Zixiang Sun
Fluids Research Centre,
School of Engineering,
University of Surrey,
Guildford, Surrey, GU2 7XH, UK
e-mail: Zixiang.Sun@surrey.ac.uk

Klas Lindblad
Volvo Aero Corporation,
Dept 7161,
SE-461 81 Trollhättan, Sweden
e-mail: Klas.Lindblad@volvo.com

John W. Chew
Fluids Research Centre,
School of Engineering,
University of Surrey,
Guildford, Surrey, GU2 7XH, UK
e-mail: J.Chew@surrey.ac.uk

Colin Young
Rolls-Royce plc,
PO Box 31,
Derby, DE24 8BJ, UK
e-mail: colin.young@rolls-royce.com

LES and RANS Investigations Into Buoyancy-Affected Convection in a Rotating Cavity With a Central Axial Throughflow

The buoyancy-affected flow in rotating disk cavities, such as occurs in compressor disk stacks, is known to be complex and difficult to predict. In the present work, large eddy simulation (LES) and unsteady Reynolds-averaged Navier-Stokes (RANS) solutions are compared to other workers' measurements from an engine representative test rig. The Smagorinsky-Lilly model was employed in the LES simulations, and the RNG k- ϵ turbulence model was used in the RANS modeling. Three test cases were investigated in a range of Grashof number $Gr=1.87$ to 7.41×10^8 and buoyancy number $Bo=1.65$ to 11.5 . Consistent with experimental observation, strong unsteadiness was clearly observed in the results of both models; however, the LES results exhibited a finer flow structure than the RANS solution. The LES model also achieved significantly better agreement with velocity and heat transfer measurements than the RANS model. Also, temperature contours obtained from the LES results have a finer structure than the tangential velocity contours. Based on the results obtained in this work, further application of LES to flows of industrial complexity is recommended. [DOI: 10.1115/1.2364192]

1 Introduction

In order to achieve further improvements in turbomachinery internal air system design and component temperature prediction, an accurate knowledge of the flow and heat transfer behavior of the system is essential. Computational fluid dynamics (CFD) is now commonly used to support the design of many elements of the internal air system, but the prediction of buoyancy-affected flow in high-pressure compressor disk cavities has proven particularly difficult. Recognizing the need for progress in this area, experimental and computational research studies of convection in rotating cavities have been included in the European Union (EU) sponsored ICAS-GT (internal cooling air systems for gas turbines) and ICAS-GT2 research. Most of the work described here was undertaken as part of the ICAS-GT2 research program and arose from the recognition that LES methods may have distinct advantages over RANS methods for this problem, which is known to give rise to large scale unsteady flow structures. In LES the larger turbulent eddies are simulated, with smaller (subgrid scale) eddies being modeled. The subgrid scale modeling is dependent on the mesh size and (unlike RANS models) is designed to allow development of the larger resolved eddies that interact with the mean flow.

A number of relevant experimental studies have been reported using various experimental techniques, such as velocity measurement, heat transfer, and/or flow visualization. The geometries considered in the experiments include simply enclosed rotating cavities (for example, [1]), the idealized plane disc with an axial throughflow [2–4], as well as geometries that are more representative of engines [5–9]. Typically, the cavity flows were found to be highly three-dimensional (3D) and unsteady, with various flow structures, such as “radial arms,” penetrating the cavities. Strong dependency on rotational speed, axial throughflow strength, and

temperature gradients has also been identified. The disk heat transfer regimes can be broadly divided into three regimes: buoyancy dominated, axial throughflow dominated, and transitional. Heat transfer correlations are often related to rotational Rayleigh or Grashof numbers, through analogy to results for free convection in gravitational fields. Dependencies on Reynolds number and Rossby number ($Ro=W/a\Omega$ where W is the axial throughflow bulk velocity and $a\Omega$ is the disk speed at the inner radius) have also been identified. In accordance with mixed convection under gravity, different regimes have been associated with different ranges of a buoyancy number ($Bo=Ro/[\beta\Delta T]^{0.5}$, where ΔT is the driving temperature difference and β is the coefficient of thermal expansion). The experimental correlations are inevitably based on particular geometries and consequently often display significant scatter. In these difficult experiments, measurement uncertainty can also be significant.

