Umesh Javiya

e-mail: u.javiya@surrey.ac.uk

John Chew¹

e-mail: j.chew@surrey.ac.uk

Nick Hills

e-mail: n.hills@surrey.ac.uk

Thermo-Fluid Systems UTC,
Faculty of Engineering and Physical Science,
University of Surrey,
Guildford, Surrey,
GU2 7XH, UK

Klaus Dullenkopf

Institut für Thermische
Strömungsmaschinen (ITS),
Karlsruhe Institute of Technology,
76128 Karlsruhe, Germany
e-mail: klaus.dullenkopf@skit.edu

Timothy Scanlon

PO Box 31,
Rolls-Royce Plc, Derby, UK
e-mail: timothy.scanlon@rolls-royce.com

Evaluation of Computational Fluid Dynamics and Coupled Fluid-Solid Modeling for a Direct Transfer Preswirl System

The prediction of the preswirl cooling air delivery and disk metal temperature are important for the cooling system performance and the rotor disk thermal stresses and life assessment. In this paper, standalone 3D steady and unsteady computation fluid dynamics (CFD), and coupled FE-CFD calculations are presented for prediction of these temperatures. CFD results are compared with previous measurements from a direct transfer preswirl test rig. The predicted cooling air temperatures agree well with the measurement, but the nozzle discharge coefficients are under predicted. Results from the coupled FE-CFD analyses are compared directly with thermocouple temperature measurements and with heat transfer coefficients on the rotor disk previously obtained from a rotor disk heat conduction solution. Considering the modeling limitations, the coupled approach predicted the solid metal temperatures well. Heat transfer coefficients on the rotor disk from CFD show some effect of the temperature variations on the heat transfer coefficients. Reasonable agreement is obtained with values deduced from the previous heat conduction solution. [DOI: 10.1115/1.4007752]

1 Introduction

In modern gas turbine engines, it is a common practice to deliver the cooling through stationary angled nozzles to the blade cooling receiver holes on the turbine disk. These preswirl nozzles swirl the air in the direction of the disk rotation, and this reduces the relative total temperature of the air entering the receiver holes at the expense of pressure loss across the nozzles. For optimal design of the preswirl system, understanding of the flow and heat transfer process is required.

Several experimental and numerical studies have been conducted on preswirl systems, generally using simplified geometries. Early experimental studies by Meierhofer and Franklin [1] confirmed the expected reduction of the relative total temperature. El-Oun and Owen [2] measured the relative total temperature in the receiver holes and correlated these with a simple model based on the Reynolds analogy. Wilson et al. [3] presented measurements and axisymmetric CFD calculations to predict the rotor disk heat transfer for a direct transfer preswirl system. Discrepancies in the heat transfer predictions were attributed to the axisymmetric assumption. Karabay et al. [4,5] and Pilbrow et al. [6] conducted experimental and numerical studies for a cover plate preswirl system. The flow structure observed was similar to a free vortex in the cover plate cavity. Experimental studies from the University of Sussex and associated theoretical studies were reported by Chew et al. [7,8] and Peng et al. [9]. Chew et al. [7] proposed a simple drag model to estimate the relative total temperature. Modified and extended versions of this model have been presented and used by Chew et al. [8] and Javiya et al. [10]. Heat transfer measurements on the rotor disk obtained with the thermochromic liquid crystal technique at the University of Bath have been published by Lock et al. [11] and Kakade et al. [12–15]. Corresponding numerical investigations are

given by Yan et al. [15], Farzaneh et al. [16], Lewis et al. [17] and Javiya et al. [18]. Varying degrees of agreement between measurements and CFD have been found. For example, Javiya et al. [18] found that disk heat transfer was well predicted for higher flow rates but there were significant differences at lower flow rates.

Significant research work on preswirl systems was undertaken within collaborative research programs funded by the European Commission; Internal Cooling Air Systems for Gas Turbines, ICAS-GT1 [19] and ICAS-GT2. Most of the preswirl system experiments were conducted at the University of Karlsruhe. Published experimental results for engine-representative preswirl geometries from these programs include measurements of discharge coefficient by Dittmann et al. [20], use of particle image velocimetry by Geis et al. [21], heat transfer measurements on the rotor disk [22] and evaluation of overall system performance in terms of measured delivery cooling air temperature by Geis et al. [23] and Bricaud et al. [24]. Computational analyses for these ICAS-GT programs have also been published by Benim et al. [25,26], Chew et al. [8], Ciampoli et al. [27] and Cagan et al. [28].

As described by Dixon et al. [29], it is conventional industrial practice to use standalone finite element (FE) analysis of components for prediction of metal temperatures and thermomechanical stresses. In this method, the convective heat transfer is modeled using correlations based on benchmark experiments, previous engine modeling experience, standalone CFD studies and/or engine test data. Direct use of CFD for convective boundary conditions is now becoming more popular. A review of such methods for turbomachinery thermal modeling is given by Sun et al. [30]. The coupled FE-CFD method used here has been selected for the advantages of flexibility in using separate solvers for the fluid and solid. Each solver has specialized numerical and modeling techniques. In coupled calculations, heat fluxes and temperatures are passed between the two solvers to ensure continuity of heat transfer across the common coupled walls. The method is readily applied to engine transient cycle thermal analysis as illustrated by Illingworth et al. [31], Sun et al. [32], Amirante et al. [33], and Ganine et al. [34].

¹Corresponding author.

