GT2011-45*-)

HEAT TRANSFER IN TURBINE HUB CAVITIES ADJACENT TO THE MAIN GAS PATH INCLUDING FE-CFD COUPLED THERMAL ANALYSIS

Antonio Guijarro Valencia, Jeffrey A. Dixon, Attilio Guardini Rolls-Royce plc, Derby, UK

Daniel D. Coren, Daniel Eastwood TFMRC, University of Sussex, Brighton, UK

ABSTRACT

Reliable means of predicting heat transfer in cavities adjacent to the main gas path are increasingly being sought by engineers involved in the design of gas turbines. In this paper an up-dated analysis of the interim results from an extended research programme, MAGPI, sponsored by the EU and several leading gas turbine manufactures and universities, will be presented. Extensive use is made of CFD and FE modelling techniques to understand the thermo-mechanical behaviour and convective heat transfer of a turbine stator well cavity, including the interaction of cooling air supply with the main annulus gas. It is also important to establish the hot running seal clearances for a full understanding of the cooling flow distribution and heat transfer in the cavity. The objective of the study has been to provide a means of optimising the design of such cavities (see Figure 1) for maintaining a safe environment for critical parts, such as disc rims and blade fixings, whilst maximising the turbine efficiency by means of reducing the fuel burn and emissions penalties associated with the secondary airflow system.

The modelling methods employed have been validated against data gathered from a dedicated two-stage turbine rig, running at engine representative conditions. Extensive measurements are available for a range of flow conditions and alternative cooling arrangements. The analysis method has been used to inform a design change which will be tested in a second test phase. Data from this test will also be used to further benchmark the analysis method. Comparisons are provided between the predictions and measurements from the original configuration, turbine stator well component temperature survey, including the use of a coupled analysis technique between FE and CFD solutions.

INTRODUCTION

The requirement for ever more efficient gas turbine engines is leading to increased gas path temperatures, creating increasingly hostile environmental conditions for the adjacent turbomachinery and support structures. Cooling air systems are designed to protect vulnerable components from the hot gas that would otherwise be entrained into the cavities communicating with the gas path through the inevitable gaps between rotating and static parts. These cooling flows are bled from the compressor stages and reduce the engine efficiency, as they can represent around 20% of the total main gas path flow. These performance penalties manifest themselves in two ways, i.e. having a direct impact on thermodynamic cycle performance, resulting from imperfect work extraction in the turbines, and in the spoiling effect of the efflux at the point where it re-enters the turbine main annulus flow, causing a reduction in stage efficiency. It is desirable therefore to minimise these cooling flows, to levels consistent with maintaining the optimum component lives and the mechanical integrity of the engine [2], [10].

Furthermore, the gas turbine companies are under a continuous obligation to reduce undesireable emissions such as CO_2 and costs in order to be more competitive.



Figure 1 Typical turbine stator well

Under the auspices of the European Commission Programme for Research and Technological Development Framework 6 - Aeronautics and Space, a consortium of European gas turbine manufacturers and universities has undertaken a five-year project to address this specific issue. This project is called Main Annulus Gas Path Interactions or MAGPI [1]. There are 5 work-packages in this project and the first of these is specifically aimed at turbine disc rim cavity heat transfer and cooling optimisation, although it also supports other work-packages studying turbine efficiency. This work builds on a number of previous studies [6] and [7], both extending the methodology, and making use of additional test data used to further validate the method.

This paper presents a description of the up-dated rig test facility and the numerical analysis work performed by Rolls-Royce plc. The aim of the work has been to further advance the understanding of cooling flow, annulus gas interaction and resultant heat transfer, in the cavities adjacent to the main annulus in multi-stage turbines. In particular:

- The flow distribution and mixing which take place in the turbine stator well.
- The influence of the geometrical features such as cooling air entry holes and interstage seal clearance.
- The interaction between the disc rim boundary layer and the main annulus gas ingestion flows.

Finite element and computational fluid dynamics models have been created to further improve the analysis tool-set and best practices for turbine stator well design. A coupled analysis technique [15] has been further developed, which enables the direct application of convective heat fluxes generated in the cavity CFD solutions, to be applied to the finite element models representing the engine hardware. This modelling capability is being validated using measured data from the two-stage turbine facility sited at the University of Sussex.

Both steady-state and stand-alone FE and CFD solutions are presented as well as coupled FE-CFD numerical simulations, with comparisons to measured data. Alternative cooling configurations have been both modelled and tested for a range of cooling flow levels. Additional work has been done to predict the potential benefits of further design changes, which will also be tested in the rig as part of the MAGPI programme.