A number of numerical simulations have also been reported. For low rotational Grashof number convection in a sealed cavity with different uniform disk temperatures, Chew [10] found that an axisymmetric, steady, laminar flow model gave good agreement with heat transfer measurements. Bohn et al. [11,12] reported that laminar steady models gave “quite good” agreement with measurements for axially heated sealed rotating cavities at higher Grashof number, and also noted the tendency of the flow to become unsteady with radial heating. It is now widely accepted that negative radial density gradients cause flow unsteadiness analogous to Rayleigh-Benard convection under gravity. Long and Tucker [13] obtained laminar unsteady CFD solutions for a heated cavity with axial throughflow and claimed reasonable accord with an earlier experimental correlation for disk surface heat transfer. However, limitations of the laminar model were also recognized. Iacovides and Chew [14] applied a steady, axisymmetric RANS model to the high Rossby number “throughflow-dominated” regime and concluded that the calculated disk heat transfer was comparable to measured data, but firm conclusions could not be drawn due to experimental and modeling uncertainties. Other studies (for example, [15,16]) have been reported using conventional k- ϵ turbulence models in unsteady 3D simulations, but the

Contributed by the International Gas Turbine Institute (IGTI) of ASME for publication in the JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received October 1, 2005; final manuscript received February 1, 2006. IGTI Review Chair: R. S. Abhari. Paper presented at the ASME Turbo Expo 2006: Land, Sea and Air (GT2006), Barcelona, Spain, May 8–11, 2006, Paper No. GT2006-90251.

Fig. 1 Schematic illustration of the rotating cavity

information available in the open literature is limited. Wong's comparisons [16] with heat transfer measurements gave mixed results. In a recent publication, King et al. [17] 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 [1] experimental correlation.

As a precursor to the present work, Sun et al. [18] studied high Rayleigh number free convection under gravity in a stationary cube and under centripetal force in a rotating cavity. Somewhat surprisingly, laminar, unsteady three-dimensional 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. Large-scale flow structures were found 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. As the laminar unsteady/LES model would be unlikely to be suitable for modeling cavities with a central axial throughflow, the Smagorinsky-Lilly model was used in the present LES study. The results obtained are compared to both experimental data and with RANS solutions. The configuration studied experimentally by Long et al. [19] was selected for this study, as it was considered to be reasonably representative of engine operating conditions, and shroud heat transfer and cavity velocity measurements were available. As far as the authors are aware, this is the first application of LES to the problem of buoyancy affected rotating cavity flow with axial throughflow.

2 Model Definition

2.1 Geometry and Meshing. The geometry considered in this work is based on the rig described by Long et al. [19]. A detailed description of the test rig and experiments performed can be found in the thesis by Alexiou [20]. This test rig included a stack of model compressor disks forming four rotating cavities. The third cavity was selected for detailed investigation and is illustrated schematically in Fig. 1. The ratio of the inner and outer radii is $a/b=0.319$. The gap ratio between the cavity width s and the outer radius b of the cavity is $G=s/b=0.195$. A stationary shaft, radius $r_s=0.886a$ formed the inboard flow boundary. During the experiments, the disks and connecting cavity shroud were rotated at a constant speed, and heating was applied to the outer surface of the rotor.

The computational meshes were generated in GAMBIT [21] by first defining a 2D mesh in the plane of Fig. 1 and then extruding this circumferentially. Following Sun et al. [18], sector models were generated instead of a full 360 deg cavity to reduce computing requirements. Three sector models, with arc lengths of 45, 90, and 120 deg, as well as a full 360 deg model were generated. The mesh sizes for these models were constructed from 1.36, 3.24,

Fig. 2 Sectional view of the 120 deg mesh

4.07, and 12.2 million hexahedral cells, respectively. Mesh spacing was such that the value of the mean dimensionless near-wall distance (y^+) was 0.6–1.0 in the LES calculations. The number of mesh cells in the wall boundary is typically 20. The expansion ratio of the boundary mesh cells is 1.1. Such a mesh resolution may be regarded as adequate from a general consideration of LES practice. Supporting evidence is also given by the mesh independence investigation conducted in the authors' previous study [18] for high Rayleigh number free convection under gravity in a stationary cube and under centripetal force in a rotating cavity, although mesh independence has not been demonstrated in the present investigation. A sectional view of the 120 deg mesh, which has 250 equally spaced points in the circumferential direction, is shown in Fig. 2.

