RECENT ADVANCES IN THE SIMULATION OF GAS TURBINE SECONDARY AIR SYSTEMS

Colin Young and Peter D. Smout
Rolls-Royce plc, Derby, UK

Keywords: *Gas Turbine, Secondary Air Systems*

Abstract

Improvements in the performance of gas turbines have traditionally been achieved through step-wise developments in the technologies underpinning the design of the main gas path components. However, as the optimum design of these major components has been approached, the investment needed to extract further performance gains has, inevitably, risen sharply.

The unremitting demand for further improvements in engine efficiency, together with increasing competition from the US and the Far East, has led to gas turbine research being pursued on a much broader front. An important aspect of this broader research has been the drive towards making more effective use of the air consumed by the secondary system of gas turbines, which is used to cool critical components, seal internal cavities and manage end loads.

The participation of European gas turbine manufacturers in the EC-funded ICAS-GT and ICAS-GT2 research programmes has been instrumental in advancing European gas turbine secondary air systems technology in recent years. This contribution provides an overview of the achievements of the ICAS-GT research programmes and identifies some areas of secondary systems research that are expected to advance future gas turbine design.

1. Introduction

Improvements in the performance of gas turbines have traditionally been achieved through step-wise developments in the technologies underpinning the design of the main gas path components. As the optimum design of these major components has been approached, the investment needed to extract further gains has, inevitably, risen sharply.

The demand for continual improvements in the operating efficiency of gas turbines, coupled with the growing need to reduce the associated environmental impact, has led to recognition of the need to pursue gas turbine research on a much broader front, in order to optimise the engine package as a whole. In a modern gas turbine up to 20% of the core air flow is bled off from various compressor stages to facilitate internal cooling, bearing chamber and rim sealing, as well as axial load management. As this secondary air (Figure 1) makes no direct contribution to engine thrust, there is a strong incentive to reduce the quantity and quality of the air drawn off from the main annulus and to maximise its effectiveness.

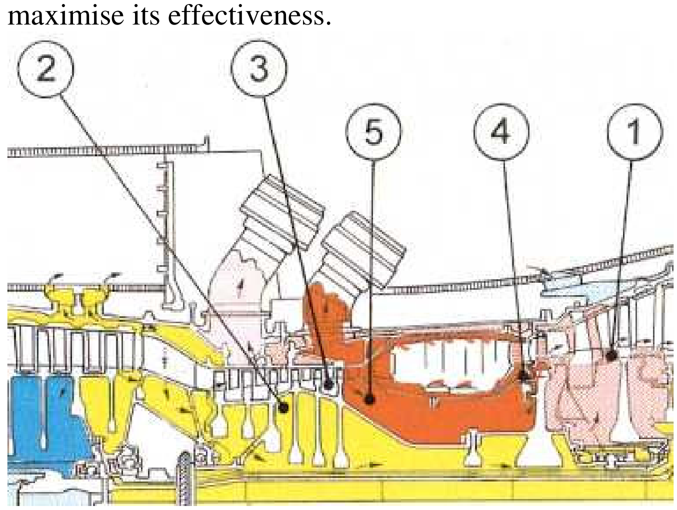

Figure 1: Internal Air System and ICAS-GT Research Task Areas

Until fairly recently, this aspect of gas turbine design has received comparatively little attention and therefore the potential for accessing direct and indirect performance gains from this area is considerable.

Direct performance gains arise from more effective use of the air consumed by the secondary system – thereby reducing parasitic losses. Indirect performance gains also accrue as a result of reducing spoiling in regions where the secondary flow re-emerges in the main annulus, e.g. at turbine rim seals.

The participation of European gas turbine manufacturers in the EC-funded ICAS-GT [1] and ICAS-GT2 [2] research programmes has been instrumental in advancing European gas turbine secondary air systems technology in recent years. This work has provided new insights into secondary air system design and has stimulated collaborative investigation into experimental and analysis techniques that would not otherwise have been possible. Exploitation of the derived technology is now well underway and the benefits of the research are being realised in the products of the participating European gas turbine manufacturer's products.

This contribution provides an overview of the achievements of the ICAS-GT/GT2 research programmes and identifies some areas of secondary systems research that are expected to advance future gas turbine design.

