Transfer	Vol. 47	Elsevier	Amsterdam	2004	pp. 5313-5322		no
11	A Data Reduction Procedure for Transient Heat Transfer Measurements in Long	J. von Wolfersdorf	Trans. ASME J of Heat Transfer	Vol. 120	ASME	New York	2008	pp.314-321		no

	Internal Cooling Channels									
12	A strategy for parameterization and optimization of turbine cooling channels	NUME	ASME Turbo Expo 2015	In preparation	ASME					no
13	Internal Cooling Channels Design Investigations: Aerothermal Optimisation of Ribbed U-Bends	Lott Philippe	IGTI-ASME 2013	Paper No. GT2013-95545	ASME		2013		http://proceedings.asmedigitalcollection.asme.org/ proceeding.aspx?articleid=1776111	no
14	Internal cooling channels design investigations : aerothermal optimisation of ribbed U-Bends	Philippe Lott; Emmanuel Chérière; Klaus Semmler; Laura Bénet	IGTI-ASME 2013	Vol 3A	ASME	San-Antonio Texas, United state	June3-7 2013			

A copy of these have been uploaded onto the electronic participant portal but not all journals were found.

TEMPLATE A2: LIST OF DISSEMINATION ACTIVITIES

NO.	Type of activities	Main leader	Title	Date/Period	Place	Type of audience	Size of audience	Countries addressed
1	Conference	C. Waidmann	10th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics	15.-19. April 2013	Lappeenranta, Finland	Scientific Community, Industry	60	All European and some overseas

2	Conference	C. Waidmann	1st International Congress on Thermodynamics	4 - 7 September 2011	Poznan, Poland	Scientific Community, Industry	40	All European and Russia
3	Conference	J. von Wolfersdorf	1st International Congress on Thermodynamics	4 - 7 September 2011	Poznan, Poland	Scientific Community, Industry	40	All European and Russia
4	Lecture	R. Poser	VKI Lecture Series 2010-05	May 2010	Brussels, Belgium	Scientific Community, Industry	80	All European
6	Lecture	J. von Wolfersdorf	VKI Lecture Series 2010-05	May 2010	Brussels, Belgium	Scientific Community, Industry	80	All European
7	Meeting	Raphael Van Liefferinge	NUMECA User Meeting 2011	15-16 November 2011	Brussels	Industry	100-200	worldwide
8	Conference	Van Liefferinge	ASME Turbo Expo 2015 (incoming)	15-19 Jun 2015	Montreal	Industry	>1000	worldwide
9	Conference	CHERIERE Emmanuel	ASME Turbo Expo 2013	6 June 2013	San Antonio, Texas	Scientific Community, Industry	30 pers.	All
10	Conference	LOTT Philippe	2012 ERCOFTAC-Belgium yearly seminar	7 December 2012	Université Catholique de Louvain, Belgium	Scientific Community, Industry	40 pers.	All
11	Presentation	LOTT Philippe	CENAERO Monthly Seminar	30 March 2012	Cenaero, Belgium	Scientific Community	50 pers.	Belgium
12	Conference (1 abstract)	R Pearce	ASME Turbo Expo 2015	15-19 June 2015	Montreal	Scientific Community	100	Worldwide
13	Conference (2 papers)	M McGilvray	10th European Conference on Turbomachinery, Fluid Dynamics and	2013	Finland	Scientific Community	100	Worldwide

			Thermodynamics					
14	Conference (2 papers)	M Mc Gilvray	ASME Turbo Expo 2014	2014	Germany	Scientific Community	100	Worldwide

2.2 Section B

No patent application have been made in the ERICKA Project

TEMPLATE B1: LIST OF APPLICATIONS FOR PATENTS, TRADEMARKS, REGISTERED DESIGNS, ETC.					
Type of IP Rights :	Confidential Click on YES/NO	Foreseen embargo date dd/mm/yyyy	Application reference(s) (e.g. EP123456)	Subject or title of application	Applicant (s) (as on the application)

Part B2

Please complete the table hereafter:

Type of Exploitable Foreground	Description of exploitable foreground	Confidential Click on YES/NO	Foreseen embargo date dd/mm/yyyy	Exploitable product(s) or measure(s)	Sector(s) of application	Timetable, commercial or any other use	Patents or other IPR exploitation (licences)	Owner & Other Beneficiary(s) involved
	Ex: New superconductive Nb-Ti alloy			*MRI equipment*	*1. Medical* *2. Industrial inspection*	*2008* *2010*	*A materials patent is planned for 2006*	*Beneficiary X (owner)* *Beneficiary Y, Beneficiary Z, Poss. licensing to equipment manuf. ABC*

All this information is reported in D6.2 and has been uploaded onto the EC the electronic participant portal

3 Report on societal implications

All this information has been reported online in the EC electronic participant portal.