Contributed by the International Gas Turbine Institute (IGTI) of ASME for publication in the JOURNAL OF ENGINEERING FOR GAS TURBINES AND POWER. Manuscript received June 24, 2012; final manuscript received August 5, 2012; published online April 18, 2013. Editor: Dilip R. Ballal.

The present study is motivated by the need for validation of CFD and coupled FE-CFD modeling for the prediction of preswirl cooling air delivery and disk temperatures. In the first part of this study CFD calculations are evaluated against measured discharge coefficients and cooling air temperatures. In the second part, FE-CFD coupled calculations are evaluated against the measured disk temperatures. The experimental details for the test rig and operating conditions considered are briefly described in Secs. 2 and 3. Standalone steady and unsteady CFD modeling is described in Sec. 4. Results for discharge coefficient and cooling air delivery temperature are presented in Sec. 5. Section 6 describes the coupled FE-CFD analysis and presents results for disk temperatures and local heat transfer. Finally, the main conclusions from this study are summarized in Sec. 7.

2 Experimental Details

Only the salient features of the experiments are given here, more details are given by Bricaud et al. [22,24]. A sectional view of the rig is shown in Fig. 1. Bricaud et al. [24] reported details for two different types of nozzles (simple straight-drilled and aero dynamic shape optimized nozzle design). However, here only simple straight drilled nozzles are considered. The preswirl plate (stator disk) contains preswirl nozzles inclined at 20 degrees to the circumferential direction and the number of nozzles can be varied by replacing the stator disk. The rotor disk contains axial receiver holes, which were open to ambient conditions. The rig offered the possibility of introducing seal flows. Different test rig builds could be made by changing the stator disk or rotor disk or by changing distance between the stator disk and rotor disk. Bricaud et al. [22] presented results for 15 different builds, which were made with different combinations of five rotors and six stators. For the zero seal flow condition considered here, the pressure drop across the labyrinth seal was adjusted to be close to zero.

The rig operated at up to 7200 rpm, resulting in a maximum tangential velocity for the receiver holes of 190 m/s and a maximum rotational Reynolds number of $Re_\phi \approx 2.6 \times 10^6$ (based on receiver holes radius, r_h and inlet condition). The overall pressure ratio across the preswirl rig was varied from 1.05 to 2.

The test rig was heavily instrumented with static pressure tapings on the static walls, total pressure tubes inside the chamber, static temperature thermocouples and total temperature probes inside the chamber, as well as rotating probes inside the receiver holes. For heat transfer measurements, the rotor disk temperatures were measured using 49 thermocouples. A telemetry unit was used to transfer rotating probe measurements. Temperature gradient across the rotor disk was created by blowing hot air at 393 K onto the downstream side of the disk as shown in Fig. 1. The recovery factor of the thermocouples on the rotor disk and inside the receiver holes was over 0.97. Hence, the temperature measure-

Fig. 1 Schematic of Karlsruhe preswirl rig (Bricaud et al. [24])

Table 1 Karlsruhe rig geometry data (Bricaud et al. [24])

	Configuration 1	Configuration 2
N_p	23	23
N_h	48	36
$r_p = r_h$	0.22 m	0.22
d_p	7 mm	7 mm
d_h	8.4 mm	10 mm
l_h/d_h	4	3.8
h	46.8 mm	46.8
s	10 mm	10 mm
Receiver holes edge	Chamfered	Sharp edge

ment errors were less than 1 K [24] for the rotor disk temperatures. However, the relative total temperature measurements highly depend on the swirl condition and flow inclination inside the receiver holes. Hence relative total temperature accuracy was estimated as 2.5 K at ~ 7000 rpm rotor disk speed.

3 Operating Conditions

Two preswirl configurations were selected for the present study and geometrical parameters for both configurations are given in Table 1. The stator with straight drilled nozzles and the preswirl chamber width were the same for both configurations. The major difference between two configurations was the rotor disk geometry. In configuration 1 the rotor disk had 48 8.4 mm diameter chamfered receiver holes, and configuration 2 had 36 10 mm diameter sharp edged receiver holes. According to the work of Cagan et al. [29], unsteady calculations, chamfered receiver holes have better aerodynamic performance than the sharp edged holes.

Five operating conditions were chosen, as given in Table 2. Following practice used in the experimental study, Re_ϕ and λ_T were calculated based on receiver hole radius r_h and preswirl chamber inlet (nozzle exit) conditions. In this study, Re_ϕ and λ_T values are higher than the work of Javiya et al. [18]. It should be noted that pressure ratio from inlet to outlet (P_{in}/P_{out}) was fixed in the unsteady computations. The mass flow rates given by the unsteady solutions were used as the boundary conditions in the steady computations.

Configuration 1 with case-1 and 2 operating conditions was used to evaluate the preswirl system performance using 3D steady and unsteady CFD calculations. Configuration 2 with operating conditions case 3, 4 and 5 was used for the disk temperature calculations with the coupled FE-CFD approach.

4 CFD Modeling

CFD simulations were carried out with second order discretization accuracy with the Hydra CFD solver [35]. Hydra is a suite of linear, nonlinear and adjoint solvers, developed by Rolls-Royce and its partner universities. In this work, a density based nonlinear solver [36] was used. This uses block Jacobi precondition and the 5 stage Runge-Kutta scheme of Martinelli [37]. Convergence acceleration was achieved with an edge collapsing based multigrid algorithm [37]. Hydra uses Oplus (Oxford parallel libraries for unstructured solvers) for parallel computations as described by Hills [38].