2. Programme Management

The research undertaken within ICAS-GT and ICAS-GT2 was, in each case, conducted by a consortium of 14 Partners, comprising four Universities and ten European gas turbine manufacturing companies.

In general, the University Partners within the Consortium assumed responsibility for the experimental investigative work, whilst the Industry Partners addressed the analysis and model development aspects of the work. It was recognised from the outset of the research that the complementary approach of experimental and theoretical investigation would be essential to the success of the Programme. More particularly, it was judged that biasing the experimental work towards the University Partners would result in test facilities and experimental techniques being established that could be exploited in a broad context and used to support and stimulate further research. At the same time, the choice to have the analysis work predominantly carried out by the Industry Partners was made to promote incorporation of new modelling methodologies into their design toolkits and processes.

The exception to this general approach was the development and first application of LES techniques to rotating cavity flow and heat transfer. In this case, it was considered more appropriate for a University Partner (Surrey University) to pilot the use of these procedures – although application within an Industrial context was subsequently demonstrated.

In each of the two ICAS-GT research programmes, the Partners were structured in five teams, under the overall co-ordination of Rolls-Royce. Each of the teams within the Consortium addressed discrete, but related, aspects of internal air system flow and heat transfer. As an example, the management structure adopted for the ICAS-GT research is illustrated below in Figure 2.

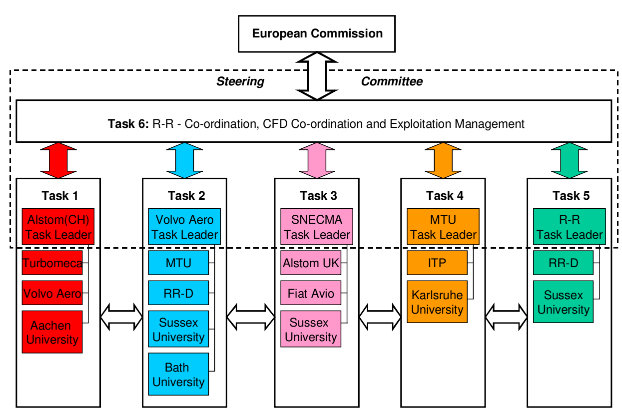

Figure 2: ICAS-GT Partners and Management Structure

Examples of the discrete research areas addressed by the two programmes are as follows:

- Turbine disc rim sealing of hot annulus gas into regions adjacent to highly stressed components.
- Rotating cavity flow and heat transfer, such as that between adjacent discs of a compressor disc stack.
- Flow and heat transfer within the stator wells of multi-stage axial compressors and turbines.
- The aerodynamic performance of pre-swirl systems, which are used to supply cooling air to high pressure turbine blades.
- The flow and heat transfer in core compressor drive cone cavities.
- Investigation of windage losses, arising from the fixing features that are commonly used in the internal cavities of gas turbine.
- Engine parts experiments – including investigation of transient thermo-mechanical behaviour following engine shut-down.

One University Partner was assigned to each team, and was responsible for conducting the experimental studies. Numerical investigations were undertaken by the Industry Partners, and included computational fluid dynamics (CFD) and finite element (FE) simulations of selected test cases. The objectives for each team were to develop a fuller understanding of the governing physics in their particular application area, and to embed this understanding in validated methods applicable to new engine designs. This contribution presents some selected results to illustrate the scope of the ICAS-GT Projects and the progress made.

3. ICAS-GT/GT2 Research Areas

Over the course of the ICAS-GT and ICAS-GT2 programmes air systems design technology was advanced in a number of distinct but related areas. Some examples of the advances made are briefly described in the following sub-sections.

3.1 Turbine Rim Seal Flow & Heat Transfer

In state-of-the-art aircraft gas turbine engines, higher-pressure turbine discs are purged with coolant air through the wheel space. This air reduces the thermal load of the discs and prevents the ingress of hot mainstream gas into the wheel space between the rotating disc and the adjacent stationary casings. The efficiency of the disc cooling and the lifetime of the rotor thereby strongly depend on the sealing efficiency. On the other hand, reducing the rim sealing would increase the turbine efficiency by reducing aerodynamic spoiling of the main annulus flow, with an associated decrease in SFC.

