

CFD for Turbomachinery Internal Air Systems

John W. Chew and Nicholas J. Hills
Fluids Research Centre
School of Engineering
University of Surrey
Guildford
Surrey, GU2 7XH, UK

Abstract

Considerable progress in development and application of CFD for aeroengine internal flow systems has been made in recent years. CFD is regularly used in industry for assessment of air systems, and the performance of CFD for basic axisymmetric rotor/rotor and stator/rotor disc cavities with radial throughflow is largely understood and documented. Incorporation of three dimensional (3D) geometrical features and calculation of unsteady flows are becoming commonplace. Automation of CFD, coupling with thermal models of the solid components, and extension of CFD models to include both air system and main gas path flows are current areas of development. CFD is also being used as a research tool to investigate a number of flow phenomena which are not yet fully understood. These include buoyancy-affected flows in rotating cavities, rim seal flows and mixed air/oil flows. Large eddy simulation has shown considerable promise for the buoyancy-driven flows, and its use for air system flows is expected to expand in the future.

Key words: CFD, Aeroengines, Turbomachinery, Heat Transfer

1. INTRODUCTION

As in other areas of aerospace engineering, computational fluid dynamics (CFD) is now an established tool for use in aeroengine internal air system design. The internal air system plays a vital role in modern gas turbines, serving to dissipate windage and heat conducted to the discs, deliver air for turbine blade cooling, maintain the required axial pressure loading on the bearings, prevent hot gas ingestion from the main annulus overheating the turbine discs, and isolate the oil system. A section of the internal air system showing the high and intermediate pressure sections for a three shaft engine is shown in figure 1. The later stages of the high pressure (HP) compressor, the combustor, the HP turbine and the intermediate pressure (IP) turbine are shown in white. The colour coding for the internal air flows relates to the compressor stage from which the cooling air originates. For example, the red areas are fed from the HP compressor delivery. Green areas denote the presence of oil in the bearing chambers. With up to 25% of compressor air being used for cooling and sealing, the air system has a significant impact on engine efficiency. Considerable attention must be given to the internal air system and its interaction with the main annulus flow.

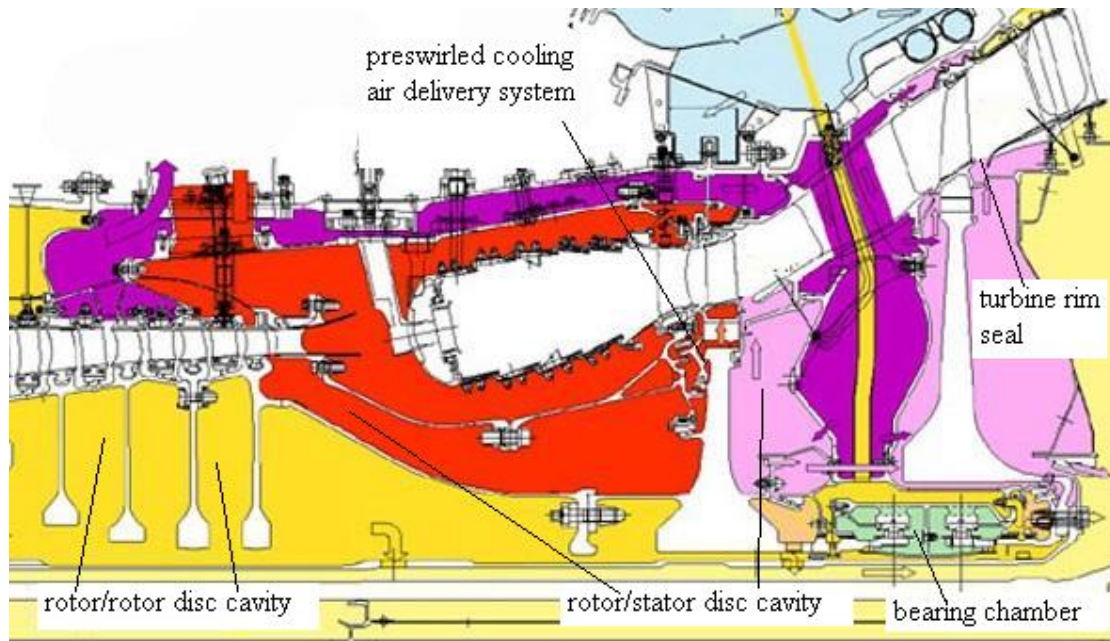


Figure 1. Section of the internal air system, showing the HP and IP sections for a three shaft engine (courtesy Rolls-Royce plc).

A wide variety of flow phenomena and types occur in internal air systems. Rotating disc cavity flows are particularly important and have been the subject of much research over the years. These flows are often dominated by rotational effects that lead to strong coupling between the components of the momentum conservation equations. In CFD studies this often results in slow convergence (or failure to converge), and care is needed in application of standard CFD procedures. Typically, comparing CFD models with the same number of mesh points, a rotating disc cavity problem requires an order of magnitude more computing time to converge than a 'standard' turbomachinery blading flow application. Thus exploitation of CFD for internal air systems naturally lagged behind that for main gas path flows and there has sometimes been a mistaken perception that the CFD methods used for internal flows were not as advanced as in other areas. Today, as interaction of disc cavity and mainstream flows receives more attention, there is common interest in establishing efficient and robust solution procedures able to calculate both air system and mainstream flows.

Sustained research in internal air systems has been conducted since at least the 1970's. Experimental, computational and theoretical approaches have been complementary, with the potential benefits of CFD recognised at an early stage. Although early calculations showed considerable promise (see, for example, Gosman et al., 1976) production of validated methods that can be applied with confidence was not straightforward. Numerical difficulties, turbulence model limitations, lack of suitable experimental measurements, and limited understanding of the flow mechanisms involved have all hampered progress. Nevertheless, CFD is now used with some confidence in industry, and is considered essential as a research tool. Continuing expansion in CFD capability and use is expected for the foreseeable future.

In this paper, some of the developments in CFD for air systems are described and illustrated. Most emphasis is placed on recent progress, but some outstanding issues and areas where greater activity is expected in the future are also highlighted.

CFD software and modelling developments are discussed in the next section. Internal air system applications are then illustrated in section 3. Interaction of air system flows with the main gas path, which is of interest to both air systems designers and turbine aerodynamicists, is discussed in section 4, with principal conclusions being stated in section 5. Whereas, earlier work and reviews in this area (see, for example, Owen and Rogers 1989, 1995 and Chew, 1990) have stressed progress in modelling of steady, axisymmetric flows, here emphasis is placed on complex, unsteady 3D flows that are now succumbing to CFD analysis.

2. CFD METHODS DEVELOPMENTS

2.1 Solution Algorithms

For industrial air system CFD applications, most early work used pressure correction solution methods since, due to the lower Mach numbers in the internal air system, the flow was generally treated as incompressible. For example, the work described by Chew (1984), used structured, Cartesian grids and was limited to simple geometries. By the mid nineties, as illustrated by Virr et al. (1994), the use of boundary fitted meshes was commonplace and steady, axisymmetric, incompressible CFD methods were established in industry. As computing power has advanced, these restrictions have been relaxed.

Fully 3D models are now common, and there is a trend to use 3D unsteady models where necessary, rather than approximate the problem. Examples of this are given throughout this paper.

Internal air system flow is no longer commonly treated as incompressible. Solution algorithms consist of either pressure correction methods extended to include compressibility effects or density based methods extended where necessary to low Mach number regimes. A driving force behind the increased use of density based methods has been the need for efficient and robust solution procedures able to calculate both air system and mainstream flows. Pressure correction methods typically do not strictly respect the strong conservation form of the governing equations and hence have not been widely used by mainstream flow CFD practitioners. To work efficiently for low Mach number flows, density based methods require preconditioning. Preconditioner performance has not always been robust, and so pressure correction methods are often preferred for internal air systems. Recent work on the efficient extension of density based methods to the solution of low Mach number flows is described by Rossow (2006).

The introduction of multigrid methods has proved extremely effective for density based solvers. For inviscid flows, Jameson and Caughey (2001) demonstrated that airfoil flow solutions could be obtained in 3-5 multigrid cycles (with convergence to the same level as the truncation error). For viscous problems, the stretched meshes typically used to resolve the boundary layers can cause much slower convergence. Various approaches have been tried to combat this, but J-line coarsening (due to Mulder, 1989), where the mesh is coarsened only in the direction normal to the solid boundary, has proved to be effective at mitigating this at the cost of programming complexity when generating the coarse grids. (See, for example, Pierce and Giles, 1997).

Commercial mesh generators have typically replaced in-house mesh generation software in industry. Links with CAD geometry definition are now widely available (although not as commonly used) and the geometrical complexity of the problems tackled has greatly increased. Due to the complexity of internal air system geometries, which are commonly determined by mechanical rather than aerodynamic considerations, unstructured meshes are in increasing use. As yet, mesh adaption has

not been widely used. Possibly this is due to the much greater difficulty in defining the parameters to adapt upon for internal air system flows compared to airfoil flows, where the gradient of pressure is commonly used for mesh adaption to enable better shock capture.

2.2 Parallelisation

One of components of advancing computer power has been the move towards PC clusters built with off-the-shelf hardware and fast inter-connects. It is clear from the current trend towards multi-core chips that future advances in computer power will come from increasing the cluster sizes (in terms of the number of cores) rather than increasing the individual core speeds. Consequently, the ability to scale problems and codes efficiently to large number of processors is assuming ever greater importance.