In all calculations, rotational Reynolds number was large enough to assume turbulent flow. Hence, all the CFD calculations used the $k-\epsilon$ turbulence model. At no-slip boundaries, wall functions were used. The standard $k-\epsilon$ turbulence model showed reasonable agreement with the measurements for previous preswirl CFD analysis by Javiya et al. [18]. However, with Hydra it was noted that the wall functions produced higher heat transfer on the rotor disk near to the nozzle jet impingement region. Hence, this limitation may be expected in the present calculations. Better results might be obtained with the use of a resolved wall treatment

Table 2 Operating conditions for the Karlsruhe rig analysis (Bricaud et al. [24])

Case	Pressure ratio (P_{in}/P_{out})	C_w	Re_ϕ	λ_T
Case-1	1.481	42,900	2.50×10^6	0.326
Case-2	2.018	69,300	2.74×10^6	0.491
Case-3	—	34,000	2.38×10^6	0.269
Case-4	—	32,600	1.35×10^6	0.407
Case-5	—	59,700	2.63×10^6	0.437

model, but this would require significantly more computation time.

4.1 Steady State Model. The steady state model was created with an axisymmetric slot representing the preswirl nozzles as in Javiya et al.'s study [18]. Configuration 1 is shown in Fig. 2(a). Periodicity of the model was assumed corresponding to the number of receiver holes. Mass flow rate, flow angle corresponding to the nozzle angle, 5% turbulence intensity and a 0.001 m turbulence length scale were specified at the inlet. Uniform static pressure was specified at the receiver hole outlet. A single rotating reference frame was used for the steady state calculations. The rotating walls are red and the stationary walls are blue in Fig. 1. The convergence criteria used follow those of an earlier study [10]. Overall mass balance errors were less than 1% of inlet mass flow. Angular momentum balance errors were less than 1% of the difference between flow of momentum into and out of the domain. Energy balance errors were less than 2% of the total change in mean flow enthalpy between inlet and exit.

Fig. 2 Steady and unsteady models for the Karlsruhe rig

In this paper, all CFD calculations presented used hexahedral meshes. Extensive mesh dependency tests were carried out for case 1 to ensure the solution was reasonably grid independent. A mesh with 0.42×10^6 nodes was used for most calculations, with dependency tests using up to 1.6×10^6 nodes. Near wall spacings were kept in the range of y^+ values from 30 to 100 over most of the walls.

4.2 Unsteady Model. For unsteady calculations, a detailed preswirl system was modeled, including nozzles and plena upstream of the nozzles and downstream of the receiver holes as shown in Fig. 2(b). In order to reduce periodic domain size, the number of nozzles was increased from 23 to 24. It is considered that this will not affect the conclusions presented here. This unsteady model is similar to that of Ciampoli et al. [27], with the addition of the inner cavity in some cases, as shown in the figure.

The calculations used stationary and rotating reference frames and a sliding plane approach as described by Ciampoli et al. [27]. No mesh dependency study was carried out for the unsteady calculations, but the mesh density was based on previous experience for similar flows. Careful attention was given to near-wall y^+ values for an initial “frozen rotor” steady simulation, which was used as the starting point for unsteady calculations. The y^+ values were found to remain in the range 30 to 100 over most of the walls throughout the unsteady calculation.

At the inlet, constant total pressure normal to the boundary and temperature were specified. At the outlet, constant static pressure was specified. It was assumed that placement of the outlet boundary did not affect the results inside the preswirl chamber. A constant rotating speed was specified for the rotating fluid zone. All walls in the domain were modeled with no-slip and adiabatic conditions.

The unsteady simulations were carried out using an implicit second order fixed time step size equivalent to 0.1 degree revolution of the rotor disk. The main interest of the unsteady simulations is the final periodic solution. The overall convergence of unsteady simulations was judged by monitoring several quantities in the unsteady domain. Moments on the stator and rotor walls inside the preswirl chamber were monitored. About 15 monitor locations were defined in the computation domain as shown in Fig. 3. Some of the probe locations correspond to experimental measurement positions. The convergence history for rotor wall moment is shown in Fig. 4. Convergence was judged to have been achieved after approximately 0.25 s, which is equivalent to about 30 revolutions of the rotor.

Computations were performed on a PC cluster with 128 nodes quad core, 2.6 GHz and Myrinet networking at the University of Surrey. The mesh sizes for the steady and unsteady model were in the range of 0.42 to 1.8×10^6 cells, respectively. The computation time was within a day for steady and within a week for unsteady calculations using 16 nodes with multiprocessors.

5 Results of Flow Analysis

5.1 Nozzle Discharge Coefficient. Following Dittmann et al. [20], discharge coefficients were calculated using time averaged values in the following equation,

$$C_D = \frac{\dot{m}_p}{\frac{A_N P_{in}}{R} \left(\frac{p_p}{P_{in}} \right)^{1/\gamma} \cdot \left[\frac{2C_p}{T_{o,in}} \left(1 - \left(\frac{p_p}{P_{in}} \right)^{\frac{\gamma-1}{\gamma}} \right) \right]^{0.5}} \quad (1)$$

where, P_{in} and $T_{o,in}$ were calculated using probe 13 and p_p was calculated using probes 14 and 16. $\dot{m}_p$ is calculated mass flow rate. A_N is nozzle cross section area, C_p is specific heat at constant pressure and R is the gas constant.