2.2 Boundary Conditions and CFD Modeling. The FLUENT CFD code version 6.1 [22] was used in all the calculations presented here. The domain boundary was divided into rotating surfaces, stationary surfaces, inlet, and exit and circumferentially periodic sections, and appropriate conditions were imposed. The inlet mass flow rate and the outlet static pressure were defined in accordance with experimental data. In FLUENT LES computations, the stochastic velocity components of the inflow are accounted for by superposing random perturbations on individual velocity components, which are obtained in terms of the assigned turbulence intensity, mean flow velocity, and the Gaussian random function. In recognition of the strong mixing effects between the central axial throughflow and swirl flow in the upstream cavities, the inlet turbulence intensity was set to 20%. For two LES cases discussed later (tests 33 and 34), the inlet mean swirl velocity was specified from the tangential velocity profile at outlet obtained from a steady, axisymmetric RANS simulation of an annular pipe flow with similar geometry. For a third case (test 50), the preswirl was simply defined consistent with "solid-body" rotation at rotor speed, as this was thought to be a fair approximation to the steady RANS solution. No-slip conditions were imposed on all solid boundaries. Rotor temperatures were specified from interpolation of thermocouple measurements for the corresponding experiment, and the shaft was assumed to be adiabatic.

The LES models used the classic Smagorinsky-Lilly subgrid scale model [23,24]. It is generally thought that the model parameter C_s needs to be tuned from case to case. According to Sagaut [25], the model constant C_s represents a relationship between the mixing length associated with the subgrid scales and filtering cut-off length and may vary from 0.10 to 0.33. The theoretical value of C_s is 0.18 for isotropic, homogeneous turbulence. For the inertial subrange of an isotropic, homogeneous turbulence, Lilly [24] further derived a value of 0.23 for the constant C_s . Therefore, the constant $C_s=0.23$ was adopted after some initial experimentation. In the near-wall regions, the FLUENT LES model is reported to assume linear profiles when y^+ values are small (as is the case

Table 1 Three test cases investigated in the study

Test case	Re_z	Re_ϕ	Ro	Gr	Bo
33	4.41×10^4	1.04×10^6	1.46	2.32×10^8	3.0
34	4.35×10^4	1.99×10^6	0.75	7.41×10^8	1.65
50	1.53×10^5	1.03×10^6	5.05	1.87×10^8	11.5
50* (CFD)	1.53×10^5	1.29×10^6	4.04	2.92×10^8	9.21

here).

To compare to the LES simulations, a RANS calculation using the same mesh was also conducted. The RANS turbulence model employed in this calculation was a two-layer RNG $k-\epsilon/k-l$ model [26,22]. This model is appropriate for the fine near-wall mesh spacing used here and had been found to perform similarly to other two-equation RANS models for this class of flow.

An implicit pressure correction solution algorithm was employed, with second-order temporal and spatial discretization for both the LES and RANS solutions. Central spatial differencing was used for the LES calculations, and upstream differencing for the RANS calculations. The time step was set to 1.0 ms (corresponding, for example, to 0.02 times the revolution time ($2\pi/\Omega$) for test 33). Solutions were obtained in the frame of reference rotating with the rotor; thus, relative velocities were smaller than they would have been in the absolute frame. Unsteady simulations were started from “part-converged” steady-state RANS solutions.