In the last few years, there appears to have been a convergence of approach to parallel CFD. Almost all parallel CFD codes now seem to be based on MPI (message passing interface standard) approaches and Karypsis and Kumar's MeTis (or the parallel version, ParMeTis) packages are apparently becoming a de facto standard for unstructured mesh partitioning. A description of the work necessary to obtain good parallel performance for an unstructured CFD code is given by Hills (2006). An example of the scaling performance that can be obtained for a combined mainstream and internal air system problem of the type discussed in section 4.2 is shown in figure 2. The problem has approximately 21 million mesh nodes and super-linear scaling can be seen in the figure for up to 1024 IBM POWER5 processors. This corresponds to around 20,000 mesh nodes per processor which seems to be the current state-of-the-art figure for scaling in CFD.

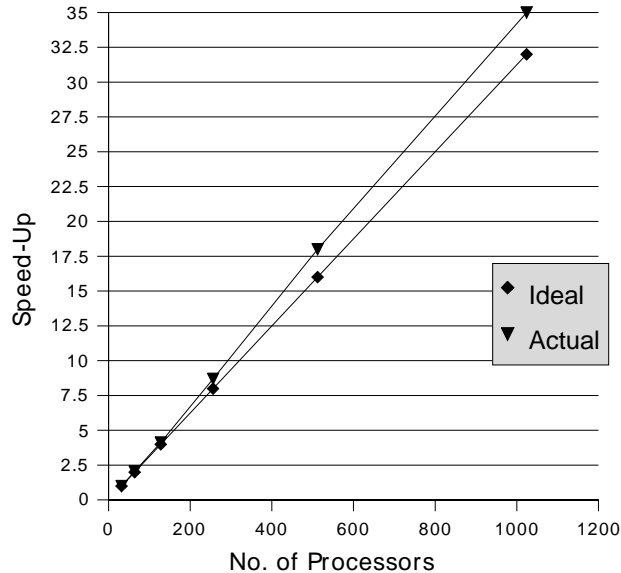


Figure 2. Scaling Performance for a Combined Mainstream and Internal Air System Problem

As the parallel efficiency of CFD solvers has improved, and the cost of large parallel clusters has dropped, CFD simulations now take considerably less elapsed time than was the case a few years ago. The CAD geometry definition, mesh generation, and post-processing stages of the process are now significantly slower than the problem solution time. Increasingly, the aim is to have an entirely parallel

system, from geometry manipulation to post-processing, and thereby avoid any bottlenecks created by parts of the process running serially. A good discussion of the issues, and a novel approach to the problem, is given by Dawes (2006).

2.3 Turbulence Modelling

While, as will be discussed later, large eddy simulation (LES) is gaining acceptance in the research community and attracting interest in industry, most or all current industrial air systems applications of CFD use the Reynolds-averaged Navier-Stokes equations (RANS) with a model of turbulence. While it is generally accepted that there is no universally valid turbulence model, interest and experimentation in the choice of models continues. In some cases this choice has a significant influence on results. For high Reynolds number flow on rotating discs the conventional k - ϵ model with standard wall function treatment has been shown to be satisfactory (Virr et al., 1994) and is used by many workers. However care is required at lower Reynolds numbers where models resolving the near-wall region such as the two layer k - ϵ/k - l model (see, for example, Iacovides and Chew, 1993) should be preferred. Other eddy viscosity models such as mixing length and the one equation Spalart-Allmaras model may also give satisfactory results, although evaluation against benchmark test cases is recommended, and can be revealing.

Some workers recommend Reynolds stress models (RSM) for rotating system flows, and improvements over k - ϵ models have been demonstrated for certain cases. For example, Lee et al. (2004) showed better agreement with velocity measurements in a bearing chamber model using an RSM model than for the k - ϵ model. RSM models may thus become more popular as their additional computing requirements become less significant. It is probably inevitable that predictions for complex flows (such as those involving separation) will be sensitive to the choice of turbulence model, but careful work is required to establish whether or not improvements in comparison to data are due to sound modelling of the physics. For the complex 3D geometries, with features such as bolt heads, nozzles and orifices, now being modelled for the internal flow system, the limitations imposed by turbulence modelling are similar to those in many other application fields. As highlighted in a workshop at the Isaac Newton Institute for Mathematical Sciences (1999), it is clear that the search for more general (or universal) turbulence models is proving extremely difficult. In a paper presented at that meeting, Lumley (1999) described the results of a collaborative effort in testing turbulence models as follows:

“... In any event, the conclusion was that none of the models can reliably predict a variety of flows with the accuracy that industry desires, but can only do so in highly restricted situations where the model has been extensively calibrated. In general, the Reynolds stress transports, or second order models, do better than the simpler models only in the sense that they are able to compute some flows that are quite beyond the simpler models, because the physical mechanism in question has not been included in the model. Where both type of model work, the more complex model is not often significantly better than the simpler one, possibly because more terms have been modeled, and the terms are less familiar, and possibly because both depend on the same inadequate dissipation equation.”

LES and its variants, such as detached eddy simulation (DES), are now well established in the research community for investigation of complex turbulent flows, and there are a growing number of examples where LES is shown to perform better than unsteady RANS models. As will be illustrated below in section 3.2, this includes air system flows. While a few years ago there was some controversy about numerical accuracy required for LES, the view that conventional engineering CFD codes can be relatively easily modified for LES has gained ground. This leaves open questions

about such issues as the influence of numerical dissipation and the exchange of turbulence energy between regions with different mesh scales or modelling assumptions. However, in a recent paper Pope (2004) notes the dependency of many LES models on the numerical methods employed, and suggests that an optimal LES method is likely to contain non-negligible numerical diffusion. It seems that, as has been the case for RANS models, LES will be accepted as an imperfect but very useful model.

2.4 Automatic Analysis and Design Optimisation

The availability of robust and flexible commercial CFD software packages has considerably helped in encouraging application of CFD in industry. However, exploitation is still limited by the relatively high analysis preparation time. A further concern is that results obtained may be user-dependent, presenting a quality control problem. Further automation of the CFD process is needed to address these issues. The advantages of automatic analysis have been demonstrated some time ago for solid mechanics. For example, the techniques introduced by Armstrong and Edmunds (1989) have had very significant impact in industry with, in many cases, automatic meshing being used to achieve a user-specified accuracy. This substantially reduces work load on users and similar capability for CFD will be needed in the future.

In some application areas, such as turbomachinery blading, considerable automation has already been achieved within industry. For air systems, the more complex and variable geometries, and the limited scope for aerodynamic design have restricted automation, but activities in this area are now growing. Figure 3 shows a parametric model of a nozzle for a pre-swirled cooling air delivery system. As will be explained more fully in section 3.3, the purpose of such nozzles is to impart a swirl velocity on the cooling air. Ciampoli et al. (2006) have demonstrated that the mesh generation and analysis for such a model can be automated and incorporated within a design optimisation algorithm. Such methods are now being explored in industry for other air system applications. It may be noted, however, that the geometry parameterisation is being done outside CAD. Potentially, integration with CAD offers significant benefits in efficiency of the CFD process and for multi-disciplinary design, and this is now receiving attention in industry.

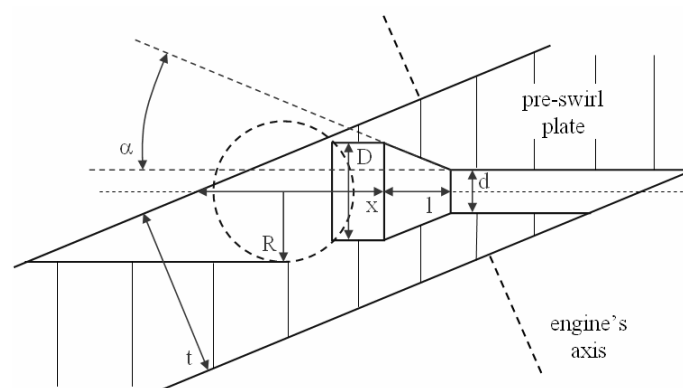


Figure 3. Parametric representation of a pre-swirl nozzle for design optimisation.

The example given above, and many other current optimisation applications, use CFD mesh generation rules derived for the specific application in mind, and care is needed to ensure mesh sensitivity does not distort the results. Facilities for mesh adaption are available in a number of widely used CFD codes but, as noted in section

2.1, take up of this technology has apparently been limited. Further work is needed in the area of mesh generation, and this is now an active area of investigation for air system flows. Recently, Volkov (2006) has demonstrated a prototype facility for the use of CFD codes within a finite element thermal modelling environment. In this approach the CFD mesh and model input data generation is embedded within the FE package. Stand-alone CFD or fully coupled FEA-CFD analyses (as described in the next section) are then launched automatically. When mature, such modelling facilities will offer considerable scope for reducing analysis time and use within multi-physics design optimisation methods.

2.5 Thermal Modelling

The use of finite element codes to model thermal and structural loads within turbomachinery is well established. These models typically use boundary conditions based on a variety of correlations and physical assumptions. The appropriate specification of these boundary conditions relies heavily on the experience and expertise of the user and requires validation (or 'matching') against engine data for use in calculations of part life. Monico and Chew (1992), for example, discuss the industrial practice for modelling engine casings and discs. In order to match the engine data, both models and correlations have become increasingly complicated as they struggle to replicate the intricacies of modern engine designs. A variety of techniques have therefore been developed for replacing these correlations with boundary conditions derived from CFD models.