Calculated time averaged C_D values are compared with measurements in Fig. 5. The C_D values significantly under predict the

Fig. 3 Unsteady monitor probe locations

Fig. 4 Converge history for moment on the stator wall inside the chamber

Fig. 5 Time averaged C_D values

measurements (by about 12%). For a similar preswirl test case, better discharge coefficient results were obtained by Cagan et al. [28] (within 3% of experiment) in their CFD calculations. This is thought to be associated with different implementations of the wall functions in the two CFD codes and sensitivity of separated flows to turbulence modeling and numerical treatment.

Calculated time-averaged values for various performance parameters are given in Table 3. It can be seen that results, with and without the inner cavity, are very similar. However, the adiabatic wall temperatures were significantly higher near to the inner radius seal with the inner cavity included. Hence, including the inner seal might significantly affect the rotor temperature.

The results of the present calculations are significantly different from those of Ciampoli et al. [27]. Present simulations predict 8% higher P_{in}/P_p than Ciampoli et al.'s simulations, despite using a

Table 3 Time average values for the unsteady model

Operating conditions	With inner cavity	Without inner cavity	
	Case-1	Case-1	Case-2
P_{in}/P_p	1.32	1.32	1.694
$SR_{p,isen}$	1.235	1.235	1.672
SR_p	0.9	0.9	1.17
η	0.728	0.728	0.7
C_D	0.606	0.606	0.637
$\Delta T = T_{o,in} - T_{rel,h}$	8.41	8.91	20.38

similar mesh and boundary conditions. The differences are attributed to different numerical techniques, including different preconditioning methods, used in different versions of the Hydra code. The separated flow in the nozzle appears to be sensitive to these changes. It should also be noted that the accuracy of the $k-\epsilon$ turbulence model for separated flows is also questionable.

5.2 Swirl Velocity. Figure 6(a) compares calculated circumferential average core swirl velocity from the steady and unsteady calculations without the inner cavity for two operating conditions. The unsteady results are time-averaged. The swirl velocity is significantly different between the steady and unsteady simulations at the nozzle radius. This is because the discrete nozzles create circumferential variation of swirl velocity, and this strongly interacts with the rotating receiver holes at the same radius. This effect is absent in the steady state calculations because of the axisymmetric slot representation of the nozzle. In previous studies, where nozzles were at a lower radius than receiver holes and flow rates were lower than considered here, [18] these effects were small. The unsteady simulations show a local minima near to the nozzle radius ($r/r_o = 0.878$) because of the nozzle jet and cross flow interaction. This can also be observed in the instantaneous swirl velocity contour plots in Fig. 6(b) for the unsteady solution (case-1). The contour plots are shown in three planes cut through the nozzle entry, nozzle exit and at $z/s = 0.5$ inside the preswirl chamber. Flow through the nozzle creates two counter rotating vortex structures and creates local minima in the wake region of the nozzle jet flow. This phenomenon can be observed with path lines viewed through the nozzles in Fig. 6(b). This phenomenon is commonly observed for jets in the cross flow, e.g., Margason [39]. This leads to the swirl velocity minima for the unsteady calculations.

5.3 Preswirl Delivery Temperature. The preswirl cooling air delivery temperature at the receiver hole outlet ($T_{rel,h}$) was calculated in terms of temperature drop, $\Delta T = T_{o,in} - T_{rel,h}$. This is plotted against isentropic swirl ratio ($SR_{isen,p}$) at the nozzle exit in Fig. 7. The steady and unsteady results are shown for models without

(a) Circumferential averaged in the core region ($z/s=0.5$)

(b) Contour plots and path lines

Fig. 6 Swirl velocity inside the preswirl chamber

the lower cavity. The black thick line represents the ideal preswirl performance calculated from the following equation.

$$\Delta T = \frac{\Omega^2 r_h^2}{0.5 C_p} (2SR_{p,isen} - 1) \quad (2)$$

Figure 7 also includes the work of Bricaud et al. [22] and their experimental data and modified simple drag model prediction given by Chew et al. [8]. The steady state CFD and simple drag

Fig. 7 Total temperature drop for the Karlsruhe rig at ~7000 rpm

models do not include the nozzle; hence $SR_{p,isen}$ was obtained by scaling the mass averaged SR_p at the inlet slot by nozzle effectiveness (η), which was calculated from the unsteady simulations using the following equation:

$$SR_{p,isen} = SR_p / \eta \quad (3)$$

Surprisingly, the simple drag model gives the best correlation with experimental data, despite exclusion of the heat transfer. The present unsteady simulations predicted very similar temperature drops to Ciampoli's [27] unsteady calculations, but predicted higher swirl velocity. Hence, the predicted temperature in Fig. 7 has moved horizontally to the right and gives better agreement with the experimental data. The steady state simulations predict similar temperatures to the unsteady simulations, despite significant unsteadiness observed for flow inside the preswirl chamber and receiver holes. However the calculated $SR_{p,isen}$ value is higher for the steady calculation and this gives a better correlation with the experimental data. These differences arise from difficulties in modeling the separated flow in the nozzles. It is expected that more consistent results and better performance would be obtained for aerodynamically designed nozzles as used in other investigations [10]. The unsteady calculations show complex flow features associated with interaction of the preswirl jet with the swirling cross flow which are absent in the steady calculations. However these effects did not significantly affect the cooling air delivery temperature. Hence, the steady state calculations give useful results with less computation effort than the unsteady calculations.