In this task, engine representative experimental data were generated by Aachen University from a single stage turbine test facility. Steady and unsteady pressure, temperature and CO_2 concentration measurements were made to determine the sealing effectiveness of various seal geometries, and to investigate mixing phenomena inboard of the seal [3]. These data were subsequently used to validate a series of CFD simulations undertaken by the Industry Partners. The arrangement of the test facility is illustrated in Figure 3, which shows the static pressure distribution at an instant in time, extracted from a 3-D unsteady solution of the entire flow domain, including the main annulus and the internal cavities.

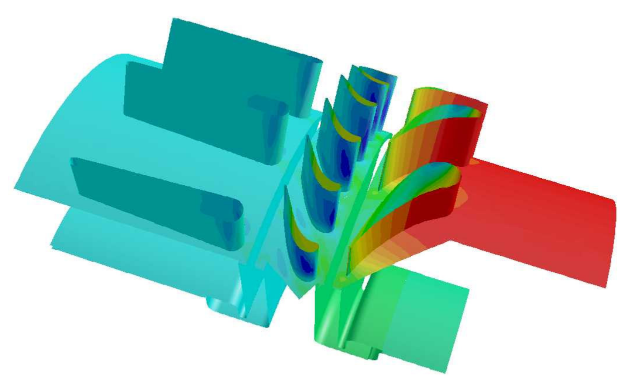

Figure 3: 3-D Unsteady CFD Calculations of Rim Seal Test Facility Flows
(shaded by static pressure where red is high, blue is low)

At lower purge flow rates, low frequency structures (LFSs) were detected within the forward wheel space experimentally. These previously unknown features were subsequently captured using 360º, 3-D unsteady CFD [4]. As can be seen from the contour plot shown in Figure 4, these structures extend some considerable distance into the cavity and have a significant effect on ingestion in this flow regime. At higher sealing flow rates, however, quasi-steady sector models have been shown to be adequate, greatly reducing the computational overhead associated with these complex simulations.

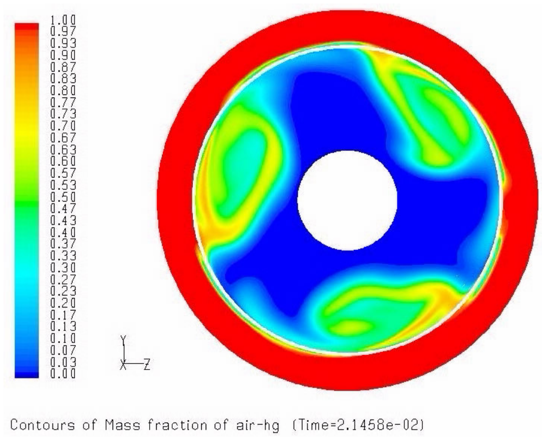

Figure 4: Rotating Structures Associated with Low Purge Flow Rates

The ingestion data obtained from the rim seal test facility have been correlated, following the methods outlined in [5] and have since been demonstrated to be applicable at engine operating conditions. In addition, the validated rim seal modelling methods developed during the course of the Project were subsequently used to design an improved rim seal. The resulting seal was demonstrated on the test facility and its performance was shown to be far superior to any of the seals tested earlier in the Project – indicating the potential for SFC improvements.

3.2 Rotating Cavity Flow & Heat Transfer

Cooling air for the air system is often extracted upstream of the high pressure compressor (HPC) and led through the annular passage between the shaft and the HPC disc bores to the turbine. Apart from the turbine cooling requirements, this air is needed to cool the HPC discs. Knowledge of the flow and heat transfer in the HPC rotating cavities is required for accurate disc stressing and lifeing predictions, but the physical mechanisms at work are not fully understood. ICAS-GT research showed that the flow can be dominated either by the axial throughflow under the disc bores, or by buoyancy forces arising from the disc metal temperature gradients [6]. During the ICAS-GT Programme, 2-D and 3-D steady and unsteady CFD modelling tasks were undertaken to establish the boundaries of these flow regimes. Figure 5 presents temperature contours from two 3-D unsteady simulations of the flow in the single cavity rig operated by Bath University for low and high rotational speeds. The stable, two re-circulating cell structure of the lower speed case is replaced by a more chaotic flow at the higher speed.