NOMENCLATURE

- GA General Arrangement
- h Surface heat transfer coefficient [W/m²·K]
- R Gas constant [J/kg·K]
- T Air Static Temperature [K]
- T_m Metal Temperature [K]
- Tt Air Total temperature [K]
- θ Non-dimensional temperature [-]
- θ_m Non-dimensional metal temperature [-]
- H Enthalpy [J/s]
- η_{eff} Thermal cooling effectiveness [-]
- η Isentropic turbine efficiency [-]
- p Fluid static pressure [Pa]
- pt Fluid total pressure [Pa]
- v Fluid kinematic viscosity, μ/ρ [m²/s]
- ρ Fluid density [kg/m³]
- m Mass Flow [kg/s]
- m' Non-dimensional mass flow [-]
- Ω Angular velocity [rad/s]
- u^{+} Shear stress near wall velocity $\sqrt{(\mu/\rho)(\partial u/\partial y)}|_{y=0}$ [-]
- y^{*} $% \left(N_{\tau},y_{P}/\mu\right) =0$ Non-dimensional wall distance, $\rho.u_{\tau}.y_{P}/\mu$ [-]
- i Subscript CFD model inlet
- Subscript CFD model outlet

THE TEST FACILITY

All tests were carried out at the University of Sussex, Thermo-Fluid Mechanics Research Centre. The test facility is shown in Figure 2. A brief overview is given here. Full details can be found in Coren et al. [3].



Figure 2 Turbine Rig Test Facility

The test section of the rig comprises a two-stage turbine, rated at 400kW, with a pressure ratio of approximately 2.5 at the design condition. Flow coefficients are 0.51 for stage 1 and 0.62 for stage 2, with work coefficients of 1.6 and 1.4 respectively. Main annulus air is supplied to the rig by an adapted aero engine driven compressor plant at 4.8 kg·s⁻¹, 3.3 bar absolute and approximately 170 °C. An Atlas Copco screw type compressor is used to provide the various cooling air supplies.

The cross section of the rig test section (Figure 3) has been designed to represent the key features of a turbine stator well. The turbine has also been designed to suit the subsequent FE and CFD analyses, with 39 nozzle guide vanes and 78 rotor blades for each stage. Thus the analysis models can be set-up at $1/39^{th}$ of the complete rotor/stator system.

Cooling Geometry and Supply

The cooling air is supplied to the hub region of the test rig via insulated transfer tubes. The rig has a split casing and is designed to allow rapid geometry changes.



Figure 3 Turbine rig section

The coolant may be introduced to the upstream stator well either radially through removable threaded inserts, or axially through removable cover plates with slot exits representing lock plate and blade fixing leakage paths. This arrangement allows 0, 13, 26 or 39 flow exits to be used at each entry point. These features are highlighted above in Figure 3.



Figure 4 Cooling Flow Paths

In order to achieve accurate metering of coolant to the stator wells, the delivery path is separated from the outer wheel space by a balance cavity sealed by two labyrinth seals. During testing this cavity is pressure balanced against the higher pressure coolant supply to prevent leakage; effectively forming a blown seal. The balance air is measured upstream of the rig, and vented from the intermediate wheelspace to prevent egress into the main annulus (see Figure 4). This arrangement also allows a known rate of egress to be specified. Cooling air flow rates are determined using hot film air mass meters.

Instrumentation

Turbine main annulus conditions are measured by temperature and pressure probes built in to the leading edges of the NGVs, avoiding the blade passage restrictions inherent with inter-stage probes. The turbine stator well and surrounding regions have been instrumented with metal and air thermocouples and static pressure tappings. The static thermocouples have been calibrated to produce an accuracy of ± 0.1 K in the measurements whilst the rotating thermocouples precision is of ± 0.2 K. The mass flows are measured with an uncertainty of ± 0.5 g/s.



Figure 5 Temperature Instrumentation

Figure 5 and Figure 6 provide an overview of the test section temperature and pressure instrumentation respectively.



Figure 6 Pressure Measurement Positions

NUMERICAL MODELLING

The overall objective of this study is to improve the modelling capability for turbine stator wells, i.e. cooling flow distribution and heat transfer management, with a view to optimising disc rim cooling and component life. It is anticipated that this will lead to further development of coupled FE/CFD modelling techniques [2, 10 and 15].

Indeed, this kind of analysis has been applied for the first time in the Sussex rig in order to validate the application as well as the CFD codes used.

This paper will detail the modelling from a standard FE model, a 3D CFD analysis and as a final step, the coupled analysis using the plug-in (communication library) SC89. The methodology and the models are described below.