Differences between CFD calculations and experiments might be explained by heat transfer process which were absent in these adiabatic calculations. For a similar preswirl system, Cagan et al.'s [28] unsteady calculations predicted coolant delivery temperatures up to 3 K lower temperature than measured. Cagan et al. stated that the heat transfer from the inner and outer radii seals might have some effect. In this calculation including the inner seal gave only 0.5 K decrease in cooling air delivery temperature. Outer labyrinth seal heat generation could increase air temperature in the order of Q/mC_p [10]. Here, Q is the frictional heat generation in the outer seal and it can be estimated from the labyrinth seal correlation given in Sec. 6.2 below. For the range of operating condition considered in this paper $\Delta T = T_{o,in} - T_{rel,h}$ could be decreased by up to 2 K. However, some heat may be transferred through the rotor back face to atmosphere and to static components. Hence, allowing for heat transfer, ΔT will decrease less than 2 K in Fig. 7. This would bring CFD results closer agreement to the measurements. Other factors could include inadequate modeling of the turbulent nozzle flow and the jet in the cross flow. The turbulence modeling limitation could be overcome with the use of higher order turbulence models, e.g., large eddy simulations (LES). Considering all these limitations, overall cooling air delivery temperature predictions are considered satisfactory with both steady and unsteady calculations.

6 Disk Temperature Predictions

In the work of Bricaud et al. [24], several thermocouples were used to measure the local rotor disk temperatures for configuration 2. To investigate the heat transfer behavior with direct prediction of these temperatures a steady state FE-CFD coupled model was used, including the inner cavity in the CFD model. The coupling approach is similar to that described by Illingworth et al. [31] and Sun et al. [30,32]. Only the model details and results are presented here.

6.1 Model Description. The FE-CFD simulations were carried out with the Hydra CFD code and Rolls-Royce's in house FE code SC03. Basic methodology and principles of the SC03 code are given by Armstrong and Edmunds [40]. The FE solid and CFD fluid domains are shown in Fig. 8. The fluid domain included

Fig. 8 Fluid and solid domains for the coupled model

the inner preswirl cavity and an axisymmetric slot representing the preswirl nozzles. The following boundary conditions were applied for the coupled model:

- seal flows were set to zero, as in the experiments
- the measured temperature profile was imposed on the downstream rotor disk face
- the stator walls (blue in Fig. 8) were assumed adiabatic
- for the coupled walls (red in Fig. 8), continuity of heat flux and temperature was imposed for the FE and CFD solutions
- the labyrinth seal at the outer radius of the rotor disk was modeled with a constant heat flux boundary. The heat flux was calculated based on a simple model, which will be described in Sec. 6.2.

The grid dependency of the complete coupled FE-CFD simulations was not checked. However, the grid dependency of the individual fluid and solid domains was considered. For the fluid domain near wall grid spacing was kept very similar to the previous steady calculations. For the solid domain grid dependency was checked using different grid sizes in standalone steady solutions with representative temperature boundary conditions. Results indicated that the grid used was sufficient to avoid significant errors in the heat conduction solution.

To compare with measurements, a steady state solution for disk temperatures was required. This was achieved by running the model in transient mode until it had stabilized at the required condition. At each time step, heat flux and temperature on the coupled wall were iteratively exchanged between FE and CFD models. Time stepping accuracy was specified so as to achieve a temperature accuracy of 0.5 K.

6.2 Labyrinth Seal Modeling. In the experiments, pressure across the outer radius labyrinth seal was balanced to eliminate leakage. A simple model was developed to estimate windage heating generated by the seal. This was based on the overall energy balance, and was verified with using CFD calculations.

With no sealing flow, the heat transfer through the labyrinth walls is equal to the frictional heat generated by the rotating labyrinth seal walls, and the moment on the rotor will be equal but opposite to that on the stator. Assuming fully mixed flow conditions and adiabatic stator walls, angular moment and energy balances give the following expression for heat flux through the rotor,

Fig. 9 CFD model and, estimated, and calculated heat flux for the labyrinth seal

$$\dot{q}_{\text{laby}} = -\frac{c}{2} \left(\frac{1}{1 + \left(\frac{A_{\text{laby}}}{A_{\text{stator}}} \right)^{1/2}} \right)^2 \rho \Omega^3 r_o^3 \quad (4)$$

where A_{laby} and A_{stator} denote surface areas for the whole labyrinth seal and outer stator wall. The friction factor c , was estimated from the free disk correlation as $c = 0.0534 \text{ Re}_\phi^{-0.2}$, following Chew et al. [7].

This correlation was verified using a separate axisymmetric labyrinth seal CFD model, as shown in Fig. 9(a). The CFD calculations employed second order accuracy for spatial discretization with the two layer $k-\varepsilon/k-l$ turbulence model in a commercial CFD package, Fluent. Near wall mesh spacing was defined such that y^+ values were well below 1 on all walls. Heat flux values were calculated assuming uniform temperature on the rotating wall and an adiabatic stationary wall. It can be seen that the estimated values match very well with the CFD calculations for the range operating condition considered.

For coupled calculations, overall computation time was about six days. Approximately, 99% of time was consumed by CFD in all calculations.