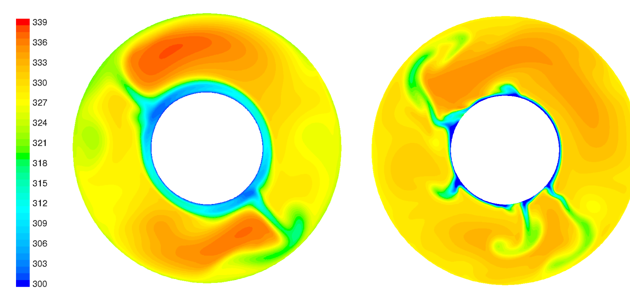

Figure 5: Rotating Cavity Flow Simulations

Low (buoyancy-dominated) and High (throughflow-dominated) Rotation Rates

At the conclusion of ICAS-GT substantial problems still remained in the prediction of these complex flows. In ICAS-GT2 large eddy simulation (LES) was applied to this class of flows for the first time, resulting in predictions – which have since been repeated in an industrial context. In both cases, the LES predictions replicate the measured flow and heat transfer data far more accurately than could be achieved using Reynolds-averaged Navier-Stokes (RANS) solvers. An example of these results is shown in Figure 6 below, which presents comparisons of the measured and predicted tangential velocity distribution within the Multi-Cavity rig operated by Sussex University. It can be seen from Figure 6 that the LES predictions are in much better agreement with the measured data than the corresponding RANS solutions attempted with different two-equation turbulence models.

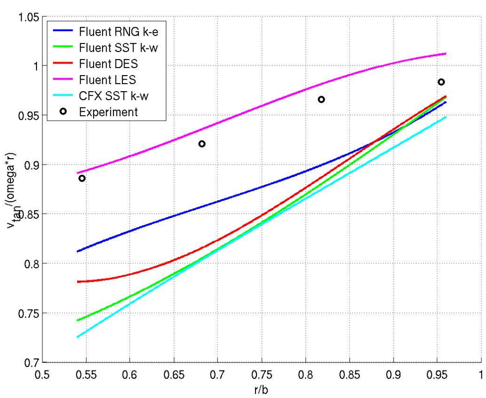

Figure 6: Comparison of LES & RANS-Based Predictions with Experimental Data

It is judged that the present research has provided an opportunity for the Industry Partners to apply LES methods to this class of flow some 3-5 years earlier than would otherwise have been the case and will permit better control to be exerted over tip clearances leading to sizable reductions in SFC.

3.3 Compressor and Turbine Stator Well Flow & Heat Transfer

Shrouded stator blades are commonly used to eliminate vibration and clearance effects in axial compressors and turbines. A trench known as a stator well has then to be provided in the rotor with the leakage of gas under the shroud being minimised by a labyrinth seal against the shroud periphery. A disadvantage of this technology is that the air in the stator well can reach high temperatures, due to viscous dissipation of energy, which results in the overheating of adjacent, highly stressed, rotating metal structures. The Industry partners in ICAS-GT evolved a method of simultaneously solving CFD and FE calculations for the flow and metal temperature gradients, respectively. This approach has been validated against experimental data generated by Sussex University from a purpose built, two-stage axial compressor and turbine facilities [7], with encouraging results – see Figure 7.

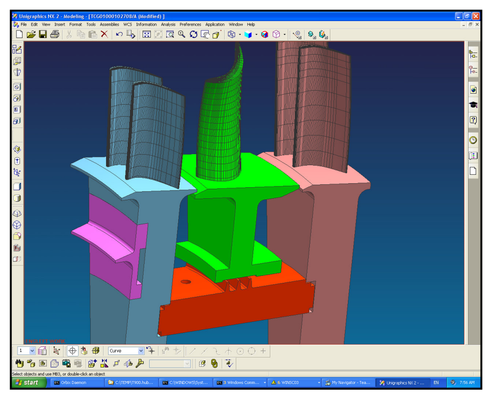

Figure 7: Finite Element Model of Test Facility Structure

An example of the model predictions obtained for a typical inter-stage seal clearance on a turbine stator well is presented in Figure 8 below.

