The Application of CFD to Underpin Material Selection for Gas Turbine Rotors

Dr Colin Young
Rolls-Royce plc
Fluid Systems Group
Derby

Abstract

The rotor assemblies of modern gas turbine engines operate in a particularly harsh environment, in which they are subjected to a combination of very large centrifugal forces and elevated temperatures. In order to endure these conditions, exotic rotor materials have been developed to sustain the imposed loading and complex air systems have been incorporated into engine designs to seal internal cavities and reduce metal temperatures.

In all engine designs a delicate economic balance must be struck between the incorporation of high temperature materials – which add significantly to unit costs - and the extent to which cooling air is drawn off from the various compressor stages. In the latter case air drawn off from the compressor stages makes no direct contribution to engine thrust and is therefore manifest as an increase in specific fuel consumption leading to higher operational costs.

In the present work computational fluid dynamics (CFD) techniques have been used to simulate the air system flow within a gas turbine rotor-stator cavity. Various rotor cooling arrangements are considered in order to use the available cooling air to best effect. The results obtained clearly demonstrate the potential for CFD to be applied to achieve the optimal economic balance between the conflicting requirements to minimise engine component and operation costs.

Keywords : CFD, gas turbines, rotor-stator cavities, turbulent flow

Introduction and Background

The rotor assemblies of modern gas turbine engines operate in a particularly harsh environment, in which they are subjected to a combination of very large centrifugal forces – arising from high rotational speeds - and elevated temperatures – in order to maximise the engine cycle thermal efficiency. In order to endure these conditions, novel manufacturing processes and exotic rotor materials have been developed to sustain the imposed loading and complex air systems have been incorporated into engine designs to seal internal cavities and cool critical components.

In all engine designs this leads to the need for a delicate economic balance to be struck between the incorporation of costly high temperature materials – which increase unit costs - and the extent to which cooling air is drawn off from the various compressor stages. In the latter case, air drawn off from the compressor stages makes no direct contribution to engine thrust and therefore detracts from engine performance. Exercising control over engine thermal conditions in this way results in undesirable increases in specific fuel consumption and hence to higher operational costs.

In principle these conflicting requirements for minimum unit and operating costs can be resolved by a break-even analysis of fixed and variable costs. In practice however the solution to this problem is greatly complicated by the need to devise the most effective air system configuration. Ideally such a system would be matched to stresses experienced by the rotor to deliver the minimum amount of cooling air to maximum effect.

Clearly our present technology falls somewhat short of this ideal and traditionally the trade off between the use of high temperature materials and the quality and quantity of air system flow has been struck on the basis of expensive engine tests. However, with the advent of flexible commercial CFD packages – such as Fluent – and cheap parallel computing platforms - in the form of PC clusters – the opportunity exists to strike this compromise in a far more informed manner than was possible hitherto. Moreover, this balance may be established at an earlier stage in the engine definition and verification process.

In the present work computational fluid dynamics (CFD) techniques have been used to simulate the air system flow within a gas turbine rotor-stator cavity. Various rotor cooling arrangements are considered in order use the available cooling air to best effect. The results obtained clearly demonstrate the potential for CFD to be applied to achieve the optimal economic balance between conflicting requirements to minimise engine component and operation costs.

CFD Model Set Up

The general arrangement of the front rotor-stator cavity of the intermediate pressure turbine (IPT) of a typical aero engine is shown in cross section in Figure 1 below. The geometric complexity of this cavity is immediately obvious from Figure 1. It can also be seen that there are several inlets to, and outlets from, this interior region of the engine as well as service pipelines which cross it radially. It is this geometric complexity and the flow field interactions arising from it that ultimately limits what can be achieved in modelling terms using network modelling techniques.