Finite Element Thermo-mechanical Models

In preparation for this objective FE models of the rig (2D axisymmetric) and test section (3D), have been created. Appropriate solid properties, e.g. thermal conductivity as a function of the metal temperature, are modelled. The 2D model has been first used in the rig design phase to help establish operating temperatures, stress levels and clearances etc. See Figure 7.



Figure 7 Finite Element model

This model has been used to reproduce the measured temperatures indicated by the thermocouples from the test facility. The CFD solution can be used to replace the more usual, empirical correlation based, boundary conditions on the FE model, i.e. to establish the disc rim cavity convective heat transfer (heat fluxes). Together they form the coupled FE/CFD solution, which can then be validated against measured surface temperatures from the test rig.

An in-house computer code, SC03 [9], has been used to generate the finite element thermo-mechanical models. The coupling of this code to the commercial CFD analysis program Fluent was reported in [8]. This methodology, first developed by Verdicchio [16], subsequently enhanced in collaboration with the Universities of Sussex and Surrey [2,10,15], is now the chief means of validating the CFD method for convective heat transfer in the stator well. A further development allows us to use an in-house CFD code Hydra [11] for the determination of cooling flow distribution and convective heat transfer, in place of the commercial code.

Computational Fluid Dynamics Models

In parallel with the development of the test facility, it has been necessary to further develop the CFD modelling capability required to analyse the flow and heat transfer in the turbine stator well. It has now been shown that this requires both steady and unsteady calculations in full 3D, particularly where cooling efflux and annulus gas ingestion are finely balanced. However in other flow conditions, e.g. un-cooled stator wells, a steady solution is adequate. Moreover, Dixon et all [7] showed that a time averaged flow solution and the mixing plane approach are undistinguishable for the proposed cooling flow configurations. The suitability of a sector model chosen to keep the computational requirements within the capability of available computer facilities is still not certain, but initial results are encouraging. The test facility was developed with this limitation in mind and the current CFD model is 1/39 of the full rotor/stator system, i.e. incorporating 1 NGV, two rotor blades, one cooling air hole or one 'lock-plate' slot, see Figure 8.



Figure 8 CFD analysis model

The computational domain was meshed using the inhouse software PADRAM (Parametric Design and RApid Meshing). This automatic meshing tool produces high fidelity fully structured and unstructured meshes. Grids of around a million cells per vane plus four million cells in the stator well cavity were created. A detail of the rotor 1-stator 2 labyrinth seal mesh is shown in Figure 9. The total mesh size is just over 9 million cells.

The domain was split in three zones separated by mixing planes. The first zone contained the first NGV, the second both rotors linked by the cavity radially inboard, and the third region contained the second NGV in between both blades. Each vane was meshed separately. In addition, the rotor vanes included half the cavity in each of the rotor domains.



Figure 9 Interstage rim gap mesh

At this stage and with the available IT resources it was necessary to produce a cut down CFD model that made the computations more affordable, verified against the complete model and with boundary conditions validated against the available test data.



Figure 10 Extent of the cut-down model

The extent of the cut-down model can be seen in Figure 10. The mesh uses the same topology and is equivalent to the aforementioned grid in terms of near well distance and cell count. Note that the extent of the annulus domain has been increased in order to include the inter-stage rim gap seals between both discs and the stator. Nevertheless, the cooling duct has not been included in the model for the analysis presented in this interim paper. The total number of cells has been reduced to 4 million, making the calculations more affordable.

At this stage the in-house CFD analysis code Hydra [11] is used, Note: Other partners in the consortium are using different commercial CFD codes [5].

The inlet and outlet boundary conditions for the model were taken from the test data and applied where available.

Turbulence Modelling

The Spalart-Allmaras turbulence model has been chosen for these calculations. Rolls-Royce plc has extensive successful experience of using Spalart-Allmaras for main gas path flows, as well as for fluid flow and heat transfer in secondary flow cavities, using the in-house code. This has allowed benchmarking of the turbulence models used by the MAGPI partners.

The CFD mesh has been adjusted to ensure that the majority of the y^+ values are within the recommended range for the Spalart-Allmaras turbulence model, (between 20 and

40), with the standard wall treatment of Spalding [12]. In the small gaps, such as the labyrinth seal and rotor/stator rim, there are difficulties in maintaining this constraint, and in some places the y+ values are within the viscous sub-layer region. In that case Hydra mimics the low Re Spalart-Allmaras model [13], by ensuring u+=y+. Such functionality (i.e. Hydra reverting back to viscous sub layer), is useful in handling the trailing edge flow, where the scale of the circulation after the separation, requires high mesh resolution that can only be met by having y+<5.