2.3 Flow Conditions. The three experimental test cases considered, tests 33, 34, and 50, are summarized in Table 1, with appropriate nondimensional parameters. The test cases were selected to cover most of the experimental range for the Buoyancy number with the rotational speeds being relatively low to minimize the mesh resolution requirements. The shroud Grashof number in Table 1 is defined as $Gr = \Omega^2 \beta \Delta T b (s/2)^3 / \nu^2$, where ΔT is the shroud to inlet air temperature difference, and ν denotes kinematic viscosity. The Rossby number Ro and the buoyancy number Bo are as previously defined. Axial and rotational Reynolds numbers ($Re_z = Wl/\nu$, and $Re_\phi = \Omega b^2/\nu$, where l is the inlet hydraulic diameter) are also given for reference. Note that in the test 50 calculations, the operating pressure was set to a higher value in the CFD than in the corresponding experiment, resulting in $Gr = 2.92 \times 10^8$ and $Bo = 9.21$, whereas $Re_z = 1.53 \times 10^5$ remained unchanged. Based on the results of other work and experimental evidence in this parameter range, it is judged that the higher operating pressure should not radically affect the Shroud Nusselt number predictions.

With regard to the computation speed, typically for a 120 deg model with a mesh of 4×10^6 cells, each time step takes ~ 4 min in CPU time when performed on a PC cluster with 20 2 GHz processors using low latency Myrinet networking. More powerful PC clusters and supercomputers will shorten the computation time.

3 Results

LES results are presented first, in the following three subsections. RANS and LES models are then compared in the section “Comparison of LES and RANS Models.”

3.1 Shroud Heat Transfer. In the experimental investigation, heat fluxes were estimated from thermocouple measurements at the midaxial position on the shroud. Shroud heat transfer was therefore selected for initial evaluation and comparison of results. Figure 3 shows a time history for the calculated circumferentially averaged midshroud heat flux. This is typical of the LES results. Considerable variations occur even after circumferential averaging, and this indicates the presence of large-scale rotating struc-

Fig. 3 Time history of shroud Nusselt number, LES, test 34, 120 deg model, $Gr = 7.41 \times 10^8$, $Bo = 1.65$

tures in the flow. (Henceforth, the term average is used to refer to a circumferential average.) Note that the shroud Nusselt number used here is defined as

$$Nu = \frac{q(s/2)}{k\Delta T} \quad (1)$$

where q is shroud heat flux.

Sensitivity to the angular extent of sector used in the model was examined for test 33, and results are shown in Fig. 4, which also presented the corresponding experimental data. The CFD results in Fig. 4 (and elsewhere when time-mean values are presented) are obtained from time averages of the monitored circumferentially averaged shroud heat flux data, typically based on 5000 time steps in the later part of the simulation, for which the time mean value had stabilized. (Henceforth, the term mean is used to denote the time average of the circumferential average monitors). For convection in a sealed rotating cavity, Sun et al. [18] showed that there was little difference in computed mean heat transfer between 45 deg sector and full 360 deg models. In the present results, some sensitivity is apparent, although there is little change in results obtained using the 120 and 360 deg models. The trend can be explained by the fact that the periodic conditions limit the appearance of larger flow structures, consequently reducing the heat transfer on the shroud. All further calculations reported here were conducted using the 120 deg sector model.

Underprediction of the shroud heat transfer is obvious in Fig. 4. Further comparisons are shown in Figs. 5 and 6. The experimental

Fig. 4 Shroud mean Nusselt number for test 33 with sector and full annulus models, $Gr = 2.32 \times 10^8$, $Bo = 3.0$

Fig. 5 Comparison of shroud mean Nusselt number between CFD and experiment, Nu versus Gr

data presented in these figures and the choice of parameters are from Long and Tucker [7] and Long et al. [19]. In Ref. [19], accuracy of the shroud heat transfer results is estimated as approximately $\pm 1\%$ to $\pm 6\%$, based on uncertainty in measured surface temperatures of ± 0.2 K. For test 50, the displacement between the test and CFD 50* on the abscissae is due to a higher value of pressure used in the calculations, as explained above. Noting the experimental trends of shroud Nusselt number at these Grashof and buoyancy numbers, this is unlikely to have a serious effect on the conclusions drawn here. Tests 33 and 34 are at relatively low buoyancy numbers. Test 50 is at a higher buoyancy number, where the axial throughflow is relatively stronger. The difference in heat transfer between the LES calculations and the measurements is between -25% and 6% . For tests 33 and 34, LES underpredicts the heat transfer, whereas for test 50 the difference may well be within experimental uncertainty. Experimental trends are followed by the model predictions reasonably well, as far as can be deduced from this limited number of tests. Also, considering that these results may be sensitive to the near-wall LES modeling and the assumed boundary conditions, the degree of agreement achieved is considered encouraging.

