SUMMARY REPORT

CONTRACT N° : **BRPR-CT97-0553**

PROJECT N° : **BE97-4021**

TITLE : **Fluid Flow and Heat Transfer within the Rotating Internal Cooling Air Systems of Gas Turbines**

ICAS-GT

PROJECT CO-ORDINATOR :Rolls-Royce plc GB

PARTNERS : **INDUSTRIAL**
RR, SNECMA, MTU, RR-D, TM, VAC, FA, AP(CH), AP(UK), ITP

UNIVERSITY
Sussex, Bath, RWTH Aachen, TU Karlsruhe

STARTING DATE : **01.01.1998** **DURATION : 36 months**

Date of issue of this report : March 2001

Project funded by the European Community either under the 'IMT/SMT' Programmes (1994-1998)

ICAS-GT Project Executive Summary

This project has addressed the problems of heat transfer and fluid flow within gas turbine internal cooling air systems, with the aim of improving air system designs for both advanced aircraft and electrical power generation gas turbines. The project consortium has brought together ten major European engine manufacturers and four Universities specialising in this type of work.

Rotating fluid flow and heat transfer has attracted considerable research interest over many years, but the majority of experiments have been run at non-dimensional conditions which fall well short of actual gas turbine engines. The approach adopted in this project has included:

- advanced experimental work to obtain improved data under representative operating conditions,
- an evaluation of flow and heat transfer models and CFD codes,
- development of improved 3D models and correlations for design purposes and,
- establishing a more standardised European approach to these problems.

Five separate, but related, topics have been studied, i.e.:

- Turbine annulus hot gas ingestion control.
- Air flow and heat transfer in rotating cavities with axial and axial/radial throughflow.
- Heating which occurs in compressor stator wells.
- Turbine cooling air pre-swirl systems.
- Air flow and heat transfer in a high pressure compressor drive cone cavity.

These aspects cover the most critical problems in the field of internal air systems design. In each case, experiments run by the University partners have yielded an extensive database of steady and unsteady pressure, temperature, heat flux and velocity measurements which have been used by the industrial partners to validate CFD modelling methods. The experiments and CFD calculations together have improved physical understanding and resulted in the formulation of design correlations. For example, turbine rim sealing effectiveness has been correlated against sealing flow for a range of rim seal geometries, and the heat transfer on the internal surface of compressor drive cones has been correlated against Reynolds number and Grasshof number. Such correlations are now being evaluated by the industrial partners for incorporation in their internal air system design software suites, and will have an immediate impact in improving detailed predictive engine design, and in reducing the overall engine production cycle time.

Exploitation of the project deliverables has been promoted further by the development within industry of CFD modelling methods and CFD practitioners. The project has coincided with the relatively widespread adoption of clustered Personal Computers as powerful and cost effective computing hardware. These machines have brought to bear sufficient processor speed and memory to make 3D unsteady CFD a viable approach for internal air system calculations, and this has been used to excellent effect. For example, by applying 3D unsteady CFD to a single rotating cavity, complex 3D flow structures have been calculated which change with rotational speed, (see figure 1). The corresponding heat flux and velocity predictions are in very good agreement with experiment. Unsteady CFD has also enabled the magnitude and influence of unsteady flow in pre-swirl systems to be quantified. The viability of simultaneously modelling flows in the main annulus and in the internal cavities of a single stage turbine has been demonstrated, (see figure 2). This approach has been valuable in understanding the unsteady fluid mixing which occurs at turbine disc rim seals. The availability of powerful computers has also promoted the development of conjugate fluid flow and heat transfer calculation methods, and the success of this approach has been demonstrated for compressor stator well and compressor drive cone geometries.

As exploitation continues, so the project will result in benefits including reduced fuel consumption (up to 1% as a result of more efficient and optimised air system designs) and longer component life due to lower cooling air temperatures within the engine. The project will enable European gas turbine engine manufacturers to remain fully competitive in world-wide markets as well as improving their growing trans-national co-operation.