Wall total temperatures, torque and power monitors were set to ensure convergence in the models. It has been noticed, however, that to achieve full convergence in Hydra with this turbulence model, a larger number of iterations is required. The Spalart-Allmaras model was originally created to better capture the separation in the boundary layer, selecting this to get more accurate predictions of the heat transfer in the rotor walls. However, this model, having just one additional turbulence equation, has been observed to struggle to drive the heat to the core of the recirculation vortexes, such as in the middle of the downstream cavity. Hence, instead of being able to achieve a converged solution within a couple of thousand iterations, typical for main annulus compressor or turbine calculations, this model, including the stator well, required about 20,000 iterations. Figure 11 shows an example of these difficulties to converge the energy equation for one of the evaluated and validated cases. Contours of absolute total temperature in a mid theta plane in the cavity have been plotted.



Figure 11 Rotor 1 trailing edge model

Coupling methodology

The work presented here is based on the coupling between the in-house software tool, SC03, and the CFD code, HYDRA. A brief summary of the method is given below, whilst a more detailed description is given by Verdicchio et al. [16] for any CFD code application.



Figure 12 Cycle definition example

Within SC03, the user defines a transient analysis by specifying an analysis cycle for the particular geometry

under investigation, i.e. the evolution of a set of environment parameters through the time span simulated. These are user-input parameters and include mass flow rates, operating temperatures and pressures (a typical example is given in Figure 12). The code is time marching and as such, SC03 needs an initial condition i.e. an initial metal temperature distribution for each node in the solid. To solve the heat equation in the solid, SC03 uses an implicit time discretisation and a Newton-Raphson solver [15].



Figure 13 Schematic representation of the coupling process

The thermo-mechanical coupling process is schematically depicted in Figure 13. First, the system calls the fluid solver, HYDRA in this application, passing to it the current values of boundary temperatures T^n (the superscript *n* indicates that these quantities refer to the time t^n). An important assumption is made here: as the fluid response to a change of operating conditions occurs on time scales much shorter than that appropriate to the metal heat conduction, the influence of unsteadiness in the fluid is expected to be negligible, and steady CFD calculations can be employed using the boundary conditions passed by SC03 [10,15]. More precisely, the CFD solver applies the wall temperature boundary conditions passed from SC03 and runs a steady state case to find the solution corresponding to these prescribed wall temperatures. After a certain degree of convergence has been achieved, based on user inputs, HYDRA outputs the heat fluxes qⁿ computed on the boundaries. These heat flux values are returned to SC03, which runs the Newton-Raphson solver to obtain an improved estimate of the temperature field at time tⁿ. The CFD solver and the FEA Newton-Raphson solver loop is then repeated until the solution has stabilised to within a user defined tolerance.

When the temperatures are stabilised (typically this requires around five iterations) the analysis moves to the next time step in the analysis cycle. The coupling communications are controlled by the plug-in (SC89) of the SC03 program. It is within SC89 that the user specifies one or more coupled walls, outlining a CFD domain, which may cover part or the whole of the finite element model.

Note that this is an alternative approach to the conjugate heat transfer analysis proposed by other partners like da Soghe et all. [5] in recent works relating to the same test facility.

OBJECTIVES

The objectives of the modelling capability are to enable the optimum level and placement of cooling air for disc rim environmental control (cooling). This includes the determination of cooling flow re-ingestion, i.e. from the upstream efflux at the front of the stage 1 disc rim, some of which is drawn into the turbine stator well cavity. The most efficient use of the cooling air is to be achieved by judicious placement of this air and the optimisation of the stator well geometry. Each of these aspects is being investigated by partners in the MAGPI Work-package 1 consortium, with data being generated on the test rig at the University of Sussex, to validate the methods used. Some of this work is still to be completed and will be published at a later date. However some results are available now and these will be presented in this progress review paper.

The disc surface thermocouple measurements facilitate validation of the method for this application, now that the fully coupled FE/CFD analysis capability has been established.