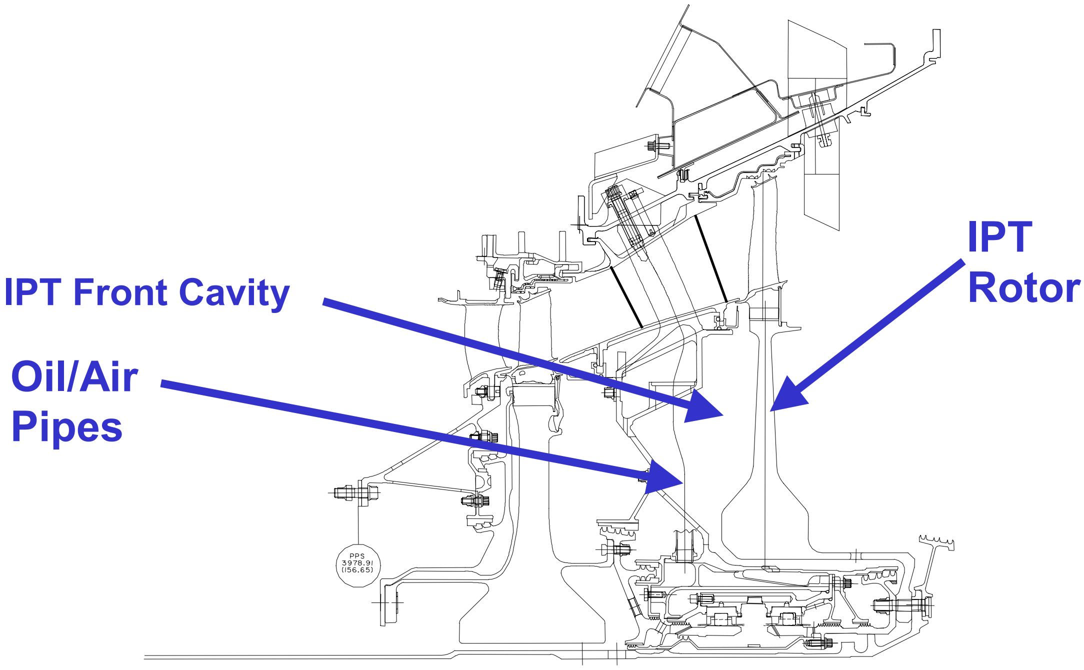
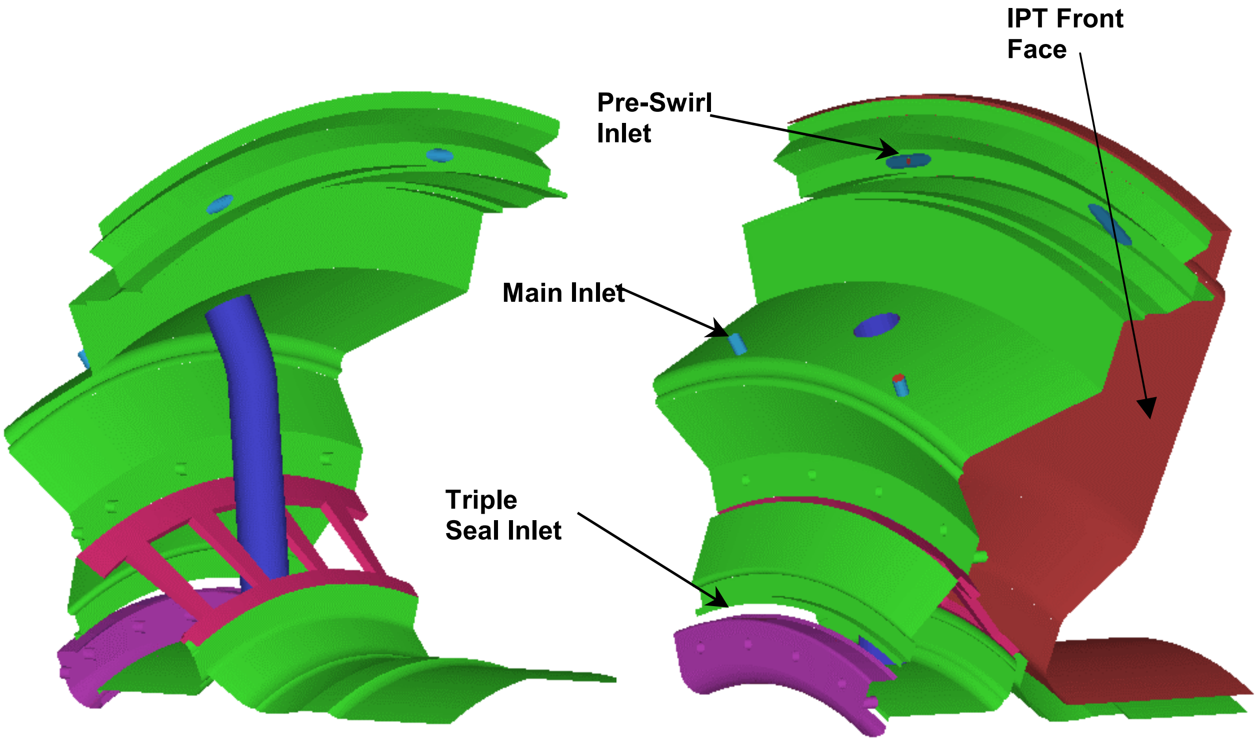