The simplest of these methods aims to derive a heat transfer coefficient distribution from the CFD model and apply this to the finite element model either directly or by fitting an existing correlation (multiplied by some correction factor) to this distribution. While the use of independent CFD solutions may be helpful in guiding the development of simple boundary conditions, it is difficult to apply the method consistently over a range of operating conditions, and the method fails to replicate effects of the metal temperature distribution on the flow solution and heat transfer.

Fully coupled methods provide a two way coupling between the CFD solution and the FE solution. Boundary conditions are passed between the two models to ensure continuity of temperature and heat flux at the fluid-solid boundary. Early applications of this type of coupling have been for turbine blade cooling by, for example, Heselhaus et al. (1994), Li and Kassab (1994), Bohn et al. (1993a,b,c and 1995a), and Chew et al. (1996). Application to internal air system cavities was demonstrated by Mirzamoghadam and Xiao (2000) and by Verdicchio et al. (2001). A typical state of the art calculation is described by Illingworth et al. (2005) who applied the method to an engine pre-swirl system and modelled a full transient engine cycle.

An alternative approach to coupling is a 'conjugate' method whereby one code is used to carry out the entire calculation rather than coupling two separate calculations together. The modifications required to add a conjugate capability to an existing Navier-Stokes code are described by Rigby and Lepicovsky (2001). Further publications in recent years indicate the increasing use of the conjugate method to solve industrial problems (see, for example, Ho et al., 1996, and a number of papers by Bohn and co-workers at Aachen). However, while conjugate methods are preferred by some workers because of the simplicity of using one code rather than two, the disadvantage is that currently available conjugate solvers generally lack the specialist thermal modelling features developed for turbomachinery applications. Furthermore, the predicted component temperature distribution may have to be exported to a solid mechanics code for use in stress analysis. Hence, while the conjugate method may be

valuable for a specific application, it is unlikely to be used for the typical models run in industry.

3. INTERNAL AIR SYSTEM APPLICATIONS

3.1 Rotor/stator disc cavities

In 1960, Daily and Nece published flow and torque measurements for a rotating plane disc enclosed by a simple stationary shroud. This has proved a useful test case for CFD, and it was established some years ago that for the higher Reynolds numbers and cavity widths most relevant to aero-engines, conventional axisymmetric CFD models could give acceptable results when applied with care (see, for example, Chew and Vaughan, 1988). This and similar test cases have been repeated by a number of workers, and engine rotor/stator disc cavities are now regularly analysed using CFD in industry. However, a number of questions regarding accuracy remain. These include modelling of 3D features such as bolt heads, and unsteady flow features such as Taylor-type vortices.

Bolt heads or other 3D features may have an important influence on the flow in disc cavities, and may dominate over disc drag in determining the rotor moment and windage heat generation. Such features have sometimes been included in axisymmetric CFD models through addition of body force or source terms in the momentum equations. Today, full 3D CFD models are increasingly used. There is little published work on the validity of either approach, and this is an area of current research, as reported, for example by Smout et al. (2002).

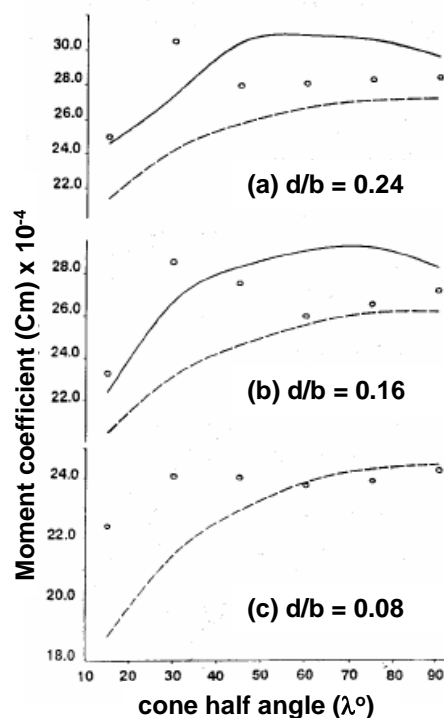


Figure 4. A comparison of predicted moment coefficients for an enclosed rotating cone with Yamada and Ito's measurements for different cavity width to radius ratios (d/b): o - measurement, solid line - CFD, broken line – integral method. (May et al., 1994)

Taylor-type vortices were reported in high Reynolds number turbulent conical rotor/stator systems with the inner cylinder rotating by Yamada and Ito (1975, 1979) for half cone angles of 30° and less. Figure 4 shows a comparison of CFD (and an integral boundary layer model) results for the rotor moment coefficient with Yamada and Ito's measurements. The CFD model is axisymmetric and uses a mixing length model of turbulence. Although the level of moment given by the CFD compares fairly well with the experiments, the CFD does not reproduce the experimental trends, particularly the peak in moment for a half angle of about 30° . This peak could well be due to the unsteady Taylor-type vortices observed experimentally. It is likely that such effects occur in engine internal flows, and this will be investigated further as use of unsteady CFD models (such as LES) increases.

3.2 Rotor/rotor disc cavities

Axisymmetric flows in co-rotating disc cavities with a radial inflow or outflow of fluid were investigated in considerable detail in the 1970's and 1980's (see, for example, Owen and Rogers, 1995). As discussed above for rotor/stator cavities, such flows are regularly calculated using CFD in industry and models are increasingly extended to include 3D features such as holes or protuberances. However, particular difficulties have been encountered in co-rotating disc cavities with little or no imposed radial throughflow. Such cavities occur in HP compressors in which the disc rims are generally hotter than the disc cobs which are cooled by a central axial throughflow. Sealed annular cavities have also been used in some engines. The positive radial temperature gradient results in strong buoyancy effects in the centripetal force field. The resulting flows have been known to be strongly 3D and unsteady for some time and have been recognised as particularly challenging for CFD.

Bohn et al.'s (1995b) experimental configuration of a sealed rotating annulus heated at the outer shroud and cooled at the inner shroud has been studied with CFD by several authors. Bohn et al. performed unsteady laminar flow computations and found irregular fluctuations in the flow. Limitations in computer power prevented a fully conclusive study. More recently King et al. (2005) applied a two dimensional, unsteady, laminar CFD model to Rayleigh-Benard convection in a sealed cavity. The calculated heat transfer was higher than Bohn et al.'s experimental correlation.

Sun et al. (2004) studied high Rayleigh number free convection under gravity in a stationary cube and under centripetal force in a rotating cavity. Somewhat surprisingly, laminar, unsteady 3D CFD models were found to give excellent agreement with accepted empirical correlations for the stationary cube and with Bohn et al.'s sealed rotating cavity results. The comparison with Bohn et al.'s correlation is shown in figure 5. This figure also includes an adaptation of Kirkpatrick and Bohn's correlation for free convection under gravity to centrifugally-driven convection. This indicates that the Coriolis force in the rotating annulus suppresses the heat transfer. Large scale flow structures were found in the CFD solutions at all conditions considered. Although the solutions showed turbulent characteristics, the smallest (Kolmogorov) turbulent length scales were not fully resolved, indicating that these calculations could be classed as large eddy simulations with the numerical viscosity contributing to turbulence energy dissipation.

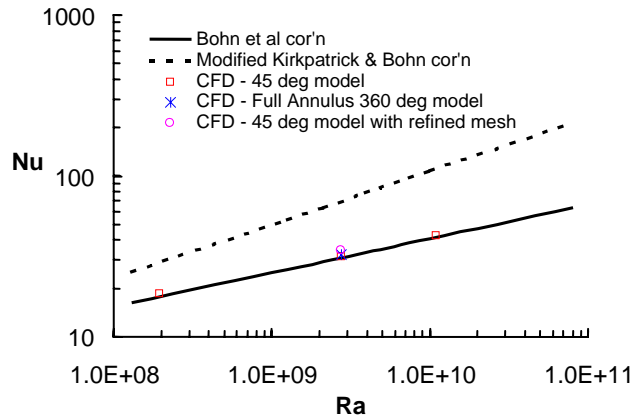


Figure 5. Comparison of heat transfer calculations for the rotating annulus with experimental correlations (Sun et al., 2004)

Other workers have applied unsteady CFD methods to a rotating cavity with a central axial throughflow. For example, Long and Tucker (1994) obtained laminar unsteady CFD solutions for a heated cavity with axial throughflow. However, limitations of the laminar model were recognized. Smout et al. (2002), Wong (2002) and Owen et al. (2006) have reported use of conventional $k-\epsilon$ turbulence models in unsteady 3D simulations, but the information available in the open literature is limited. Both Wong's and Owen et al.'s comparisons with heat transfer measurements gave mixed results. In a recent publication Sun et al. (2006) presented LES results for a model HP compressor disc cavity with axial throughflow and compared with heat transfer and velocity measurements from the rig described by Long et al. (2003). It was concluded that the LES results were clearly in better agreement with the measured data than those obtained using a $k-\epsilon$ model.