RESULTS

Cooling delivery options Air Mass-flow Level

An investigation into the effects of cooling air mass flow level has been carried out with the multiple reference frame CFD models (steady, adiabatic solution) recognizing that there will be some quantitative limitations on the predictions of gas ingestion, but anticipating that qualitative results will give a good indication of trends. Table 1 shows the cases run during the full test matrix:

Test type	lock plate slots	drive arm holes	Cooling flow (% of MA)	Completed
Temperature, pressure and gas concentration tests	0	39	0.61	\checkmark
			1.12	\checkmark
			1.43	✓
			1.84	✓
		0	0.61	✓
	39		1.12	 ✓
			1.43	\checkmark

Table 1 Cooling Flow Level Investigation

The range was selected in order to cover different levels of efflux to the main stream in the search of an optimum cooling flow configuration. The egression of this flow will have an impact in the turbine performance which was also studied in support to other Work Packages within the project [1]. In a first phase, four levels of coolant were tested. The minimum flow rate resulted in ingestion of hot gas to the cavity, the next to minimum level gave transient ingestionegression and the other two flow rates gave a high level of efflux through the rim seal between the rotor 1 and stator 2. This led to the decision of discarding the highest mass flow rate for the lock-plate leakage slot tests, supported also by the CFD analysis. Coren et all. [4] discuss the test matrix in more depth.

In this paper, pre-test CFD predictions have been added to the validated results shown in [7]. Alternative geometries have been designed for the Sussex rig following the Design of Experiments carried out by da Soghe et all. [5]. The aim of including these geometries is to validate the code against dramatic changes in cooling delivery in the search of the best cooling performance. Figure 14 depicts a detail of the GA of the different geometries.



Figure 14 Cooling configuration examples

A measurement of this cooling performance has been agreed between the partners, as described in equation (1), namely thermal cooling effectiveness.

$$\eta_{eff} = \frac{T_{hot} - T_{wall}}{T_{hot} - T_{cool}} \tag{1}$$

where *hot* denotes the inlet total temperature to the rig in Kelvin, and *cool* the relative total temperature of the cooling flow at the chosen delivery option inlet.

Figure 15 shows contours of adiabatic thermal effectiveness on the rear surface of the stage 1 disc rim, for a range of cooling flows (supplied at the drive arm holes from 0 to 1.84% and the lock-plate slots position from 0 to 1.43%). Note that the disc face benefits from the configurations that look for breaking the thermal boundary layer. This becomes clearer for the constant rate of 0.61% of main annulus flow.

No Cooling 0.61% MA 1.12% MA 1.43% MA 1.84% MA



Figure 15 Predicted Effect of cooling flow on Thermal Effectiveness



Figure 16 Adiabatic Thermal Effectiveness with different cooling flow levels

Figure 16 plots area average values of thermal effectiveness for different levels of mass flow. The chart shows how thermal effectiveness on the rear face of the stage 1 disc rim varies with cooling flow level; this for the three alternative cooling flow entry locations, i.e. the conventional drive-arm holes and the lock-plate slot locations. In addition to that, a new flow delivery option, angling the drive arm hole 22.5° towards the aforementioned rear face of the first rotor disc has been analysed. It is evident from these predictions that the lock-plate entry position requires less cooling air for a given disc rim temperature (compared with the drive-arm hole position). Also shown is a calculation of turbine stage efficiency, which follows an expected trend in reduction of stage efficiency with additional cooling air efflux. The isentropic efficiency of the turbine rig is calculated automatically by HYDRA using the expression described in equation (2).

$$\eta = \frac{\left\{\sum_{i=inlets} (\dot{m}H_0)_i - \sum_{i=exits} (\dot{m}H_0)_i\right\}}{\left\{\sum_{i=inlets} (\dot{m}H_0)_i - \sum_{i=secondary_{exits}} (\dot{m}H_0)_i - (\dot{m}H_{0_{ideal}})_{main_{exit}}\right\}}$$
(2)

where the ideal exit total enthalpy is calculated by isentropically expanding each gas stream to the total pressure of the mainstream rotor exit. The resulting values have been included in the chart in the text boxes, coloured by case. The efficiency of the turbine drops with the mass flow (due to the spoiling effect on the mainstream gas) and strongly depends on the cooling flow configuration.

The additional case, namely angled hole, sits between both lines. In any of the configurations the S-shape curve plotted becomes flat for mass flow rates over the labyrinth seal demand. This can be taken as a limit of benefit, in the amount of coolant that effectively contributes to the reduction of temperature.

Figure 17 shows a comparison of normalised temperature, θ , for a cooling air supply level of 1.12% for each of the entry points, based on the measured data. The results confirm the benefit of introducing the cooling flow at the lock-plate slot position (Coren et all show further experimental evidence [4]).



Figure 17 Measured Thermal Effectiveness

FE-CFD coupling

Further development of the in-house code HYDRA in parallel with its implementation into SC03 via the communication library SC89 has allowed the replacement of the 'hand-crafted' thermal boundary conditions in the FE model by a simpler and more robust analysis technique.

In this paper, the results for a case without cooling in any of the possible delivery options are presented. The test data for validation of the method was extracted from the reingestion experiment carried out by the University of Sussex and depicted in Figure 18. A full description of this test and its results has been published by Eastwood et all [8].