Figure 1 – General Arrangement of the IPT Front Rotor-Stator Cavity

For all practical purposes the geometry shown in Figure 1 may be regarded as one-eighth symmetric and so, by taking advantage of this, the computational requirements of this application can be significantly reduced. A three-dimensional CAD model of the flow domain under consideration was created from the basic 2-D

CAD cross section using the ICEMCFD software package DDN (Reference 1). Two views of the resulting eighth sector geometry are shown below in Figure 2.



view with IPT rotor omitted for clarity

Figure 2 – IP Turbine Front Cavity Model Geometry

It can be seen from Figure 2 that the main and pre-swirl inlet ports to the cavity are located near the mid and outer radii of the stator. Furthermore, it can be seen that the inlet ports at the mid radius are arranged normal to the stator surface while those towards the periphery are angled so as to impart a significant degree of swirl to the incoming fluid in order to maximise cooling of the rotor rim. An additional a ventilation source is situated close to the inner radius, where air emanates from a triple seal between the HP rotor assembly and the stator.

The ventilation air exits from the cavity by either passing along the inboard annular passage formed between the IPT rotor and the stator, through the rear seal between the HP rotor and the stator or flowing back into the main annulus via the outboard seal.

Using the geometry presented in Figure 2, the ICEMCFD Hexa mesh generation package (Reference 2) was used to generate a hexahedral calculational mesh for the model. This is shown in Figure 3 and consisted of approximately two million calculational cells.

Based on this calculational mesh, a model of the flow was set up using version 5.4 of the FLUENT CFD software package (Reference 3). Rotational periodic boundary conditions were imposed at the azimuthal faces – shown in blue in Figure 3 – and a combination of defined mass flow and pressure boundary conditions were imposed at the inlet and outlet zones, respectively.

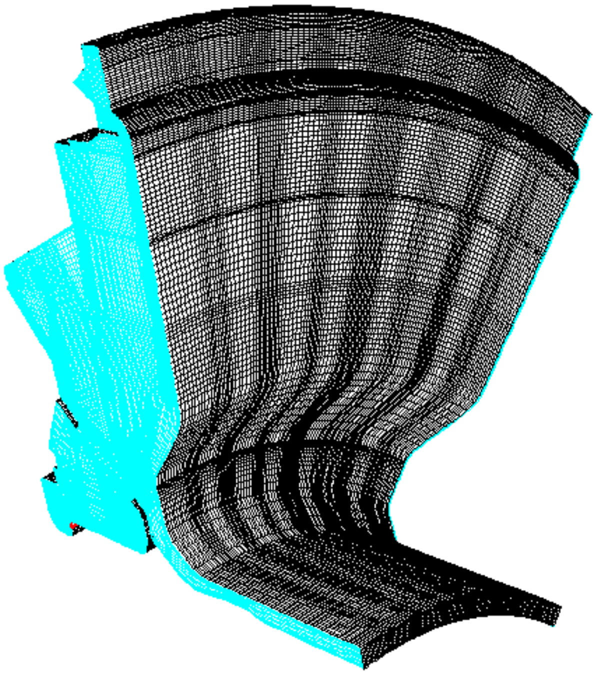


Figure 3 - IPT Front Cavity Model Calculational Mesh

The stator and rotor wall were assumed to be adiabatic for the purposes of this analysis and the fluid flow equations were solved in a rotating frame of reference.