The cavity geometry and some results from Sun et al.'s study are shown in figures 6 to 8. The choice of abscissa in figure 7, which compares measurements and LES results, follows that of Long et al. (2003). Note that the "other rig data" referred to in this figure comes from earlier, somewhat different, builds of the same experimental rig. LES shroud heat transfer predictions were within 25% of measurements for the corresponding experiments, tests 33, 34 and 50. Good agreement with tangential velocity measurements was also demonstrated. Figure 8 shows an instantaneous, mid-axial contour plot of static temperature. The colder central throughflow can clearly be seen. A cold plume emanates from the throughflow jet, and hot plumes are formed from the heated shroud. Comparison of the unsteady flow from these LES solutions with unsteady RANS solutions confirmed that, as would be expected, the LES gives a more convincing representation of the turbulence. However, the LES calculations are computationally demanding, to the extent that Sun et al.'s investigation was limited to just 3 experimental conditions. Further work has been recommended, including investigation of more experimental conditions, and careful consideration of the near-wall LES modelling and interaction of the flow in the central jet with the main cavity flow. As recent calculations on the UK national HPCx computing facility show, the capability for LES calculations is advancing rapidly and further development and application of LES for this class of problem is to be expected.

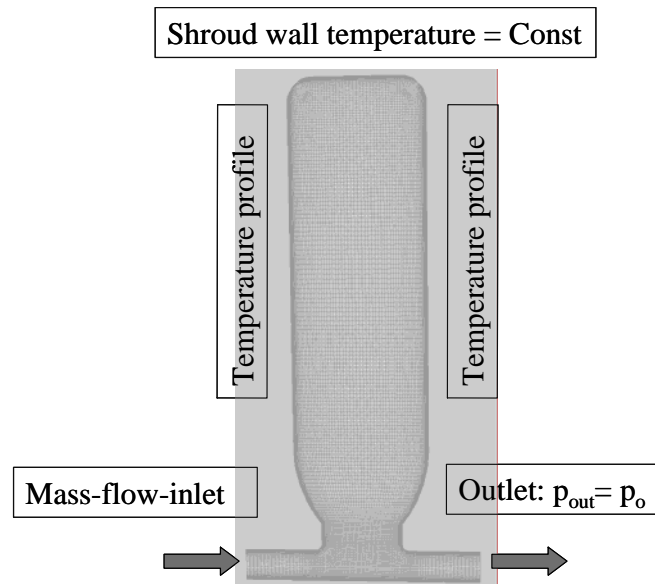


Figure 6. Section of solution domain and mesh for LES study of compressor disc cavity (Sun et al., 2006).

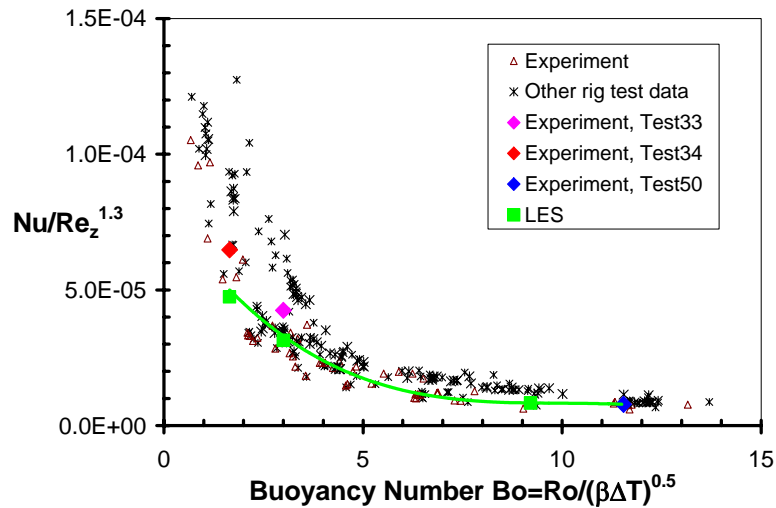


Figure 7. Comparison of shroud mean heat transfer between LES and experiment, Nusselt number (Nu) vs Buoyancy number (Bo) (Sun et al., 2006).

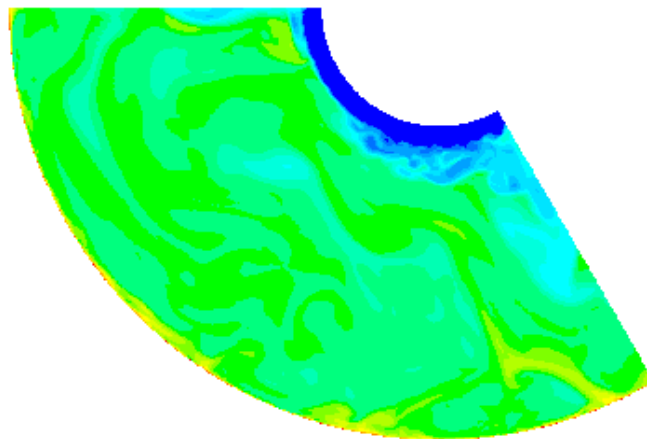


Figure 8. Instantaneous temperature contours on the mid axial plane.

3.3 Pre-swirled cooling air delivery systems

One of the main aims of the internal air system is to deliver cooling air to the vanes and blades at the minimum possible temperature. For the rotating components, it is possible to reduce the temperature of the coolant air relative to the blades by expanding the air in stationary nozzles angled in the direction of rotation. Most modern gas turbines use such a ‘pre-swirl’ system for the HP stage. A typical HP stage pre-swirl system is shown in figure 9. Flow is expanded in the pre-swirl nozzles, thereby acquiring a component of swirl velocity, before passing through the coverplate receiver holes onboard the rotating component and travelling out to the blade cooling passages. The pre-swirled flow can be contaminated by hotter, non pre-swirled flow from the inner labyrinth seal shown and various designs have been considered to avoid this contamination. This system is an important component of the internal air system of the engine and provides a good example of how methods within industry have evolved over the last few years.

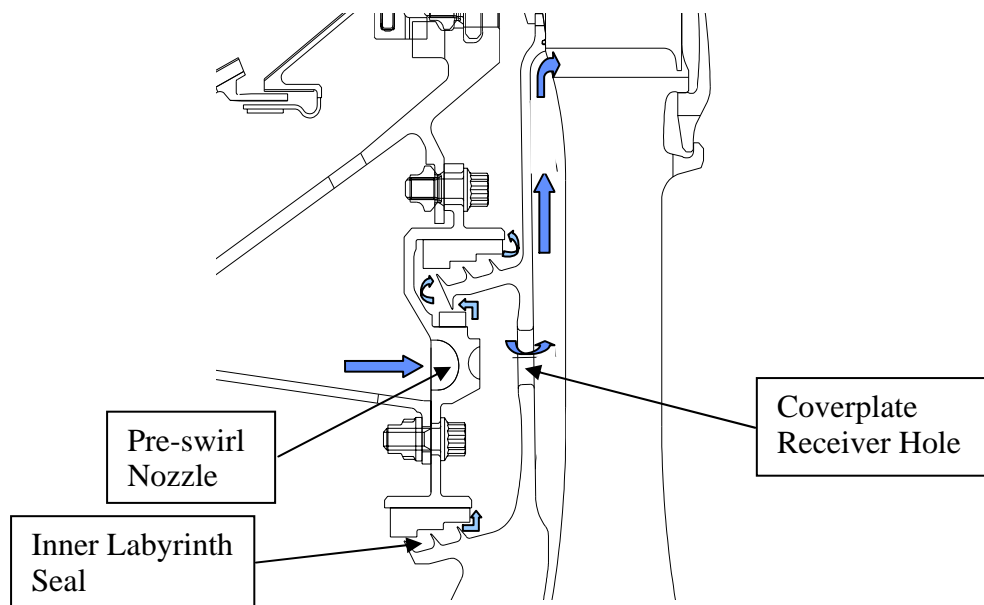


Figure 9. Typical HP Stage Pre-swirl System

Traditionally, research on pre-swirl systems has been carried out by rig and engine testing. The earliest published work is by Meierhofer and Franklin (1981) who described an experimental investigation demonstrating the potential benefits. Since the research work has primarily been experimental, various university groups with an appropriate experimental rig have published series of papers on this topic. There are experimental rigs at the universities of Bath (Lewis et al., 2006), Karlsruhe (Geis et al., 2003), and Sussex (Chew et al., 2003). Further references to earlier work at each group can be found in the references cited.

Early use of CFD for pre-swirl systems was reported by Staub (1992). The experimentally studied system was simplified in order to make the calculation possible, but nonetheless the general flow behaviour of the system was reproduced. Since then, all the above groups have applied CFD to support the interpretation of the experimental data. Various levels of approximation have been used. For example, Wilson et al. (1997) compare heat transfer results obtained from steady axisymmetric

models with their experimental data, obtaining reasonable agreement. Chew et al. (2005) show steady 3D models with the pre-swirl nozzles approximated as axisymmetric slits in order to reduce the problem to steady state.

With the growth in computing power discussed earlier, fully unsteady 3D modelling of pre-swirl systems has become possible both within industry and academia. Snowsill and Young (2006) describe the current position within industry. Fully unsteady 3D models are possible with the available computing hardware and examples are shown. However, the timescales for running these models are still very large, and approximations are sought to make CFD modelling feasible at early design stages.