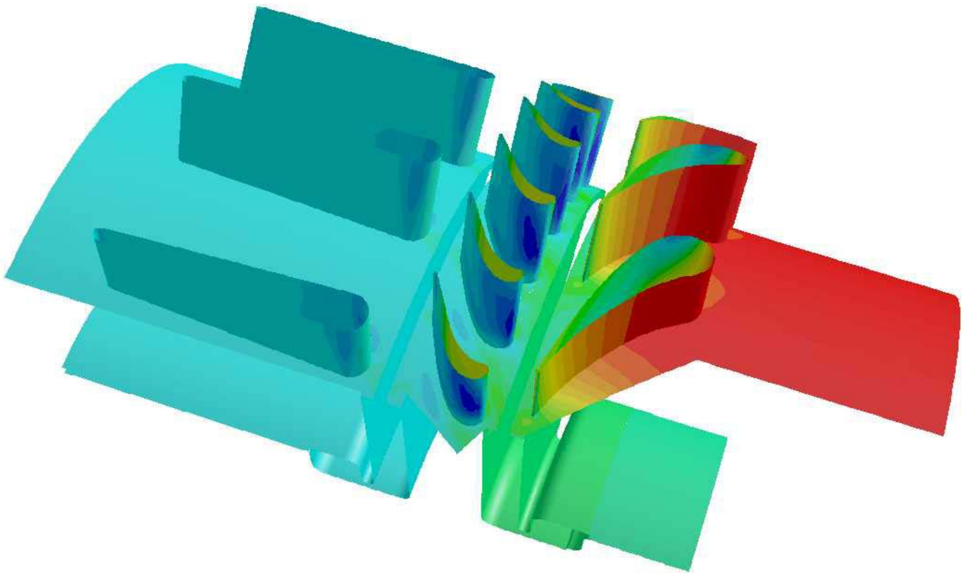

Figure 1: 3-D Grid for Unsteady CFD Calculations of Rim Seal Test Facility Flows, (shaded by static pressure where red is high, blue is low)

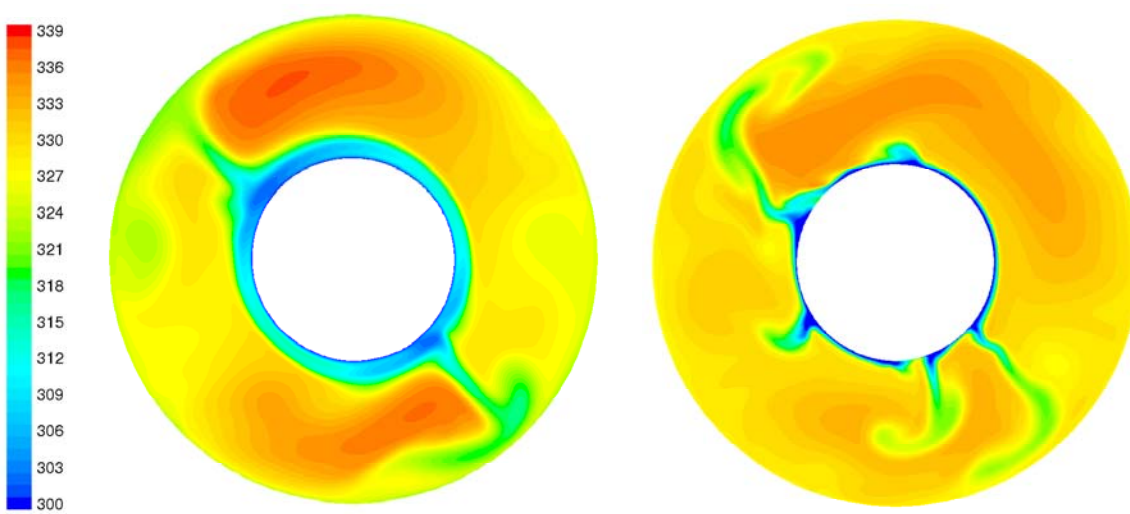

Figure 2: Rotating Cavity Flow Simulations for Low (Buoyancy Dominated) and High (Inertially Dominated) Rotation Rates. The figure shows contours of static temperature on a radial-tangential plane in the middle of the cavity