The modelling options were set up to solve the mass, momentum, turbulence and energy equations, using the segregated implicit solver, with the second order discretisation scheme.

The effects of turbulence were represented using the standard k-ε model, in conjunction with a standard logarithmic law to represent near-wall behaviour. In addition, the effects of heating as a result of viscous dissipation were also included.

The physical properties of the fluid medium (air) were defined by selecting an ideal gas relationship for the density, whilst the dynamic viscosity, thermal conductivity and specific heat capacity were defined as piece-wise linear functions of static temperature.

Preliminary Flow Simulation

In an attempt to gain an insight into the likely accuracy of the IPT front cavity model it was initially run using a boundary condition set representative of a current Trent engine for which certification test data were available. In this case all of the cavity ventilation air was supplied via the inlet orifices at the mid-radius of the cavity. The model was run for approximately 25,000 iterations which were executed over a period of around 36 hours on a PC cluster configured with 22 - 1.7 GHz processors, at the University of Surrey.

Using engine test data the cavity air temperature, as measured by a thermocouple near the outer radius of the cavity, has been correlated with the bore temperature for a range of operating conditions. These data were used as the basis from which to assess the predictive capability of the model and are compared in Figure 4 below.

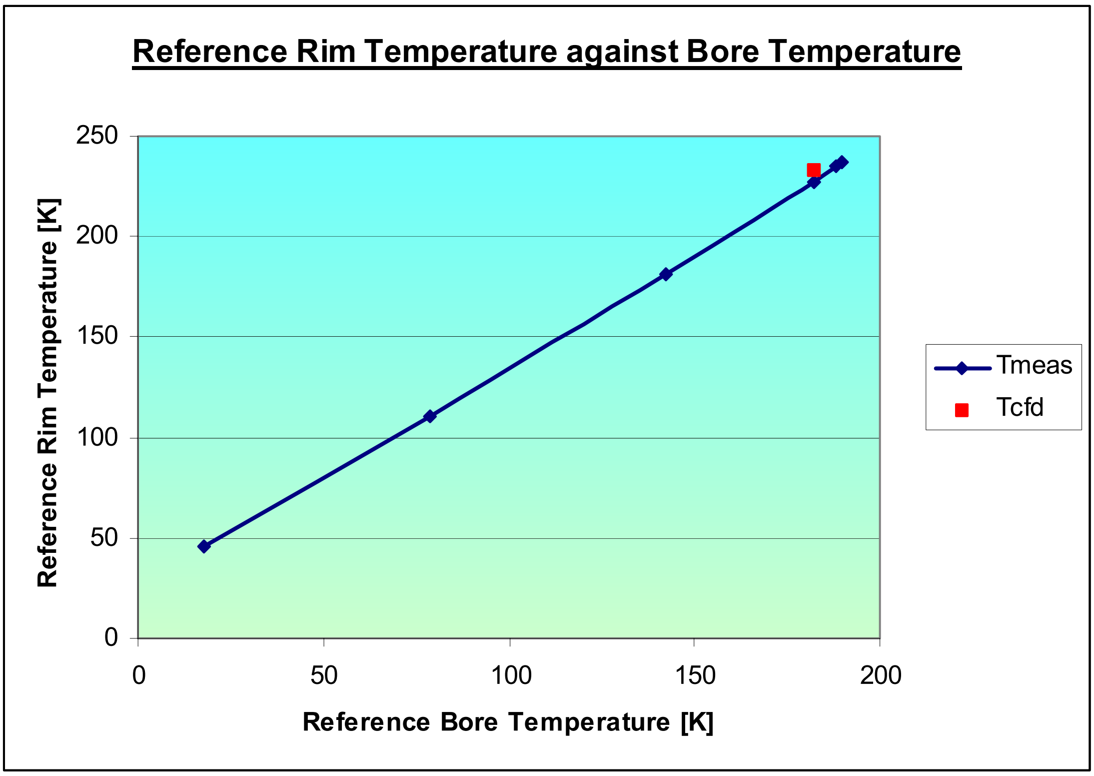
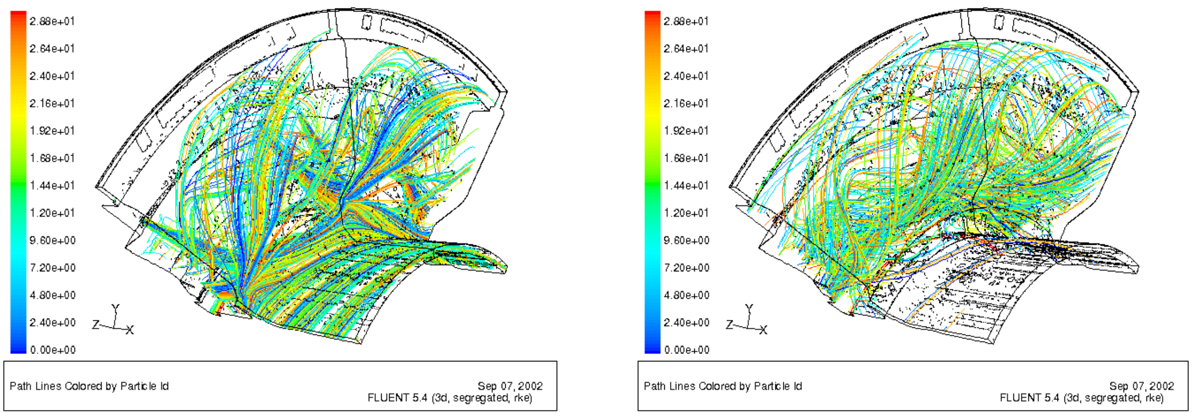


Figure 4 – Comparison of Measured data with CFD Model Prediction