In brief, a small quantity of mass flow (0.3%-0.8%) of the main stream flow) is injected in the front cavity. The expected outcome of the experiment was to evaluate the amount of mass flow re-ingested in to the stator well cavity). The test data showed that just below an 18% of mass flow will be swallowed by the rear cavity. This translates in 0.09% of the main annulus mass flow and a 9% of the labyrinth seal demand.



With these considerations, the problem was modelled assuming a case with no coolant or air other than the main stream.

SC03 model set up

The existing SC03 model [7] was modified by the addition of a CFD solution for the disc rim region, based on the measured inlet conditions to the rig and then matched to test data around the stator well cavity. In the regions where

CFD was not available, convective boundary conditions where applied based on empirical correlations and a matching factor was applied.

The cycle defined in SC03 for the time marching steady-state calculation was a ramp from t=0 s to t=60 s and a flat segment until t=5000 s.

In a first stage, as detailed in [7] the CFD was used to extract heat transfer profiles that were then applied to the FE model. The temperature profiles applied to the walls of the CFD model were created based on the discrete measurements from the thermocouples placed in the rig. This procedure helped to speed up the matching task as it reproduced the actual power input to the model from the rotor, drive arm and stator foot walls. The same will apply to the coupled walls.

In the main stream, a so-called convecting zone was applied at the aerofoils, i.e. a thermal boundary condition equivalent to an area of infinite heat capacity. The absolute total temperature applied to the stator vanes came from CFD which, as showed in previous papers [7], matched up very accurately to the experiments. At the blades, no test data is available and the predictions calculated by HYDRA were directly applied.

The matching exercise of the boundary conditions of the front and rear cavities could be solved in some few runs of the SC03 model. The predicted temperature distribution in the model is shown in Figure 19.



Figure 19 Test section temperature

Table 2 compares measured temperatures, (in a nondimensional form $\theta = T_t/T_{ti}$), minus predicted values at the thermocouples highlighted in Figure 22. Temperature predictions are within 2% of the measured values at all locations.

	Test (M)	SC03-HYDRA	Delta	
		(P)	(P-M)	Delta (%)
	θ	θ	Δ	Δ (%)
MP067	0.726	0.724	-0.002	-0.178
MP070	0.755	0.749	-0.006	-0.606
MP073	0.803	0.801	-0.002	-0.161
MP079	0.841	0.846	0.005	0.468
MP097	0.821	0.808	-0.013	-1.286
MP106	0.707	0.710	0.003	0.276
MP112	0.675	0.676	0.001	0.085
MP124	0.683	0.680	-0.003	-0.278
MP127	0.659	0.663	0.005	0.457
MP015	0.687	0.689	0.001	0.145
MP017	0.714	0.697	-0.018	-1.761
MP024	0.682	0.683	0.001	0.149

Table 2 CFD/FE prediction vs. measurements

CFD validation

In parallel, the air temperature predictions in the stator well cavity were compared to the test data. The thermocouple positions are detailed in Figure 20.



Figure 20 Air thermocouple location

A range of validation for the whole text matrix in the test phase 1 was carried out including comparison between adiabatic runs and non-adiabatic calculations using temperature profiles created based on the test data. In this paper, only the results for the un-cooled non-adiabatic configuration described above have been included in Table 3. A complete set of validation results against test data can be found in [7]. The table shows very accurate matching between the predicted air temperatures and the measurements of the air thermocouples.

		Test	HYDRA		
	MP No	θ	θ	Δ	Δ (%)
	MP 025	0.8907	0.8871	-0.0036	-0.3612
N	MP 026	0.8901	0.8912	0.0011	0.1129
	MP 027	0.8950	0.8939	-0.0011	-0.1129
8	MP 028	0.8964	0.8935	-0.0029	-0.2935
Ö	MP 029	0.8968	0.8937	-0.0032	-0.3160
2	MP 030	0.8948	0.8953	0.0005	0.0451
	MP 032	0.8903	0.8892	-0.0011	-0.1129

Table 3 Uncooled non-adiabatic configuration results

Coupling set up

The coupled analysis is set up in the graphical interface of the communication library SC89 into the main code SC03.

The convective thermal boundary conditions remain unchanged in the FE model. However, all the convective boundary conditions inside the cavity were deleted and replaced by coupled boundary conditions (one per CFD boundary wall), that exchange the heat fluxes and wall temperatures with HYDRA, as described in previous sections. The initial domain temperature, T^n at time t=0, was assigned to be ambient temperature.