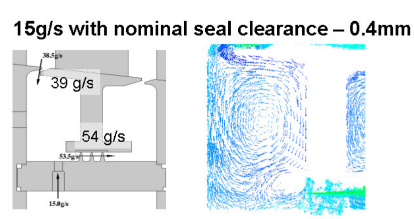

Figure 8: Example Turbine Stator Well Flow Predictions

The results of this work have provided clarity in terms of the current design best practice for compressor and turbine stator wells. Of potentially greater importance, however, is the discovery of a more effective cooling and sealing strategy, which achieves intimate contact between the secondary air supplied to the cavity and the critical components [8]. Exploitation of this novel cooling strategy will permit corresponding reductions in engine SFC to be accessed.

3.4 Turbine Pre-Swirl System Performance

High pressure turbine blades are typically supplied with cooling air at the lowest possible temperature, using a feed system known as "pre-swirl", in order to extend component life whilst minimising the impact on engine performance. Cooling air is expanded through nozzles located on stationary components and collected by rotating receiver holes in a cover-plate fixed to the turbine disc, or discharged directly to the blade roots. An example of a typical coverplate pre-swirl system is illustrated in Figure 9 below.

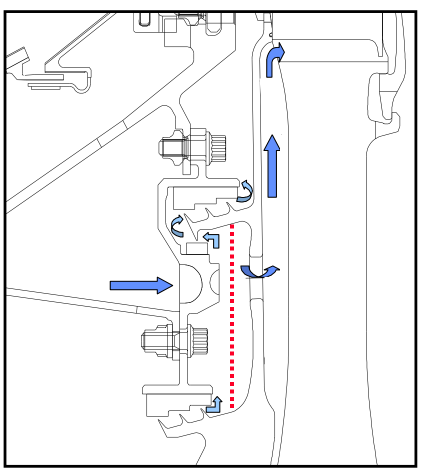

Figure 9: General Arrangement of a Typical Turbine Pre-Swirl System

In ICAS-GT, the flow characteristics of the stationary and rotating holes were established, and the effectiveness of the system was quantified for a range of geometries and pre-swirl seal flows. CFD solutions ranging from 2-D axi-symmetric to fully 3-D unsteady were run and were subsequently validated against experimental data generated by Karlsruhe University.

ICAS-GT2 built on this work by investigating the 3-D unsteady flows which are critical to pre-swirl system optimisation. More specifically, a variety of 3-D unsteady & quasi-steady modelling methodologies, e.g. [9], were developed and validated against the pre-swirl database – a typical example of these results is presented in Figure 10 below.

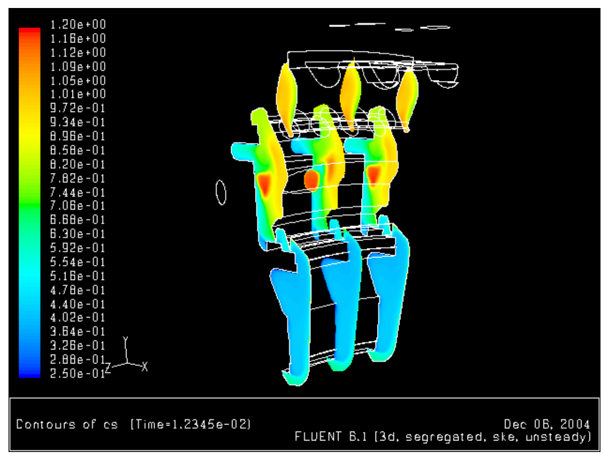

Figure 10: Contours of Swirl Fraction within a Typical Pre-Swirl System

The insight gained from this work has been documented in modelling best practice guidelines and pre-swirl system behaviour has been characterised within a 1-D design methodology, applicable to front-line engine design.

Finally, experimental and numerical investigations have been undertaken to demonstrate the secondary functionality of pre-swirl systems as a particle separator. This secondary functionality is of particular interest in industrial gas turbine applications, in which airborne particulates can result in the blockage of the internal cooling passages of turbine blades. In the limit, the blockage of cooling paths in critical components results in the failure of the engine. The occurrence of unplanned outages represents the ultimate inconvenience for the end user and undermines service provider's base/peak load matching strategies. Furthermore, such action inevitably leads to inefficient use of the service provider's resources.

The results of the ICAS-GT2 research support the assertion that, through appropriate pre-swirl system design, the additional functionality of particulate separation could be exploited, thereby reducing downtime and unplanned maintenance on industrial gas turbine power plant.