The research challenges for pre-swirl systems are, then, very similar to other application areas. Faster solution methods (or good approximate solution methods) will enable earlier use of CFD in the design process. This will also require better integration of CFD with CAD, and more automatic meshing, to enable the entire process to run faster. Better integration of CFD with CAD will allow for more accurate engine geometry to be modelled. Increasingly, CFD will be coupled with the solid modelling to predict the metal temperatures directly.

3.4 Mixed Oil/Air Flows

As mentioned in the introduction, a further function of the air system is to isolate the oil system, preventing leakages. Thus, for example, a positive pressure difference is maintained across bearing chamber seals, with air flowing into the chamber and mixing with lubrication oil. Further mixed oil/air flows are found in internal gear boxes and hydraulic seals, and the effects of any oil leakage into the internal air system are of interest. An interesting account of a recent major European research project on engine oil systems is given by Klingsporn (2004).

Bearing chamber flows have been the subject of significant research effort. Much of this has been experimental work with a number of studies being published from Karlsruhe University; see, for example, Wittig et al., 1994, Glahn and Wittig, 1995, and Glahn et al. 1995, Gorse et al., 2003. Such work has shown complex flow features with droplets from the bearing spraying across the chamber, an oil film containing some air bubbles forming on the outer housing, and interaction of the air flow and droplets with the film. Early CFD analysis by Glahn et al. considered the air flow and droplet motion. A simplified integral analysis of the oil film (Chew, 1996) showed some very encouraging agreement with Wittig et al.'s heat transfer data. Film modelling has been further considered using integral methods and a commercial CFD code by Farall et al. (2003, 2004) with considerable attention given to the droplet/film interactions.

Apart from the film modelling studies, most two-phase flow CFD studies for internal fluid systems have used models developed for other application areas and available in commercial CFD codes. CFD capability for two phase flow continues to evolve quite rapidly and application is expected to increase in the future. For example, Young and Chew (2005) have considered use of the volume-of-fluid model introduced by Hirt and Nichols (1981). Some basic evaluations were carried out and successful application to a hydraulic seal was demonstrated. Denecke et al. (2006) have also applied the volume-of-fluid method to hydraulic seals. Further studies were reported by Robbe (2003). An interesting example is shown in figure 10. This presents CFD results for a simplified representation of the sump flow in a bearing chamber. A liquid film enters the sump through a uniform film at the left-hand side of the domain and must leave through a scavenge pipe. Solutions are obtained using the volume-of-fluid method. The two solutions show the effect of introducing a fillet on the scavenge exit at a fixed flow rate. Introducing the fillet produced a large reduction

in the depth of liquid in the sump. Such sensitivities confirm some of the difficulties inherent in the prediction of two phase flows.

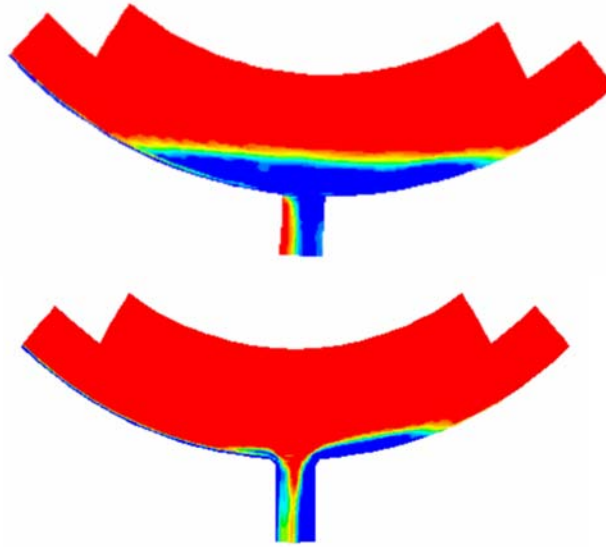


Figure 10. Contours of volume fraction for a sump flow model; top- without fillet on scavenge pipe entry, bottom - with fillet.

Further recent work (Sun, 2005) has shown that other two phase calculation techniques, such as the Euler-Euler (or two fluid) model, can be useful in internal flows. An interesting overview of multiphase CFD methods is given by van Wachen and Almstedt (2003) who conclude that application of multiphase CFD is very promising but requires further development. Work is needed to extend such models for engine applications. There is a growing body of experimental data suitable for evaluation of CFD. This includes data on droplet generation due to oil film and jet break-up (Glahn et al., 2002, 2003a, 2003b) and film behaviour (Eastwick et al., 2006). Such studies can provide the building blocks for a more comprehensive capability. However the complexity of two phase flows will ensure that modelling assumptions will be required and care required in application of CFD.

4. MAIN GAS PATH INTERACTION

The prevention or suppression of hot mainstream annulus gas ingestion into the inter-disc cavities is important in controlling disc temperatures. Thus turbine rim seal design and the cooling flow rates required to prevent ingestion have been subject to considerable investigation over the last few decades. As a result of this, semi-empirical methods have been developed for estimating ingestion due to disc pumping (e.g. Chew et al., 1992) and ingestion driven by circumferential pressure asymmetries in the main annulus (e.g. Scanlon et al., 2005, Johnson et al., 2006). Such methods are used in industry and may be supplemented by CFD calculations. However there is still considerable doubt as to the level of CFD modelling required for accurate modelling of the ingestion process.

In recent years, the need for more detailed consideration of secondary flow path features, such as the disc cavities, has been recognised by turbine and compressor aerodynamicists. Wellbourne and Okiishi (1998) report both experimental and CFD results for an axial compressor. These indicate that the effect of leakage flows on blading aerodynamics can result in higher losses than expected from simple mixing. In studies by Rosic et al. (2005) and Cherry et al. (2005) cavity flows were

included in multistage turbine models with mixing planes used at interfaces between rotating and stationary components. Cherry et al. recommend further modelling of secondary flow paths as CFD improves in the future. Rosic et al. state that full calculation of leakage and cavity flows is needed to obtain good agreement with experiment.

The following subsection presents some recent results for turbine rim seal flows, focussing on conditions in which ingestion of annulus gas into the disc cavity does occur. These gives some insight into the complex flow physics associated with ingestion. Subsection 4.2 then considers CFD modelling for whole turbines, including detailed geometrical features and disc cavities. With the advances made in parallel computation described in section 2.2, such calculations are now feasible.

4.1 Turbine rim seal flows

Figure 11 shows a two stage turbine, as studied by Cao et al. (2003). In this turbine the disc rim gap under consideration consists of a simple axial gap between the rotating disc and the stationary vane diaphragm. While this is not typical of aeroengines, the flow phenomena predicted and measured in this case are of particular interest. Cao et al.'s CFD models included the disc cavity and main flow annulus, but did not include rotating blades or stationary vanes. However, even with fully axisymmetric and steady boundary conditions, the flow was predicted to be 3D and unsteady. Figure 12 shows the CFD results for the radial component of velocity on an axial plane mid-way across the outer section of the cavity. At the rim gap, alternate regions of cavity inflow and outflow occur around the circumference. When viewed as a time series in the rotating frame of reference, this pattern rotates slowly in the opposite direction to the disc rotation. Thus in the absolute frame of reference, the flow pattern rotates at slightly less than disc speed.

Cao et al. obtained experimental confirmation of their CFD results from the root pressure transducers shown in figure 1. As shown in figure 13, Fourier analysis of the pressure signal reveals similar frequencies to the CFD. The measurements in this figure are for a range of rotor speeds at constant pressure ratio and level. The frequencies in question only increase slightly with rotor speed, and can be compared with the engine orders that are just visible as diagonal straight lines.

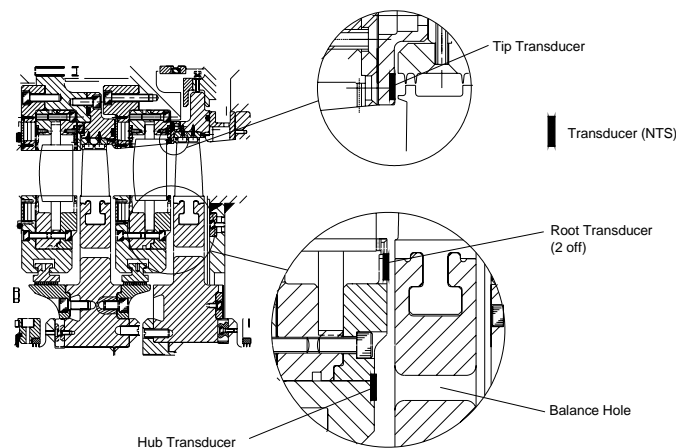


Figure 11. Detail of the stage 2 turbine considered by Cao et al. (2003).

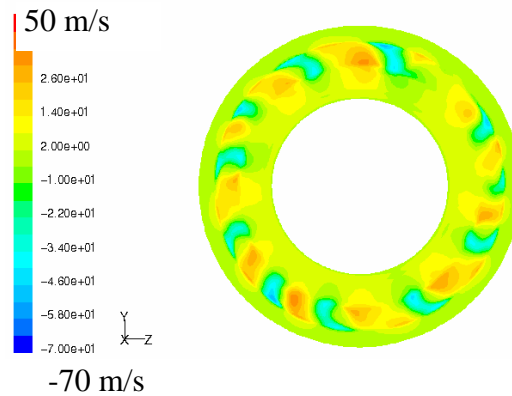
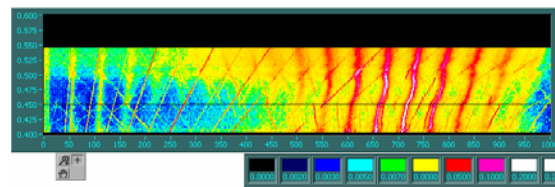


Figure 12. Instantaneous radial velocities on the mid-cavity axial plane.