3.2 Mean Velocity and Turbulence Characteristics. Experimental velocity measurements from this test rig have been reported by Alexiou [27]. These show that the mean flow is almost in “solid-body” rotation in the outer part of the cavity when buoyancy number is small. This behavior is captured in the present LES calculations for tests 33 and 34, as shown in Fig. 7. Here, the time mean and circumferentially averaged relative tangential velocity at the midaxial position divided by the disk speed is plotted

Fig. 6 Comparison of shroud mean heat transfer between CFD and experiment, Nu versus Bo

Fig. 7 Comparison of mean relative tangential velocity profiles on the midaxial plane

against nondimensional radial position. A direct comparison of the tangential velocity profile with measurement for the test case test 50, for which $Bo=11.5$, is also given in this figure. It can be seen that the LES predictions are in good agreement with the measured data.

As previously noted, the predicted flow and heat transfer is essentially three-dimensional and unsteady, as observed in experiment. Although the existence of large-scale unsteady structures is widely accepted, the degree of turbulence that is present is less well established. Indeed, it has been hypothesized that regions of laminar flow may occur. Monitoring point histories of temperature and relative tangential velocity at the domain central point (CP) are shown in Figs. 8 and 9. Corresponding normalized turbulence energy spectra are shown in Figs. 10 and 11. The data shown are for test 34. Figures 10 and 11 include a $-5/3$ trend line to allow comparison to the expected trend for isotropic turbulence in the inertial subrange as observed in many experiments (see, for example, [28]). Clear irregular fluctuations can be seen in Figs. 8 and 9. The corresponding spectra in Figs. 10 and 11 show that the resolved turbulence energy spectra have regions where the slope is $-5/3$. Qualitatively, the spectra have the expected properties with

Fig. 8 Time history of temperature at the central point, LES, test 34, 120 deg model, $Gr=7.41 \times 10^8$, $Bo=1.65$

Fig. 9 Time history of relative tangential velocity at the domain central point, LES, test 34, 120 deg model, $Gr=7.41 \times 10^8$, $Bo=1.65$

foreshortening of the $-5/3$ region at high frequencies due to dissipation associated with the sub-grid-scale turbulent viscosity and (possibly) numerical diffusion.

3.3 Flow Structures. Large-scale flow structures are revealed by instantaneous, midaxial contour plots of static temperature ($T - T_{in}$)/ ΔT , relative tangential velocity $V_{t,rel}/r\Omega$ and radial veloc-

Fig. 10 Temperature fluctuation spectrum at the domain central point, LES, test 34, 120 deg model, $Gr=7.41 \times 10^8$, $Bo=1.65$

Fig. 11 Relative tangential velocity spectrum at the domain central point, LES, test 34, 120 deg model, $Gr=7.41 \times 10^8$, $Bo=1.65$

Fig. 12 Instantaneous temperature contours on the midaxial plane

ity $V_r/b\Omega$ in Figs. 12–14. The colder central throughflow can clearly be seen in Fig. 12. It may also be observed that hot and cold radial “arms” or “plumes” penetrate the cavity. Test 34, with the highest Grashof number and lowest buoyancy number, shows the strongest cold plume emanating from the throughflow jet. The flow structures for this case are possibly less ordered than at lower Grashof number. CFD 50*, corresponding to Test 50, with the highest buoyancy number, exhibits the sharpest definition of the jet boundary and has a generally cooler core flow within the cavity.

The instantaneous velocity contours in Figs. 13 and 14 show larger flow structures compared to those from the instantaneous temperature contours in Fig. 12. King et al. [17] noted even more pronounced differences between velocity and temperature contours in some of their 2D laminar-sealed cavity simulations, and

Fig. 13 Instantaneous relative tangential velocity contours on the midaxial plane

Fig. 14 Instantaneous radial velocity contours on the midaxial plane

attributed this to vacillation of the flow structure. Examination of transient behavior does confirm that the flow structure is continually changing.