Except at the walls, the CFD boundary conditions are not changed from the previous models. While the option exists within SC89 to use temperatures and mass flows from the FE model at the CFD inlets, in this case, for model stability, the inlet temperature was kept constant during the cycle.

At this stage there is just one additional parameter to set up, which is the number of iterations per CFD call, i.e. the convergence level in the CFD, before passing the information to the FEA model. The maximum number of iterations was set to 100 for every CFD call. A sensitivity study was carried out, running the models to a certain convergence criteria until the maximum change in temperature between solutions was less than 1 K. This did not ensure the full convergence of the CFD model in a single CFD call. However, SC89 calls HYDRA until it ensures the CFD model convergence checking that the temperature difference between T^n and T^{n+1} is lower than 0.1K before going to the next time step and therefore producing the required number of iterations for full convergence, as discussed in previous sections.

Coupling results

The calculations were conducted on a 64 bit IBM E5450 Processors at 3.00 GHz Linux cluster on 16 processors. The running time was between 3 to 4 days for the steady state calculation.

Figure 21 shows combined contours of nondimensional metal temperature in the metal $\theta_m = T_m/T_{ti}$ (right hand side key) and the air $\theta = T_t/T_{ti}$ (bottom key) for the coupled solution. The combined illustration allows us to make a simultaneous description of the interaction between the fluid and solid as the coupling process sees it.

The air ingested in the cavity is cooler in respect to the rotor disc as it has lost energy in the turbine blade. When entering the cavity, the fluid sticks to the stator foot wall and will heat up the static part of the cavity. Then this air recirculates in the cavity heated up by the viscous work done by the rotor as well as the convection from the rotor disc. The mass flow through the recirculation has been calculated at almost twice the flow predicted by the free disc entrainment correlation, 1.7 % of main stream flow against $\dot{m} = 0.88$.

The calculations showed that 0.79 % of the main stream flow will be demanded by the labyrinth seal. This air is then heated up in the labyrinth seal before being egressed back to the main annulus. Note that this hot air created a radial gradient of temperature in the stator foot, being hotter at the inner diameter than at the blade platform. This value is in good agreement with the measurements reported by Eastwood and Coren [8,4] as well as with the predictions obtained by other CFD calculations carried out by other MAGPI partners.



Figure 21 Non dimensional temperature contours in the metal and air.

The temperatures predicted by the coupling have been compared in a number of thermocouple locations in the rig

against the available test data and the original matched calculation. The thermocouples have been split into three groups as shown in Figure 22. The orange locations correspond to thermocouples in the disc or where the heat conduction from the SC03 boundary conditions will dominate the solution. The next group of thermocouples, in black, are located in rotating walls in the fluid whilst the positions coloured in green define the static metal thermocouples.



Figure 22 Metal Thermocouples in the rig





Figure 23 Rotor metal temperatures

As anticipated, the first group of thermocouples is mainly dominated by the conduction in the disc and the thermal matching of the boundary conditions in the front face of the disc. This is confirmed by looking at the comparison between the manual linking and the coupled calculations. The temperatures are in very good agreement with the test data.





Figure 24 Rotor wall metal temperatures

The coupled predictions line up again very well with the test data. The discussion about the results in this subsection must be focused in the thermocouples placed in the rotor rear wall, MP091, MP097 and MP100. The gradient between the bottom of the cavity, MP091 and MP100 has not been accurately predicted. The test data shows that the thermocouple located in the blade platform, MP100, is colder than the other two in the disc.

Nevertheless, in absolute values the prediction is very accurate, especially in the drive arm, where the differences are less than the thermocouple accuracy.

- Stator foot temperatures.

The biggest differences have been found in the static wall temperatures although again fairly accurate. Note in Table 2 that the biggest delta was observed at thermocouple MP017 equal to a 1.8% difference, and the coupling has now offered a better answer. At the outboard region, close to the platform wall, the predictions match up very well the measurements being within the thermocouple accuracy in most on the cases.

At the bottom of the stator foot, differences bigger than 1 % have been observed. This suggests room for improvement in the solution of the labyrinth seal as the drive arm thermocouples, MP109 and MP115 were very well matched, as discussed in the previous section. As Eastwood [8] has shown in the latest tests in the experimental facilities, the labyrinth seal running clearance might be bigger than the cold build clearance. This will have an impact for cooled cases and has a potential benefit to reduce the observed deltas between measurements and coupling predictions.



Figure 25 Stator foot metal temperatures

CONCLUSIONS

Considerable progress has already been made in the MAGPI Work-package 1 partnership.