X – Axis: Frequency, Y – Axis: rotor u/C_o ,
Intensity: $p(\text{rms}) / p_{\text{Abs}} * 100\%$

Figure 13. Fourier analysis of results from ‘root 2’ transducer from a run with varying rotor speed.

Instabilities and flow unsteadiness in rim seal flows are discussed further by Boudet et al. (2005, 2006). For a rim seal geometry representative of aeroengines, and previously studied experimentally by Gentilhomme (2004), unsteady CFD solutions including both blades and vanes were obtained. The turbine geometry studied and the Fourier decomposition of the pressure at a reference point in the outer part of the disc cavity are shown in figure 14. Further investigation of the Fourier component at 44% of blade passing frequency showed it to be associated with instabilities of the flow within the rim seal itself. The signals at 12% and 56% of blade passing frequency result from non-linear interactions. Further investigations of such phenomena are clearly needed, including full 360° turbine calculations and the use of large eddy simulation to investigate the effects of turbulence within the rim seals. It may be noted that both Cao et al. and Boudet et al. have used conventional Reynolds-averaged models of turbulence in their calculations.

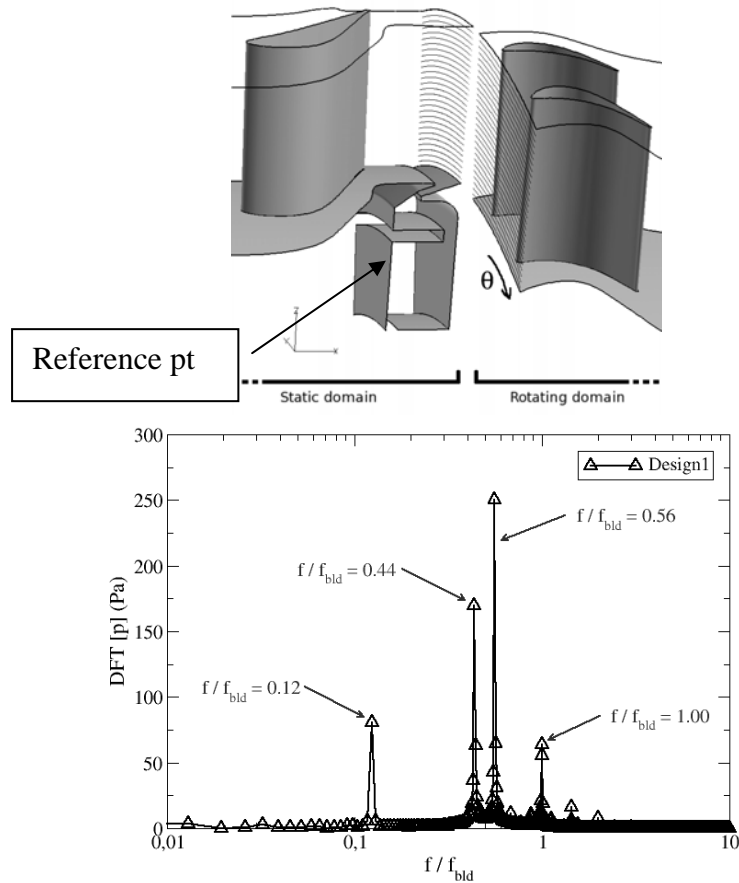


Figure 14. Turbine stage considered and Fourier components of a pressure at a reference point in the disc cavity (Boudet et al., 2006).

4.2 Whole turbine modelling

As was noted in the introduction to this section, over the last few years mainstream blade turbomachinery computations have increasingly attempted to include the secondary flow path features. Traditionally, the annulus has been approximated as a smooth idealised geometry, although leakage and cavity flows are now sometimes included. Figure 15 shows a state of the art computation of this type, in which some of the rim seal geometry has been modelled. A full multi-stage calculation has been carried out for a three shaft turbine consisting of one HP stage, one IP stage, and 5.5 low pressure (LP) stages. This model has 12 million mesh nodes, 37 million cell edges, and converges to steady state in 1 to 3 hours on 256 IBM POWER5 processors.

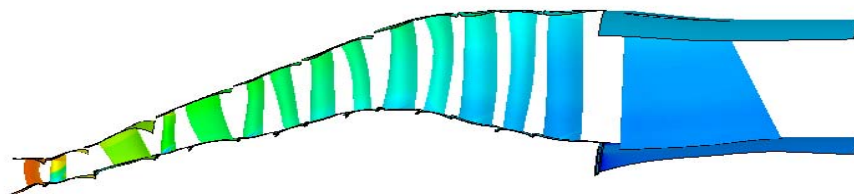


Figure 15. Full Turbine CFD Model for the Rolls-Royce plc Trent 500 Turbine with Idealised Annulus Geometry

As improvements in compressor and turbine performance have become increasingly hard to achieve, attention has focussed on the interactions of the secondary path flows with the main annulus flow, and on the effect of modelling the real engine geometry. This requires a much greater integration of the CFD process with the CAD geometry. An example of this type of calculation is shown in figures 16 and 17. Figure 16 shows the HP stage from a Pro/Engineer model of the Rolls-Royce plc Trent 500 turbine. The model is parameterised to allow the cold to hot geometry changes for various running conditions to be applied via the model parameters. A mesh can then be generated from this model using a commercial mesh generator. The coloured surfaces delineate the boundaries of the domain to be meshed and a typical computational domain for an HP stage computation is shown in figure 17. The mesh for this case has 19 million nodes with 91 million edges. A full unsteady HP stage computation of a 72° sector for this domain (consisting of 8 vanes, 14 rotor blades, and 9 bolts) takes around 24 hours on 256 IBM POWER5 processors. This calculation is now being extended to include the IP and LP turbines. Such calculations have become feasible due to the advances in parallel processing and will become more prevalent as computing power increases.

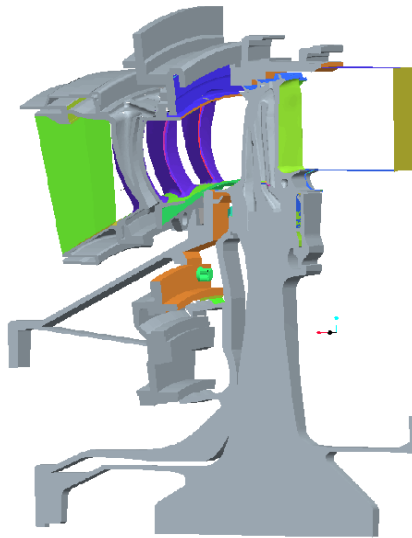


Figure 16. CAD Model of HP Stage for Rolls-Royce plc Trent 500

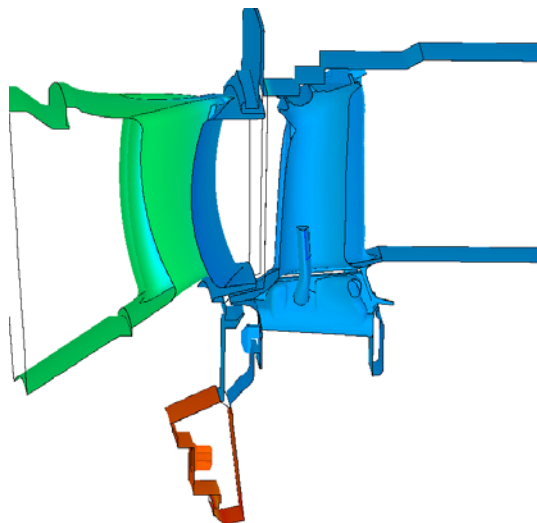


Figure 17. CFD Domain for HP Stage Model

5. CONCLUSIONS

As illustrated above, computer modelling is having a major impact in air systems research and design. Where appropriate and with some care, CFD may be used in the design process with considerable confidence and it is being embedded within design optimisation methods and within thermal analysis methodology. While at one time specialist CFD codes were used for air systems, now most air system studies are conducted using commercial CFD packages or more general proprietary codes. As compressor and turbine aerodynamicists include more geometric detail in their CFD models, further convergence in choice of CFD codes can be expected. The Hydra code developed by Rolls-Royce and its University partners (Crumpton et al., 2002, Shahpar et al., 2003, Hills, 2006) is a good example of this.

CFD has been shown to be contributing to understanding of buoyancy-driven rotating cavity, mixed oil/air, and rim sealing flows. For the cavity flows, LES techniques are looking very promising, giving a more consistent representation of the large scale turbulence than conventional Reynolds-averaged methods. Two phase flow CFD methods are now gaining acceptance for some practical applications in industry. For rim sealing flows, CFD has shown the shortcomings in commonly used engineering methods which cannot capture the complex flow physics involved. Further advances in all these areas can be expected in the next few years.

Parallel computing has had a large impact in CFD in the last few years, and this trend is set to continue. Excellent parallel efficiency has been demonstrated on over a thousand computer nodes with about 20,000 mesh nodes per processor. As computer hardware continues to develop machines with many thousands of processors will become available, and there is a need to further develop engineering software and systems to exploit these. Input generation, links to CAD, and output processing need to be considered concurrently with the CFD solvers. Further automation of the CFD process, particularly mesh generation and adaption, are required to fully exploit the present CFD capability.