6.3 Rotor Disk Temperatures. Figure 10 shows the calculated rotor disk temperature contour plots for case-3, 4, and 5 operating conditions. The figure includes comparison with thermocouple measurements in terms of temperature difference ($\Delta T = T_w - T_{\text{rel, in}}$). In this view the disk rotates in the clockwise direction. As shown by the comparison of measurements and predictions on the right of the figure, disk temperatures in the preswirl chamber away from the receiver hole are generally well predicted. Temperatures at locations 12 to 16 are predicted within 3 K in all three cases. For locations 17 and 18, temperature differences might be explained by the neglect of any stator heat transfer and approximate modeling of the labyrinth seal windage heating. Inner cavity temperatures (locations 8 and 9) are significantly over predicted. This may be due to the neglect of the stator heat transfer, with some uncertainty regarding the significance of this in the experiments. Over prediction of temperatures at locations 10 and 11 may be associated with the (larger) over prediction in the inner cavity.

The rotor disk temperatures near to the receiver holes (shown on the left of Fig. 10) are reasonably well captured for case-3 and 4

Fig. 10 Rotor disk temperatures

but under predicted for case-5. At these locations, the maximum temperature difference between calculated and measured temperature difference is 5 K. The flow physics near the receiver holes is complex and some discrepancies due to turbulence modeling inaccuracies might be expected, as discussed by Javiya et al. [18].

Fig. 11 Radial variation of circumferential averaged Nusselt number

6.4 Rotor Heat Transfer. Heat transfer coefficients on the rotor disk were calculated using two approaches. First, from the coupled FE-CFD calculations which account for the temperature variation on the disk. Second, from standalone CFD calculations with uniform disk temperature. The disk heat transfer is presented in terms of Nusselt number using the following equations:

$$Nu = \frac{hr}{k} \quad h = \frac{\dot{q}_w}{T_w - T_{w,ad}} \quad (5)$$

$$T_{w,ad} = T_{o,in} - \frac{v_{\phi,p}^2}{2C_p} + Pr^{1/3} \frac{\Omega^2 r^2}{2C_p} \left(1 - \frac{v_{\phi,p}}{\Omega r}\right)^2 \quad (6)$$

where, $v_{\phi,p}$ is swirl velocity, which was calculated as the mass averaged value at the inlet slot and $T_{o,in}$ was calculated as average

Fig. 12 Comparison of Nusselt numbers from the coupled calculations

total temperature at three radial locations at $r/r_o = 0.851$, 0.915 , and 0.974 at the midsection of the cavity ($x/s = 0.5$).

Calculated circumferentially-averaged Nusselt numbers from uniform temperature and coupled solutions are plotted against radius in Fig. 11. Bricaud et al.'s [22] values, based on their conduction solution, are also shown. The peak at the radius $r/r_o = 0.878$ is due to inlet jet impingement on the rotor disk. Some differences due to the effect of surface temperature distribution can be observed between the coupled and the uniform temperature calculations. Away from the impingement region, the work of Bricaud et al. shows good agreement with the CFD for case-3 and 4 but are significantly lower for case-5. In the impingement region the coupled calculations predict higher heat transfer than Bricaud et al.'s [22] conduction solution for case 3 and 5, and lower for the case 4. Note that the disk surface temperatures in Bricaud et al.'s analysis, which were specified from the measurements, will also differ from those calculated in the coupled analysis.

Comparison of the coupled FE-CFD results for the three conditions considered is shown in Fig. 12. For case-4 and 5, the λ_T flow parameter (~ 0.4) and inlet swirl conditions are similar, and Re_ϕ is sufficiently high to indicate the flow is well in the turbulent range. Hence the Nusselt number may be expected to scale as $Nu \propto Re_\phi^{-0.8}$. This scaling can be observed in Fig. 12 over most of the rotor disk for case-4 and 5 where $\lambda_T \sim 0.4$. Divergence occurs at the nozzle radius, where jet impingement will dominate. For case-3 λ_T and the inlet swirl fraction differ from values for case-4 and case-5, so different values of $Nu \times Re_\phi^{-0.8}$ are expected.

7 Conclusions

CFD modeling has been compared with experimental data for a preswirled cooling air delivery system. Unsteady simulations showed significant unsteadiness inside the preswirl chamber especially at the nozzle radius because of the interaction of the discrete nozzle jet flow with the rotating receiver holes. However, unsteadiness inside the preswirl chamber did not significantly affect the cooling air delivery temperatures. Good agreement with measurements was obtained for the cooling delivery temperature but the nozzle discharge coefficients were significantly under predicted. This is attributed to sensitivity of the separated flow in the present nozzles to turbulence modeling and numerical treatment. Use of aerodynamically designed nozzles has the advantage of improving system predictability as well as performance. The inclusion of the inner cavity in the model did not significantly affect the coolant delivery temperature but affected the disk temperature significantly.

A coupled FE-CFD capability has been demonstrated and evaluated against measurements for prediction of disk temperatures. The coupled calculations gave satisfactory agreement with measured disk temperatures within the preswirl chamber. Some differ-

ences with measurements were observed near the inner and outer seals. These effects might be associated with neglect of stator wall heat transfer.

Disk heat transfer was calculated from the couple calculation and with uniform wall temperature. Some effect of surface temperature distribution was observed, especially in the jet impingement region. Comparison of disk heat transfer with previous estimates from a heat conduction solution showed good agreement over most of the rotor at low flow rates, but some differences at higher flow rate. For similar $\lambda_T \sim 0.4$, Nu the rotor disk wall showed $Nu \propto Re_\phi^{0.8}$ scaling except the nozzle jet impingement region.

Considering the coupled model boundary assumptions and near wall turbulence modeling limitations, the disk temperature predictions are considered satisfactory and encouraging for use of the method in industry.