3.4 Comparison of LES and RANS Models. For comparison to the LES results, an unsteady RANS calculation for test 50 was conducted using the RNG $k-\epsilon$ model with the 120 deg sector model and same mesh as for the LES calculation. As shown in Fig. 15, it was found that the RANS calculation gave much poorer predictions in terms of shroud heat transfer and tangential velocity profiles. The amplitude of shroud heat transfer fluctuations is also weaker in the RANS calculation. A further RANS calculation, using a coarser, 2×10^6 cell mesh for the full 360 deg domain, also gave similar results.

Comparison of the RANS prediction of turbulence viscosity to the LES sub-grid-scale turbulence viscosity at the cavity midaxial position shows the former to be around two orders of magnitude larger, as might be expected. This, of course, affects the mean flow field and its fluctuations, as confirmed by the center point tangential velocity results in Fig. 16. At first sight, the normalized turbulence energy spectra from the two calculations are perhaps surprisingly similar. Note, however, that it is difficult to identify clear trends from these data; longer simulation times are really required to reduce the statistical scatter and aliasing. This does of course raise interesting questions regarding the relationship between unsteady RANS calculations and LES as often practiced. In this case, the basic instability of a rotating flow with a negative radial density gradient makes some unsteadiness in the RANS solution inevitable, and the energy contained in large-scale fluctuations, must presumably be convected away or dissipated. The subgrid turbulence time scale in the LES calculation is given by L_{sgs}^2 / ν_{sgs} , where L_{sgs} and ν_{sgs} denote the subgrid length and the subgrid turbulence kinematic viscosity, respectively. The subgrid turbulence time scale is of order 10 ms at the cavity center point, which is consistent with dissipation of turbulence energy (and departure from the $-5/3$ slope in Fig. 16) for frequencies greater than 100 Hz. The turbulence time scale for the RANS solution at the cavity center is ~ 3 ms. This corresponds to a frequency of ~ 300 Hz, and, compared to the LES result, gives a less convincing representation of the large scale turbulence.

Differences between large-scale flow structures presented in the LES and RANS results are more obvious in the instantaneous, midaxial plane temperature and velocity contours. For the RANS

(15-a) Shroud Nusselt number, test 50

Fig. 15 Comparison of flow and heat transfer between LES, RANS, and measurement

calculations, these are given in Fig. 17, which should be compared to Figs. 12–14. In the case of the RANS results, the structures appear sharper and better defined.

4 Conclusions

Large eddy simulations of flow and heat transfer in a heated rotating cavity with a central axial throughflow have been obtained and compared to the available experimental data and unsteady RANS model calculations. The results of the LES models are very encouraging in that they were clearly in better agreement with the measured data than those obtained using a $k-\epsilon$ model. In the LES study, shroud heat transfer predictions were within 25% of measurements, and good agreement with tangential velocity measurements was demonstrated. However, the LES calculations are computationally demanding, to the extent that the present investigation was limited to just three experimental conditions. Further work is recommended, including investigation of more experimental conditions and careful consideration of the near-wall LES modeling 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.

Both LES and RANS models reveal large-scale flow structures. These are better defined for the RANS calculations, and there is a higher level of fluctuation in the LES. Temperature contours from the LES results show a finer structure than the corresponding velocity contours. For both RANS and LES models, results from a 120 deg sector model gave very similar mean flow quantities to those obtained from a full 360 deg model. For smaller sector arc lengths, the suppression of the larger structures by the periodic boundary condition does affect the mean flow and heat transfer.

Fig. 16 Comparison of RANS and LES velocity predictions, test 50

From a more general point of view, it may be observed that the current study adds to the growing evidence that LES can be of use in flows of industrial complexity and offer advantages in terms of accuracy over unsteady RANS models. A review of progress in this area is beyond the scope of this paper, but examples may be found in the open literature. Further turbomachinery applications of LES are given by Viswanathan and Tafti [29], who illustrate LES applications for turbine blade internal cooling, and Sarkar and Voke [30], who show LES calculations for flow over a low-pressure turbine blade.