It may be concluded that although CFD for internal air systems and understanding of these flows has progressed tremendously in the last 30 years, there is considerable scope for further improvements, and these can be expected.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge contributions to the work described above from colleagues at the Thermo-Fluid System University Technology Centre, Rolls-Royce plc and other collaborating institutions. Financial support from Rolls-Royce plc, the Engineering and Physical Sciences Research Council, the Department of Trade and Industry, the European Commission, Alstom Power and the University of Surrey is also gratefully acknowledged.

In this brief review it has not been possible to include reference to the many papers that have influenced our work over the years. Apologies to those authors whose work we have failed to include.

REFERENCES

Armstrong, I. and Edmunds, T.E., 1989. Fully automatic analysis in the industrial environment. *Proc. 2nd Int. Conf. on Quality Assurance and Standards in Finite Element Analysis*, NAFEMS.

Bohn, D., Bonhoff, H., Schonenborn, H., and Wilhelmi, H., 1995a. Validation of a Numerical Model for the Coupled Simulation of Fluid Flow and Diabatic Walls with Application to Film-Cooled Turbine Blades. *VDI-Berichte*, **1196**, 259-272.

Bohn, D., Deuker, E., Emunds, R., Gorzelitz, V., 1995b, "Experimental and theoretical investigations of heat transfer in closed gas-filled rotating annuli", *ASME Journal of Turbomachinery*, **117**, 175-183.

Bohn, D., Ren, J., Kusterer, K., 2003a. Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries. *Proc. ASME Turbo Expo*, paper 2003-GT-38369.

Bohn, D., Heuer, T., and Kusterer, K., 2003b. Conjugate Flow and Heat Transfer Investigation of a Turbo Charger. Part I: Numerical Results. *Proc. ASME Turbo Expo*, paper 2003-GT-38445.

Bohn, D., Moritz, N., and Wolff, M., 2003c. Conjugate Flow and Heat Transfer Investigation of a Turbo Charger. Part II: Experimental Results. *Proc. ASME Turbo Expo*, paper 2003-GT-38449.

Boudet, J., Autef, V.N.D., Chew, J.W., Hills, N.J. and Gentilhomme, O., 2005. Numerical simulation of rim seal flows in axial turbines. *The Aeronautical Journal*, **109**, 373- 383.

Boudet, J., Chew, J.W., and Hills, N.J., 2006. Numerical simulation of the flow interaction between turbine main annulus and disc cavities. *Proc. ASME Turbo Expo*, paper GT2006-90307.

Cao, C., Chew, J.W., Millington, P.R. and Hogg, S., 2003. Interaction of rim seal and annulus flows in an axial flow turbine. *Proc. ASME Turbo Expo*. Paper GT2003-38368. (Also *ASME J. Eng. for Gas Turbines and Power*, **126**, 786-793.)

Cherry, D., Wadia, A., Beacock, R., Subramanian, M. and Vitt, P., 2005. Analytical investigation of low pressure turbine with and without endwall gaps seals and clearance features. *Proc. ASME Turbo Expo*, paper GT2005- 68492.

Chew, J.W., 1984. Development of a computer program for the prediction of flow and heat transfer in rotating cavities. *Int. J. Num. Methods in Fluids*, **4**, 667-683.

Chew, J.W., 1990. Prediction of rotating disc flow and heat transfer in gas turbine engines. *Proc. 3rd Int. Symp. Transport Phenomena and Dynamics of Rotating Machinery*, Honolulu, Hemisphere Pub., 145-160.

Chew, J.W., 1996. Analysis of the oil film on the inside surface of an aero-engine bearing chamber housing. ASME Gas Turbine Conference, paper 96-GT-300.

Chew, J.W., Ciampoli, F., Hills, N., and Scanlon, T., 2005. Pre-swirled cooling air delivery system performance. *Proc. ASME Turbo Expo*, paper GT-2005-68323.

Chew, J.W., Dadkhah, S. and Turner, A.B., 1992. Rim sealing of rotor-stator wheelspaces in the absence of external flow. *ASME J. Turbomachinery*, **124**, 306-315.

Chew, J.W., Hills, N.J., Khalatov, S., Scanlon, T., and Turner A.B., 2003. Measurement and analysis of flow in a pre-swirled cooling air delivery system. *Proc. ASME Turbo Expo*, paper GT-2003-38084.

Chew, J.W., Taylor, I.J., and Bonsell, J.J., 1996. CFD Developments for Turbine Blade Heat Transfer. *Proc. 3rd Int. Conf. on Reciprocating Engines and Gas Turbines*, IMechE.

Chew, J.W. and Vaughan, C.M., 1988. Numerical predictions for the flow induced by an enclosed rotating disc. ASME Gas Turbine and Aeroengine Congress, paper 88-GT-127.

Ciampoli, F., Chew, J.W., Shahpar, S. and Willocq, E., 2006. Automatic optimisation of pre-swirl nozzle design. *Proc. ASME Turbo Expo*, paper GT2006-90249. (Also to appear in *ASME J. Engng Gas Turbines and Power*.)

Crumpton, P.I., Muller, J-D., Giles, M.B., 2002. Edge based multigrid schemes and preconditioning for hybrid grids. *AIAA Journal*, **40**, 1954-1960.

Daily J.W. and Nece, R. 1960. Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs. *ASME Journal of Basic Engineering*,

82, 217-232.

Dawes, W.N., 2006. Eliminating serial bottlenecks not just in the flow solving but also in the mesh generation and geometry management. *Proc. ASME Turbo Expo*, paper GT-2006-90620.

Denecke, J., Schramm, V., Dullenkopf, K., Bauer, H-J, Klingsporn, M., Hein, S. and Peitsch, D., 2006. Advanced hydraulic seal design for high temperature environments. *Proc. ASME Turbo Expo*, paper GT2006-90651.

Eastwick, C., Bertin, L. and Johnson, G., 2006. The effect of obstacles in a liquid film. *Proc. ASME Turbo Expo*, paper GT2006-90520.

Farrall, M., Hibberd, S. and Simmons, S., 2003. Modelling oil droplet/film interaction in an aero-engine bearing chamber. ICLASS Conference, Sorrento.

Farrall, M., S., Simmons, Hibberd, S. and Gorse, P., 2004. Modelling oil droplet/film interaction in an aero-engine bearing chamber and comparison with experimental data. *Proc. ASME Turbo Expo*, paper GT-2004-53698.

Geis, T., Dittmann, M., and Dullenkopf, K. (2003). Cooling air temperature reduction in a direct transfer preswirl system. *Proc. ASME Turbo Expo*, paper GT-2003-38231.

Gentilhomme, O.J.P., 2004. Turbine rim seal ingestion. DPhil thesis, University of Sussex.

Glahn, A., Blair, M.F., Allard, K.L., Busam, S., Schäfer, O. and Wittig, S., 2003. Disintegration of oil jets emerging from axial passages at the face of a rotating cylinder. *ASME J. Engng for Gas Turbines and Power*, **125**, 1003-1010.

Glahn, A., Blair, M.F., Allard, K.L., Busam, S., Schäfer, O. and Wittig, S., 2003. Disintegration of oil jets emerging from radial holes in a rotating cylinder. *ASME J. Engng for Gas Turbines and Power*, **125**, 1011-1020.

Glahn, A., Busam, S., Blair, M.F., Allard, K.L. and Wittig, S., 2002. Droplet generation by disintegration of oil films at the rim of a rotating disc. *ASME J. Engng for Gas Turbines and Power*, **124**, 117-124.

Glahn, A. and Wittig, S., 1995. Two-phase air/oil flow in aero engine bearing chambers – Characterisation of oil film flows. ASME Turbomachinery Conference, paper 95-GT-114.

Glahn, A., Kurreck, M., Willmann, M. and Wittig, S., 1995. Feasibility study on oil droplet flow investigations inside aero engine bearing chambers. ASME Turbomachinery Conference, paper 95-GT-100.

Gorse, P., Willenborg, K., Busam, S., Ebner, J., Dullenkopf, K. and Wittig, S., 2003. 3D-LDA measurements in an aero-engine bearing chamber. *Proc. ASME Turbo Expo*, paper GT2003- 38376.

Gosman, A.D., Koosinlin, M.L., Lockwood, F.C. and Spalding, D.B., 1976. Transfer of heat in rotating systems. ASME Turbomachinery Conference, paper 76-GT-25.

Heselhaus, A., Vogel, D.T., and Krain, H., 1994. Coupling of 3D Navier-Stokes External Flow Calculations and Internal 3D Heat Conduction Calculations for Cooled Turbine Blades. AGARD, Heat Transfer and Cooling in Gas Turbines, 40.1-40.9

Hills, N.J., 2006. Achieving high parallel performance for an unstructured unsteady Turbomachinery CFD code. *The Aeronautical Journal* (Special Edition), submitted for publication.

Hirt, C.W., and Nichols, B.D., 1981, “Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, *J. Comput. Phys.*, **39**, 201-225.

Ho, Y.H., Athavale, M.M., Forry, J.M., Hendricks, R.C. and Steinetz, B.M., 1996. Numerical Simulation of Secondary Flow in Gas Turbine and Disc Cavities

including Conjugate Heat Transfer. ASME Turbomachinery Conference, paper 96-GT-67.