As can be seen from Figure 4, the model predictions are very close to the operating characteristic of the test engine, with the CFD model slightly over predicting the temperature in the vicinity of the disc rim. It should also be noted that, as the simulated operating condition corresponded to the maximum take-off, the model predictions were expected to lie close to the upper limit of the engine test characteristic.

The CFD model predictions were also used to gain a detailed insight into the nature of the complex flow that takes place within this cavity. This was done by plotting the trajectories of particles released from the two ventilation sources (the inboard triple seal and the main ventilation nozzles located at the mid radius of the stator.
The pathline plots obtained from this solution are presented below in Figure 5.



particles released from vent nozzles particles released from the triple seal

Figure 5 – Flow Pathlines within the Cavity

It can be seen from these pathline plots that the flow leaving the main ventilation nozzles discharges as coherent jets (Ma ~ 1), that then impinge on the inboard structure and interact with each other promoting mixing. The smaller flow emanating from the inboard triple seal is prevented from flowing across the incoming jets and is either entrained into the main jets or flows around them, spiralling outwards between the coherent jets. These interactions give rise to strong mixing and subsequent scalar transport tracking indicates that almost complete mixing of the two air sources takes place.

Evidence of the impingement of the main jets on the inboard structure is evident from the reference temperature contours presented in Figure 6 below.