3.5 Compressor Drive Cone Flow & Heat Transfer

In addition to the air which flows through the HPC bore, it is common practice to channel high pressure air from the compressor delivery to the turbine via the cavity bounded by the HPC drive cone and the combustor flame tube casing. This air is subject to a significant degree of windage heating, which degrades its cooling effectiveness. If such temperature rises could be substantially avoided, cooling air flows could be reduced, or alternatively turbine entry gas temperatures could be increased, to realise SFC improvements.

A facility for investigating these conical cavity flows was built and commission at Sussex University around a modern gas turbine engine HP spool – illustrated in Figure 11.

Figure 11: Drive Cone Cavity Test Rig

Conventional pressure and temperature measurements were complimented by rotor temperature and heat flux measurements and LDA velocity measurements in the cavity to isolate the flow structures and quantify heat transfer rates. Encouraging agreement was achieved with 2-D and 3-D steady CFD calculations of the cavity flows, and with a conjugate prediction of the fluid and metal temperatures.

The 3-D steady CFD has been used to understand the influence of compressor blade over-tip leakage on boundary conditions at the cone cavity inlet, and the effect of a downstream row of compressor outlet guide vanes is being assessed by running a fully 3-D unsteady solution [10] and [11].

3.6 Rotating Cavity Windage Losses & Engine Parts Experiments

The same engine parts rig described in sun-section 3.5 was modified and re-instrumented and subsequently used to validate thermo-mechanical models simulating shutdown transients – which give rise to in service problems such as rotor bow.

In addition to the work referred to above, a dedicated windage test rig has been built and commissioned to facilitate investigation of losses associated with features and fixings commonly used in the rotor cavities of gas turbines, as shown in Figure 12.

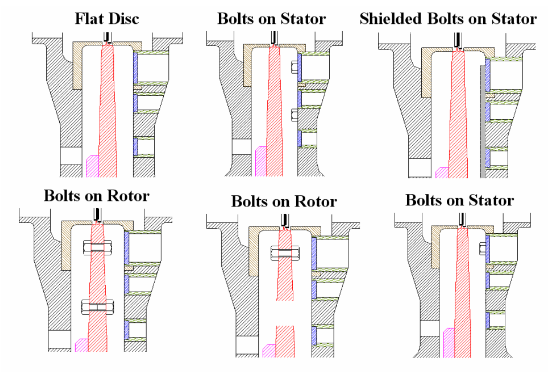

Figure 12: Examples of Protrusions Investigated for Windage Losses

The results of this work have been used to provide detailed insight into the nature of windage losses and the base data have been used to validate CFD predictions of windage loss for protrusions presented to the rotating cavity flow in different orientations. An example of this work is presented in Figure 13 below.

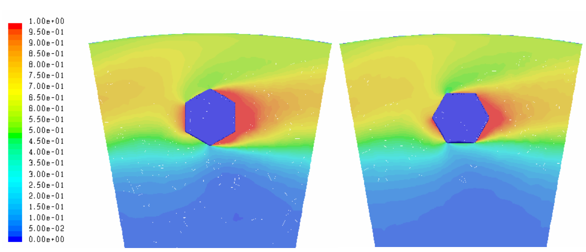

Figure 13: Typical CFD Predictions of Windage Losses

Swirl fraction contours for different bolt head orientations

Based on the results obtained, a suite of correlations has been compiled for use in front line air system design. These data will provide a firm foundation upon which to assess internal windage losses in future and will provide the impetus to work towards more aerodynamic cavity design in future engine projects, allowing the associated SFC reductions to be accessed.

4. Conclusions

The specific objectives of the ICAS-GT/GT2 research programmes have been met in that a truly representative, internal air systems experimental database has been established, and used to validate CFD and FE methods for predicting flows and heat transfer. These methods, which generally capture all the governing physics, are already being exploited by the Industrial Partners in their new product designs. In parallel with this, the University Partners have built and commissioned a range of engine-representative test facilities and have devised a range of new experimental techniques, e.g. 3-component LDA measurements with a rotating multi-cavity rig.

First use of LES has been demonstrated in rotating cavity flow – in both the academic and industrial contexts. The results obtained using LES techniques are far superior to those generated using unsteady RANS methods. Further work is needed in this area, however the results to date have been extremely encouraging.