Uncertainty associated with prediction of separated flow in the "straight drilled" preswirl nozzles would be avoided by the use of aerodynamically designed nozzles, as has been previously considered.

Acknowledgment

The authors are grateful to the EPSRC and Rolls-Royce for financial support of this work through a Dorothy Hodgkin Postgraduate Award, and to colleagues for helpful comments. The authors gratefully acknowledge the permission of Rolls-Royce plc to publish this paper.

Nomenclature

- A = cross section area, m^2
- c = angular moment coefficient ($=M/0.5\rho\Omega^2 b^5$)
- C_D = discharge coefficient
- C_p = specific heat capacity at constant pressure, J/kg K
- C_w = nondimensional mass flow rate ($=\dot{m}/\mu r_p$)
- D = moment on the stator walls, Nm
- d_h = receiver hole diameter, m
- d_p = nozzle diameter, m
- h = heat transfer coefficient, $W/m^2 K$
- k_a = thermal conductivity for air, (W/mK)
- k = turbulence kinetic energy, m^2/s^2
- l_h = receiver hole length, m
- $\dot{m}$ = mass flow rate, kg/s
- M = moment on the rotor walls, Nm
- N_h = number of receiver holes
- N_p = number of preswirl nozzles
- Nu = Nusselt number ($=hr/k_a$)
- p = static pressure, N/m^2
- P = total pressure, N/m^2
- Pr = Prandtl number, ($=C_p\mu/k_a$)
- q_w = wall heat flux, W/m^2
- r = radius, m
- r_h = receiver hole radial location, m
- r_o = rotor outer radius, m
- r_p = nozzle radial location, m
- R = gas constant (for air $R = 286.897$), J/kg K
- Re_ϕ = rotational Reynolds number, ($=\rho\Omega b^2/\mu$)
- s = preswirl chamber width, m
- SR = swirl fraction, ($=v_\phi/\Omega r_p$)
- SR_p = nozzle exit or chamber inlet swirl ratio, ($=v_{\phi,p}/\Omega r_p$)
- t = time, s
- T = static temperature, K
- T_o = total temperature, K
- T_{rel} = relative total temperature, K
- T_w = wall temperature, K
- $T_{w,ad}$ = adiabatic wall temperature, K
- ΔT = temperature difference, K
- u_τ = friction velocity, m/s
- v = velocity, m/s

v_ϕ = swirl/tangential velocity, m/s
 $v_{\phi,p}$ = swirl velocity at nozzle exit, m/s
 y = distance normal to the wall, m
 y^+ = nondimensional wall distance ($=\rho y u_\tau / \mu$)
 η = velocity coefficient, ($=v_{\phi,p} / v_{\phi,p,isen}$)
 ε = turbulence dissipation rate, m^2/s^3
 ρ = density of air, kg/m^3
 λ_T = mass flow-rate parameter ($=C_w \times \text{Re}_\phi^{-0.8}$)
 μ = molecular viscosity of air, Ns/m^2
 Ω = rotation speed of the rotor disk, rad/s

Subscripts

ad = adiabatic
 h = at receiver hole outlet
 in = at inlet
 $isen$ = isentropic
 $laby$ = for labyrinth seal
 mix = for fully mixed condition
 N = for nozzle
 o = total value in stationary frame
 p = preswirl
 r = at radial location r
 s = stator
 $static$ = for static walls
 w = rotor wall
 ϕ, r, x = circumferential, radial and axial coordinates