A good quality set of test data is available for the twostage turbine and stator well cavity, covering two alternative cooling configurations and a range of cooling flow levels. The test data set also includes cooling air efflux re-ingestion data. Further data for an 'optimised' cooling configuration geometry, will be obtained within the remaining period of the research programme.

CFD and FE models of the turbine rig configuration have been produced and run to replicate the test conditions. A fully automated CFD/FE coupling capability has been demonstrated and results from these models have been compared with the available test data, with encouraging levels of agreement, including the prediction of component temperatures within the turbine stator well. There is more work to do, including demonstrations of cooled cavity configurations coupled CFD/FE analysis, with all the relevant cavities modelled simultaneously. A further extension of the coupled CFD/FE modelling capability, to demonstrate a transient modelling capability, is confidently anticipated.

This modelling capability, adequately validated by the test data from the two-stage turbine rig, will provide gas turbine engineers with the necessary tools to optimise the cooling flows to this type of component and so reduce the environmental impact of gas turbine emissions and minimise the costs of ownership, as well as provide improved confidence in the analysis tools providing alternative means of compliance to the engine tests.

ACKNOWLEDGEMENTS

The present investigations were supported by the European Commission within the Framework 6 Programme, Research Project 'Main Annulus Gas Path Interactions (MAGPI)', AST5-CT-2006-030874. This financial support is gratefully acknowledged. Thanks also to our university and industrial partners at The University of Florence, The University of Sussex, The University of Surrey, Avio (Italy), MTU (Germany), Siemens (UK) and Turbomeca (France).

A special mention must be made to Rolls-Royce colleagues Christopher Barnes for his technical support and advice using SC89 and Leigh Lapworth for the recent development of the plug-in.

REFERENCES

[1] SPECIFIC TARGETED RESEARCH PROJECT Project Fact Sheet
Project acronym: MAGPI
Project full title: Main Annulus Gas Path Interactions
Proposal/Contract no.: 30874
Date of preparation of Annex I: May 2006

[2] Amirante D., Hills N., "A Coupled Approach for Aerothermal Mechanical Modelling for Turbomachinery", 1st International Conference on Computational Methods for Thermal Problems, 2009

[3] Coren, D. D., Atkins, N. R., Childs, P. R. N., Turner, J. R., Eastwood, D., Davies, S., Dixon, J., Scanlon, T. "An Advanced Multi Configuration Turbine Stator Well Cooling Test Facility", ASME Paper GT2010-23450

[4] Coren, D, 2011, "The Effect of Stator Well Coolant Delivery Passage Configurations on Cooling Effectiveness" ASME Paper GT2011-46448

[5] Da Soghe, Andreini, Facchini "Turbine stator well CFD studies: Effects of coolant supply geometry on cavity sealing performance", ASME Paper GT-2009-59186

[6] Dixon, J.A., Brunton I.L., Scanlon T.J., Wojciechowski G., Stefanis V., Childs P.R.N. "Turbine stator well heat transfer and cooling flow optimisation" ASME paper GT2006-90306.

[7] Dixon, J.A. Guijarro Valencia, A. Bauknecht, A, Coren, D.D. Atkins, N.R. "Heat Transfer in Turbine Hub Cavities Adjacent to the Main Gas Path" ASME Paper GT2010-22130

[8] Eastwood D., Coren D. D., Long C. A, Atkins N.R., Turner J. R., Childs P. R. N., Scanlon T. J., Dixon J. A., Guijarro Valencia, A. "Experimental Investigation Of Turbine Stator Well Rim Seal, Reingestion and Inter-Stage Seal Flows Using Gas Concentration" ASME Paper GT2011-45874

[9] Edmunds T., "Practical three dimensional adaptive analysis". In Proceedings of 4th International Conference on Quality Assurance and Standards, NAFEMS, 1993.

[10] Illingworth, J., Hills, N., Barnes C.J., "3D Fluid-Solid Heat Transfer Coupling of an Aero-engine Preswirl System", ASME Paper GT2005-68939

[11] Lapworth, L. The Hydra's User Guide for version 6.1.7 beta. Rolls-Royce plc, 2009.

[12] Launder, B.E. and Spalding, D.B., The numerical computation of turbulent flows. Comp. Meth. Appl. Mech. Eng., 1974, 3, 269–289.

[13] Spalart, P. R., and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA 92-0439, 1991

[14] Sun, Z, Chew J., Hills N., Volkov K., Barnes C.J., "Efficient FEA/CFD thermal coupling for engineering applications", ASME Paper 2008.GT2008-50638

[15] Verdicchio, J. A., "The validation and coupling of computational fluid dynamics and finite element codes for solving industrial problems". DPhil Thesis, University of Sussex, July 2001.