Iacovides, H. and Chew, J.W., 1993. Prediction of heat transfer in rotating disc systems. *Int. J. Heat and Fluid Flow*, **14**, 146-154.

Isaac Newton Institute for Mathematical Sciences, 1999. Workshop on future strategies towards understanding and prediction of turbulent systems. Lecture notes issued by INIMS, University of Cambridge, Cambridge, U.K.

Illingworth, J., Hills, N. and Barnes, C., 2005. 3D Fluid-Solid Heat Transfer Coupling of an Aero-engine Preswirl System, *Proc. ASME Turbo Expo*, paper 2005-GT-68939.

Jameson, A. and Caughey, D.A., 2001. How many steps are required to solve the Euler equations of steady compressible flow: in search of a fast solution algorithm. AIAA paper 2001-2673.

Johnson, B.V., Jakoby, R., Bohn, D.E. and Cunat, D., 2006. A method for estimating the influence of time-dependent vane and blade pressure fields on turbine rim seal ingestion. *Proc. ASME Turbo Expo*, paper GT2006-90853.

Karypsis, G. and Kumar V., 1999. A fast and high quality scheme for partitioning irregular graphs. *SIAM J. Scientific Computing*, **20**, 359-392.

King, M. P., Wilson, M., and Owen, J. M., 2005, "Rayleigh-Benard Convection in Open and Closed Rotating Cavities", *Proc. ASME Turbo Expo 2005*, paper GT2005-68948.

Kirkpatrick, A.T. and Bohn, M., 1986. An experimental investigation of mixed cavity convection in the high Rayleigh number regime. *Int. J. Heat Mass Transfer*, **29**, 69-82.

Klingsporn, M., 2004. Advanced Transmission and Oil System Concepts for Modern Aero-Engines *Proc. ASME Turbo Expo*, paper GT2004-53578.

Lee, C.W., Palma, P.C., Simmons, K. and Pickering, S.J., 2004. Comparison of CFD and PIV data for the airflow in an aero-engine bearing chamber. *Proc. ASME Turbo Expo*, paper GT2004-53281.

Lewis, P., Wilson, M., Lock, G., and Owen, J.M., 2006. Physical interpretation of flow and heat transfer in pre-swirl systems. *Proc. ASME Turbo Expo*, paper GT-2006-90132.

Li, H., and Kassab, A.J., 1994. A Coupled FVM/BEM Approach to Conjugate Heat Transfer in Turbine Blades. AIAA paper 94-1981.

Long, C.A., Alexiou, A. and Smout, P.D., 2003, "Heat transfer in H.P. compressor gas turbine internal air systems: measurements from the peripheral shroud of a rotating cavity with axial throughflow", HEFAT2003, 2nd Intl Conf. on Heat Transfer, Fluid Mechanics and Thermodynamics, paper LC1.

Long, C.A.; Tucker, P.G. Source, 1994, "Numerical computation of laminar flow in a heated rotating cavity with an axial throughflow of air", *International Journal of Numerical Methods for Heat and Fluid Flow*, **4**, 347-365.

Lumley, J.L., 1999. Prediction Methods in Turbulence, Workshop on Future Strategies towards Understanding and Prediction of Turbulent Systems, Isaac Newton Institute, Cambridge.

May, N.E., Chew, J.W. and James, P.W., 1994. Calculation of turbulent flow for an enclosed rotating cone. *ASME J. Turbomachinery*, **116**, 548-554.

Meierhofer B. and Franklin C. J., 1981. An investigation of preswirled cooling airflow to a turbine disc by measuring the air temperature in the rotating channels. ASME Turbomachinery Conference, paper 81-GT-132.

Mirzamoghadam, A.V., and Xiao, Z., 2000. Flow and Heat Transfer in an Industrial Rotor-Stator Rim Sealing Cavity. ASME Turbo Expo, paper 2000-GT-285.

Monico, R.D. and Chew, J.W., 1992. Modelling the Thermal Behaviour of

Turbomachinery Discs and Casings. AGARD PEP Symposium on Heat Transfer and Cooling in Gas Turbines, 24.1 – 24.9.

Mulder, W., 1989. A new approach to convection problems. *J. Comp. Phys.*, **83**, 303–323.

Owen, J.M., Abrahamsson, H. and Lindblad, K., 2006. Buoyancy-induced flow in open rotating cavities. *Proc. ASME Turbo Expo*, paper GT-2006-91134.

Owen, J.M, and Rogers, R.H., 1989. *Flow and Heat Transfer in Rotating Disc Systems: Rotor-Stator Systems (Vol. 1)*, Research Studies Press, Taunton (John Wiley, New York).

Owen, J.M, and Rogers, R.H., 1995. *Flow and Heat Transfer in Rotating-Disc Systems : Rotating Cavities (Vol. 2)*. Research Studies Press, Taunton (John Wiley, New York).

Pierce, N.A. and Giles, M., 1997. Preconditioned multigrid methods for compressible flow calculations on stretched grids. *J. Comp. Phys.*, **136**, 425-445.

Pope, S.B., 2004, Ten questions concerning the large-eddy simulation of turbulent flows. *New Journal of Physics*, **6**, 35.

Rigby, D.L. and Lepicovsky, J., 2001. Conjugate Heat Transfer Analysis of Internally Cooled Configurations, *Proc. ASME Turbo Expo*, paper 2001-GT-0405.

Robbe, E., 2003. Unpublished work. University of Surrey.

Rosic, B., Denton, J.D. and Pullan, G., 2005. The importance of shroud leakage modelling in multistage turbine flow calculations. *Proc. ASME Turbo Expo*, paper GT2005-68459.

Rossow, C-C., 2006. Toward efficient computation of compressible and incompressible flows. AIAA paper 2006-3522.

Scanlon, T., Wilkes, J., Bohn, D. and Gentilhomme, O., 2004. A simple method of estimating ingestion of annulus gas into a turbine rotor stator cavity in the presence of external pressure gradients. *Proc. ASME Turbo Expo*. Paper GT2004-53097.

Shahpar, S., Giacche, D. and Lapworth L., 2003. Multi-objective design and optimisation of bypass outlet guide vanes. *Proc. ASME Turbo Expo*, paper GT2003-38700.

Smout, P.D., Chew, J.W. and Childs, P.R.N., 2002. ICAS-GT: A European collaborative research programme on internal cooling air systems for gas turbines. *Proc. ASME Turbo Expo*, paper GT-2002-30479.

Snowsill, G., and Young, C., 2006. The application of CFD to underpin the design of gas turbine pre-swirl systems. *Proc. ASME Turbo Expo*, paper GT-2006-90443.

Staub, F.W., 1992. Rotor Cavity Flow and Heat Transfer with Inlet Swirl and Radial Outflow of Cooling Air. ASME Turbomachinery Conference, paper GT-92-378.

Sun, Z., 2005. Unpublished work. University of Surrey.

Sun, Z, Kilfoil, A., Chew, W. J and Hills, N. J., 2004, “Numerical simulation of natural convection in stationary and rotating cavities”, *Proc. ASME Turbo Expo*, paper GT2004-53528.

Sun, Lindblad, Chew and Young, 2006. LES and RANS investigations into buoyancy-affected convection in a rotating cavity with a central axial throughflow. *Proc. ASME Turbo Expo*, paper GT2006-90251.

Van Wachen, B.G.M and Almstedt, A.E., 2003. Methods for multiphase computational fluid dynamics. *Chemical Engineering Journal*, **96**, 81-98.

Verdicchio, J., Chew, J.W. and Hills, N.J., 2001. Coupled fluid/solid heat transfer computation for turbine discs. *Proc. ASME Turbo Expo*, paper 2001-GT-205.

Virr, G.P., Chew, J.W. and Coupland, J. 1994. Application of computational fluid dynamics to turbine disc cavities. *ASME J. Turbomachinery*, **116**, 701-708.

Volkov, K., 2006. Unpublished work. University of Surrey.

Wilson, M., Pilbrow, R., and Owen, J.M., 1997. Flow and heat transfer in a pre-swirl rotor-stator system. *ASME J. Turbomachinery*, **119**, pp 363-373.

Wellbourn, S.R. and Okiishi, T.H., 1998. The influence of shrouded stator cavity flows on multistage compressor performance. ASME Gas Turbine Expo, paper 98-GT-12.

Wittig, S., Glahn, A. and Himmelsbach, J., 1994. Influence of high rotational speeds on heat transfer and oil film thickness in aero engine bearing chambers. *ASME J. Engng for Gas Turbines and Power*, **116**, 395-401.

Wong, L-S., 2002. "Flow and Heat Transfer in Rotationally Induced Buoyancy Flow", D. Phil thesis, University of Sussex.

Yamada, Y. and Ito, M., 1975. On the frictional resistance of enclosed rotating cones. *Bulletin of the JSME*, **18**, 1026.

Yamada, Y. and Ito, M., 1979. Frictional resistance of enclosed rotating cones with superposed throughflow. *ASME J. of Fluids Engineering*, **101**, 259.

Young, C. and Chew, J.W., 2005, Evaluation of the volume of fluid modelling approach for simulation of oil/air system flows. *Proc. ASME Turbo Expo*, paper GT2005-68861.