Acknowledgment

Most of this research was conducted within the EU project on Internal Cooling Air Systems for Gas Turbines 2 (ICAS-GT2). Funding from the industrial partners and the EU is gratefully acknowledged, as well as the experimental data from the University of Sussex. Thanks also go to Rolls-Royce plc and the Engineering and Physical Sciences Research Council UK Applied Aerodynamics Consortium for high-power computing for supporting the 360 deg model LES solution. Technical discussions with and comments from all of the ICAS-GT2 partners are also gratefully acknowledged, with particular thanks to Dr. Nick Hills, Professor Alan Robins, Professor Peter Voke and all colleagues at the TF-SUTC at the University of Surrey, and staff at Fluent Europe.

Nomenclature

- a = inner radius of cavity
- b = outer radius of cavity
- Bo = buoyancy number $= Ro/(\beta \Delta T)^{0.5}$
- C_p = specific heat at constant pressure
- G = gap ratio of cavity $= s/b$
- Gr = shroud Grashof number $= \Omega^2 \beta \Delta T b (s/2)^3 / \nu^2$
- l = hydraulic diameter of annular inlet
- Nu = shroud Nusselt number $= q(s/2) / \kappa \Delta T$

(17-a) Temperature

(17-b) Relative tangential velocity

(17-c) Radial velocity

Fig. 17 Instantaneous temperature and velocity contours on the midaxial plane, test 50, RANS calculation

- Pr = Prandtl number $= \mu C_p / \kappa$
- q = shroud heat flux
- r = radius
- r_s = shaft radius
- Ra = Rayleigh number $= Pr Gr$
- Re_z = axial through flow Reynolds number $= \rho W l / \mu$
- Re_ϕ = rotational Reynolds number $= \rho \Omega b^2 / \mu$
- Ro = Rossby number $= W / a \Omega$
- s = cavity width
- T = temperature
- T_{in} = cooling air temperature at inlet
- T_{sh} = shroud wall temperature
- W = bulk velocity
- y^+ = normalized wall distance $= \rho u_\tau y / \mu$

Greek Symbols

- β = coefficient of thermal expansion
- ΔT = temperature difference between shroud and cooling air
- κ = thermal conductivity
- μ = dynamic viscosity
- ν = kinematic viscosity $= \mu / \rho$
- ρ = density
- τ = stress tensor or wall friction stress
- Ω = angular velocity of rotor

References

- [1] Bohn, D., Deuker, E., Emunds, R., and Gorzelitz, V., 1995, "Experimental and