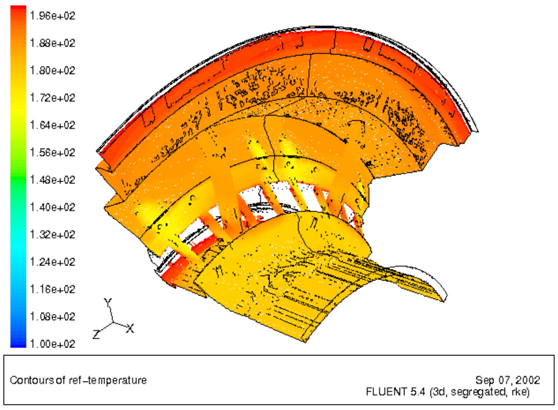


Figure 6 – Contours of Reference temperature on the Stator Structure

The corresponding reference temperature plot for the IPT rotor surface is also presented below in Figure 7 and this exhibits the characteristic high temperature region at the rim, which coincides with the most limiting region of the stress field.

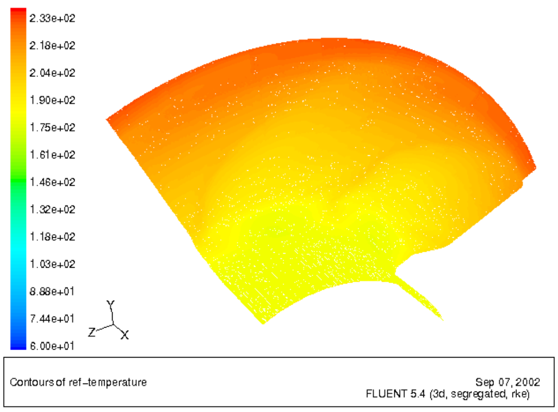


Figure 7 – Contours of Reference Temperature on the Rotor

Design Studies with Pre-Swirled Rim Cooling

It is evident from the preliminary analysis that significant advantage could be gained by introducing additional cooling airflow at the disc rim. However, as noted earlier, this would detract from the engine performance. In the present study, the total cavity ventilation flow rate was maintained at the level employed in the preliminary design work described above, however, additional (pre-swirl) nozzles were introduced in the vicinity of the disc rim.

With this revised arrangement the CFD model was run with the split of cooling air flow between the outboard and mid radius nozzles set in the ratios 100:0, 70:30 and 50:50. The aim of this study was to assess the viability of using CFD directly in the design process to support the disc material selection. The results of these analyses are presented below in Figures 8, 9 and 10, respectively.

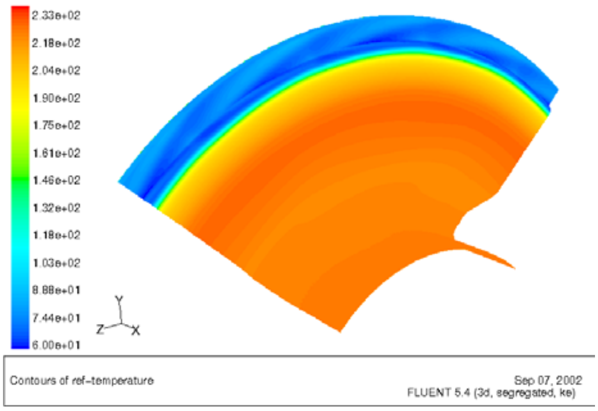


Figure 8 – Contours of Rotor Reference Temperature - 100:0 Supply Split

It can be seen from Figure 8 that the addition of the outboard pre-swirl arrangement, together with minor sealing modifications, has had a dramatic effect in reducing the disc rim temperature. This simulation demonstrates the ultimate extent to which the rim may be cooled preferentially at the rim. In fact, the preferential cooling of the disc rim is so successful in this case that one could envisage the thermal stresses resulting from the high temperature gradient inboard of the rim could be significant. However, a formal investigation of this is clearly outside the scope of the present work.