Considerable progress has been made in simplifying complex modelling methodologies to make these calculation techniques accessible to a wider community. Characterisation of the behaviour of complex systems also permits significant reductions to be made in application times without unduly compromising the accuracy of the result. Exploitation of the comprehensive database and modelling methodologies will continue over the next few years, with the confident expectation that the reductions in SFC targeted by these Projects will be met or surpassed.

Consistent with the objective of optimising the engine package as a whole, future research efforts will be targeted towards understanding the interactions between the main gas path and the secondary air flows and the spoiling that results as a consequence.

To this end, a Consortium of research Partners - including many of the ICAS-GT/GT2 participants - has been assembled and its efforts will be coordinated by Rolls-Royce Deutschland. The so-called MAGPI (Main Annulus Gas Path Interactions) programme of research will be of four years duration and will commence on the 1st September 2006. Funded by the European Commission, MAGPI represents the spiritual successor to the ICAS-GT/GT2 programmes and its challenging targets will deliver the performance improvements needed to maintain the European gas turbine manufacturers position within a competitive market and to meet the urgent need to make effective use of non-renewable resources.

5 References

[1] Smout, P.D., ICAS-GT – EU Research into Gas Turbine Internal Air System Performance, *Proc. Air and Space Aeronautical Days 2000.*

[2] Young, C., Flow and Heat Transfer within the Rotating Internal Cooling Air System of Gas Turbines (ICAS-GT2), *Proc. Air and Space Aeronautical Days, 2006.*

[3] Bohn, D.E., Decker, A., Ohlendorf, N., and Jakoby, R., Experimental Investigations of the Influence of Sealing Air Mass Flow on the Adiabatic Wall Temperature Distribution on the Surface of the Rotor Blisk in a 1.5 Stage Turbine, GT2006-90452, *Proc. ASME Turbo Expo, Barcelona, Spain, 2006.*

[4] Jakoby, R., Zierer, T., Lindblad, K, Larson, J., De Vito, L., Decker, A., Bohn, D.E., and Funke, J., Numerical Simulation of the Unsteady Flow Field in an Axial Gas Turbine Rim Seal Configuration, GT2004-53829, *Proc. ASME Turbo Expo, Vienna, Austria, 2004.*

[5] .Johnson, B.V., A Turbulent Diffusion Model for Turbine Rim Seal Cavity Seals, United Technologies Research Center, Report UTRC 75-17, 1976.

[6] Alexiou A., Hills N. J., Long C. A., Millward J. A., and Turner A. B., Heat Transfer in a Gas Turbine HP Compressor Internal Air System, *ISROMAC-8, March 2000.*

[7] Scott R.M., Childs P.R.N., Hills, N.J. and Millward, J.A., Radial Inflow for Downstream Compressor Stator Well Cavities, *ASME Turbo Expo, Munich, Germany, 2000.*

[8] Dixon, J.A., Brunton, I., Scanlon, T.J., Childs, P.R.N., Wojciechowski, G., and Stefanis, V., Turbine Stator Well Heat Transfer and Cooling Flow Optimisation, GT2006-90306, *Proc. ASME Turbo Expo, Barcelona, Spain, 2006.*

[9] Snowsill, G.D., and Young, C., The Application of CFD to underpin the Design of Gas Turbine Pre-Swirl Systems, GT2006-90443, *Proc. ASME Turbo Expo, Barcelona, Spain, 2006.*

[10] Smout, P.D., Chew, J.W., and Childs, P.R.N., ICAS-GT: A European Collaborative Research Programme on Internal Cooling Air Systems for Gas Turbines, *Proc. ASME Turbo Expo, Amsterdam, The Netherlands, 2002.*

[11] Long, C.A., Turner, A.B., Kais, G., Tham, K.N., and Verdicchio, J.A., Measurement and CFD Prediction of the Flow within an H.P. Compressor Drive Cone, GT2002-30239, *Proc. ASME Turbo Expo, Amsterdam, The Netherlands, 2002.*

Acknowledgements

The financial support provided by the European Commission over the course of the ICAS-GT and ICAS-GT2 Projects is gratefully acknowledged. In addition, the technical contribution and commitment of all of our colleagues, within academia and industry, who worked within the ICAS-GT and ICAS-GT2 Consortia is also recognised.