- Theoretical Investigations of Heat Transfer in Closed Gas-Filled Rotating Annuli," ASME J. Turbomach., **117**, pp. 175–183.
- [2] Farthing, P. R., Long, C. A., Owen, J. M., and Pincombe, J. R., 1992, "Rotating Cavity With Axial Throughflow of Cooling Air: Heat Transfer," ASME J. Turbomach., **114**, pp. 229–236.
 - [3] Kim, S. Y., Han, J. C., Morrison, G. L., and Elovic, E., 1993, "Influence of Surface Heating Condition on Local Heat Transfer in Enclosed Rotating Disks With Axial Throughflow," ASME Paper No. 93-GT-258.
 - [4] Bohn, D. E., Deutsch, G. N., Simon, B., and Burkhardt, C., 2000, "Flow Visualisation in a Rotating Cavity With Axial Throughflow," ASME Paper No. 2000-GT-280.
 - [5] Burkhardt, C., Mayer, A., and Reile, E., 1993, "Transient Thermal Behavior of a Compressor Rotor With Axial Cooling Air Flow and Co-rotating or Contra-rotating Shaft," AGARD CP527, pp. 21.1–21.9.
 - [6] Alexiou, A., Hills, N. J., Long, C. A., Turner, A. B., and Millward, J. A., 2000, "Heat Transfer in High-Pressure Compressor Gas Turbine Internal Air Systems: A Rotating Disc-Cone Cavity With Axial Throughflow," Exp. Heat Transfer, **13**, pp. 299–328.
 - [7] Long, C. A., and Tucker, P. G., 1994, "Shroud Heat Transfer Measurements From a Rotating Cavity With an Axial Throughflow of Air," ASME J. Turbomach., **116**, pp. 525–534.
 - [8] Owen, J. M., and Powell, J., 2004, "Buoyancy-Induced Flow in a Heated Rotating Cavity," ASME Paper No. GT2004-53210.
 - [9] Lin, J. D., Paolillo, R., Johnson, B. V., and Daniels, W., 2004, "Flow Characteristics and Stability Analysis of Variable-Density Rotating Flows in Compressor Disk Cavities," ASME Paper No. GT2004-54279.
 - [10] Chew, J. W., 1985, "Computation of Convective Laminar Flow in Rotating Cavities," J. Fluid Mech., **153**, pp. 339–360.
 - [11] Bohn, D., Dibelius, G. H., Deuker, E., and Emunds, R., 1994, "Flow Pattern and Heat Transfer in a Closed Rotating Annulus," ASME J. Turbomach., **116**, pp. 542–547.
 - [12] Bohn, D., Emunds, R., Gorzelitz, V., and Kruger, U., 1996, "Experimental and Theoretical Investigations of Heat Transfer in Closed Gas-Filled Rotating Annuli. II," ASME J. Turbomach., **118**, pp. 11–19.
 - [13] Long, C. A., and Tucker, P. G., 1994, "Numerical Computation of Laminar Flow in a Heated Rotating Cavity With an Axial Throughflow of Air," Int. J. Numer. Methods Heat Fluid Flow, **4**, pp. 347–365.
 - [14] Iacovides, H., and Chew, J. W., 1993, "The Computation of Convective Heat Transfer in Rotating Cavities," Int. J. Heat Fluid Flow, **14**, pp. 146–154.
 - [15] 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," ASME Paper No. GT-2002-30479.
 - [16] Wong, L.-S., 2002, "Flow and Heat Transfer in Rotationally Induced Buoyancy Flow," D. Phil. thesis, University of Sussex.
 - [17] King, M. P., Wilson, M., and Owen, J. M., 2005, "Rayleigh-Benard Convection in Open and Closed Rotating Cavities," ASME Paper No. GT2005-68948.
 - [18] Sun, Z., Kilfoil, A., Chew, W. J., and Hills, N. J., 2004, "Numerical Simulation of Natural Convection in Stationary and Rotating Cavities," ASME Paper No. GT2004-53528.
 - [19] 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 No. LC1.
 - [20] Alexiou, A., 2000, "Flow and Heat Transfer in Gas Turbine H.P. Compressor Internal Air Systems," D.Phil. thesis, School of Engineering, University of Sussex.
 - [21] FLUENT Inc, <http://www.FLUENT.com>, Gambit User's Guide.
 - [22] FLUENT Inc, <http://www.FLUENT.com>, Fluent User's Guide.
 - [23] Smagorinsky, L., 1963, "General Circulation Experiments With the Primitive Subgrid-Scale Eddy Viscosity Model," Mon. Weather Rev., **91**, pp. 99–164.
 - [24] Lilly, D. K., 1966, "On the Application of the Eddy Viscosity Concept on the Inertial Subrange of Turbulence," NCAR Manuscript 123.
 - [25] Sagaut, P., 2002, *Large Eddy Simulation for Incompressible Flows*, Springer-Verlag, Heidelberg.
 - [26] Yakhot, A., Orszag, S. A., Yakhot, V., and Isreali, M., 1989, "Renormalization Group Formulation of Large-Eddy Simulation," J. Sci. Comput., **4**, pp. 139–158.
 - [27] Alexiou, A., 2002, "ICAS-GT2 Year 1 Progress Report. Work Package 2: Rotating Cavity Flow and Heat Transfer Experimental Investigation," Sussex research report.
 - [28] Pope, S. B., 2000, *Turbulent Flow*, Cambridge University Press, Cambridge, England.
 - [29] Viswanathan, A. K., and Tafti, D. K., 2005, "Large Eddy Simulation in a Duct With Rounded Skewed Ribs," ASME Paper No. GT2005-68117.
 - [30] Sarkar, S., and Voke, P. R., 2005, "Large-Eddy Simulation of Unsteady Pressure Over a LP Turbine Blade due to Interactions of Passing Wakes and Inflectional Boundary Layer," ASME Paper No. GT2005-6886.