Comparison of the temperature contours presented in Figures 7 and 8 show the extreme temperature distributions that could be imposed on the disc. By modifying the pre-swirl to main nozzle flow split ratio to 70:30, the model predictions indicate the expected modest increase in disc rim temperature, whilst the temperature over the main part of the disc was reduced slightly. The is shown in Figure 9

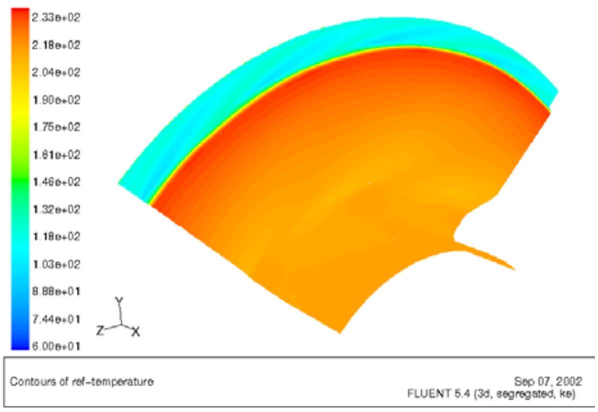


Figure 9 – Contours of Rotor Reference Temperature - 70:30 Supply Split

Finally the pre-swirl to main nozzle flow rate split ratio was reduced to 50:50. Once again, the model predictions indicate a modest increase in the disc rim temperature, whilst the temperature on the main body of the disc is seen to reduce, as shown in Figure 10 below.

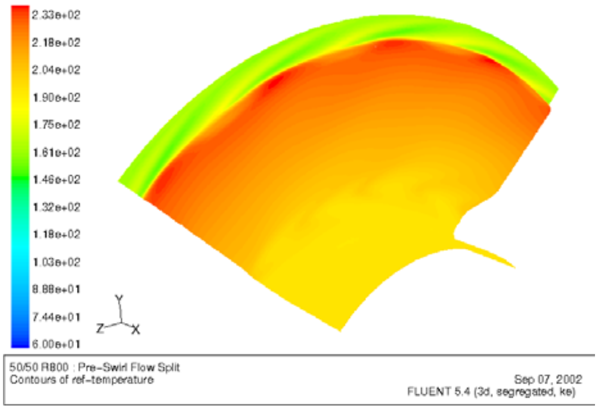


Figure 10 – Contours of Rotor Reference Temperature - 50:50 Supply Split

The foregoing analyses indicate that there is significant potential to augment the effectiveness of the cooling to the IPT disc of this development engine. Moreover, these results indicate that this improvement in disc rim cooling can be achieved without the need to draw additional air from the main annulus. In addition, the more effective cooling arrangement outlined in this work allowed the selection of a less exotic disc material on thermal grounds, allowing a significant reduction in unit cost to be exploited.

Work is continuing within Rolls-Royce to further optimise other aspects of the secondary air system of this development engine using CFD and it is intended that this work will also include optimisation of the main annulus seal in this cavity.

Conclusions

The principal aim of the present work was to assess whether, by improving the effectiveness of the secondary air system a less exotic material could be adopted for the IPT rotor disc. This primary aim was successfully met in this application without the need to increase the amount of air drawn off from the main annulus.

This in itself represents a significant advance, in terms of the particular engine development programme to which it contributed. However, it is evident from this work that there is enormous potential for CFD modelling techniques to be applied to improve the secondary air systems of gas turbines in general.

Finally, it is also evident from this work that combination of reliable and robust flow solvers (e.g. Fluent) and relatively cheap high performance computing platforms (PC clusters) that CFD has effectively “come of age”. As such, these techniques are starting to be routinely applied to flows in highly complex situations, about which little may be known, and used as one of the main drivers in the overall design process.

References

1. ICEM CFD Tutorial Manual : Direct CAD Interfaces, Version 4.0, ICEM CFD, March 2000.

2. ICEM CFD Tutorial Manual : Meshing Modules, Version 4.0, ICEM CFD, February 1999.

3. Fluent 5 User's Guide, Volumes 1-4, Fluent Incorporated, 1998.

Acknowledgements

The co-operation of Rolls-Royce plc in allowing publication of this work and the assistance provided by Prof Chew and Dr Hills at the Universities of Surrey and Sussex is gratefully acknowledged.