References

- [1] Meierhofer, B., and Franklin, C. J., 1981, "An Investigation of Pre-Swirl Cooling Airflow to a Turbine Disk by Measuring the Air Temperature in the Rotating Channels," ASME Paper No. 81-GT-132.
- [2] El-Oun, Z. B., and Owen, J. M., 1989, "Pre-Swirl Blade-Cooling Effectiveness in an Adiabatic Rotor-Stator System," ASME J. Turbomach., **111**, pp. 522–529.
- [3] Wilson, M., Pilbrow, R., and Owen, J. M., 1997, "Flow and Heat Transfer in a Pre-Swirl Rotor-Stator System," ASME J. Turbomach., **119**, pp. 364–373.
- [4] Karabay, H., Chen, J. X., Pilbrow, R., Wilson, M., and Owen, J. M., 1999, "Flow in a Cover-Plate Pre-Swirl Rotor-Stator System," ASME J. Turbomach., **121**, pp. 160–166, November 24–27.
- [5] Karabay, H., Pilbrow, R., Wilson, M., and Owen, J. M., 2000, "Performance of Pre-Swirl Rotating-Disk Systems," ASME J. Eng. Gas Turbines Power, **122**, pp. 442–450.
- [6] Pilbrow, R., Karabay, H., Wilson, M., and Owen, J. M., 1999, "Heat Transfer in a 'Cover-Plate' Preswirl Rotating-Disk System," ASME J. Turbomach., **121**(2), pp. 249–256.
- [7] Chew, J. W., Hills, N. J., Khalatov, S., Scanlon, T., and Turner, A. B., 2003, "Measurements and Analysis of Flow in a Pre-Swirl Cooling Air Delivery System," ASME Paper No. GT2003-38084.
- [8] Chew, J. W., Ciampoli, F., Hills, N. J., and Scanlon, T., 2005, "Pre-Swirl Cooling Air Delivery System Performance," ASME Paper No. GT2005-68323.
- [9] Peng, Z., New, P., Turner, A. B., Long, C. A., and Childs, P. R. N., 2007, "The Operating Characteristics of a High Radius Pre-Swirl Cooling System," J. Aerosp. Power, **22**, pp. 849–858, available at: http://caod.oriprobe.com/articles/12584471/Operating_characteristics_of_a_high_radius_pre_swirl_cooling_system.htm
- [10] Javiya, U., Chew, J. W., Hills, N., and Scanlon, T., "A Comparative Study of Cascade Vanes and Drilled Nozzle Design For Pre-Swirl," ASME Paper No. GT2011-46006.
- [11] Lock, G. D., Wilson, M., and Owen, J. M., 2005, "Influence of Fluid Dynamics on Heat Transfer in a Pre-Swirl Rotating Disk System," ASME J. Eng. Gas Turbines Power, **127**, pp. 791–797.
- [12] Kakade, V. U., Lock, G. D., Wilson, M., Owen, J. M., and Mayhew, J. E., 2009, "Effects of Radial Location of Nozzles on Heat Transfer in Pre-Swirl Cooling Systems," ASME J. Turbomach., **133**, p. 021023.
- [13] Kakade, V. U., Lock, G. D., Wilson, M., Owen, J. M., and Mayhew, J. E., 2009, "Accurate Heat Transfer Measurements Using Thermochromic Liquid Crystals. Part 1: Calibration and Characteristics of Crystals," Int. J. Heat Fluid Flow, **30**, pp. 939–949.
- [14] Kakade, V. U., Lock, G. D., Wilson, M., Owen, J. M., and Mayhew, J. E., 2009, "Accurate Heat Transfer Measurements Using Thermochromic Liquid Crystals Part 2: Application to Rotating Disk," Int. J. Heat Fluid Flow, **30**, pp. 950–959.
- [15] Yan, Y., Farzaneh-Gord, M., Lock, G., Wilson, M., and Owen, J. M., 2003, "Fluid Dynamics of a Preswirl Rotor-Stator System," ASME J. Turbomach., **125**, pp. 641–647.
- [16] Farzaneh-Gord, M., Wilson, M., and Owen, J. M., 2005, "Numerical and Theoretical Study of Flow and Heat Transfer in a Preswirl Rotor-Stator System," ASME Paper No. GT2005-68135.
- [17] Lewis, P., Wilson, M., Lock, G. D., and Owen, J. M., 2007, "Physical Interpretation of Flow and Heat Transfer in Preswirl Systems," ASME J. Eng. Gas Turbines Power, **129**, pp. 769–777.
- [18] Javiya, U., Chew, J. W., Hills, N. J., Lock, G. D., Wilson, M., and Zhou, L., 2010, "CFD Analysis of Flow and Heat Transfer in a Direct Transfer Pre-Swirl System," ASME J. Turbomach., **134**, p. 031017.
- [19] 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. GT2002-30479.
- [20] Dittmann, M., Geis, T., Schramm, V., Kim, S., and Wittig, S., 2002, "Discharge Coefficients of a Preswirl System in Secondary Air Systems," ASME J. Turbomach., **124**, pp. 119–124.
- [21] Geis, T., Rottenkolber, G., Dittmann, M., Richter, B., Dullenkopf, K., and Wittig, S., 2002, "Endoscopic PIV-Measurements in an Enclosed Rotor-Stator System With Pre-Swirl Cooling Air," 11th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal, July 8–11.
- [22] Bricaud, C., Dullenkopf, K., and Bauer, H. J., 2005, "Heat Transfer Measurements at the Rotor Disk of a Direct Transfer Preswirl System," 17th International Symposium on Airbreathing Engines, Munich, Germany, September 4–9, Paper No. ISABE-2005-1073.
- [23] Geis, T., Dittmann, M., and Dullenkopf, K., 2003, "Cooling Air Temperature Reduction in a Direct Transfer Pre-Swirl System," ASME J. Eng. Gas Turbines Power, **126**, pp. 809–815.
- [24] Bricaud, C., Dullenkopf, K., Bauer, H. J., and Geis, T., 2007, "Measurement and Analysis of Aerodynamic and Thermodynamic Losses in Preswirl System Arrangements," ASME Paper No. GT2007-27191.
- [25] Benim, A., Bonhoff, B., Bricaud, C., Brillert, D., and Cagan, M., 2005, "Computational Analysis of Flow and Heat Transfer in a Direct Transfer Pre-Swirl System," Sixth European Conference on Turbomachinery, Lille, France, March 7–11.
- [26] Benim, A., Brillert, D., and Cagan, M., 2004, "Investigation Into the Computational Analysis of Direct Transfer Preswirl Systems for Gas Turbine Cooling," ASME Paper No. GT2004-54151.
- [27] Ciampoli, F., Hills, N. J., Chew, J. W., and Scanlon, T., 2008, "Unsteady Numerical Simulation of the Flow in a Direct Transfer Preswirl System," ASME Paper No. GT2008-51198.
- [28] Cagan, M., Benim, A. C., and Gunes, D., 2009, "Computational Analysis of Gas Turbine Pre-Swirl System Operation Characteristics," WSEAS Trans. Fluid Mech., **4**(4), pp. 117–126.
- [29] Dixon, J. A., Verdicchio, J. A., Benito, D., Karl, A., and Tham, K. M., 2004, "Recent Developments in Gas Turbine Component Temperature Prediction Methods, Using Computational Fluid Dynamics and Optimization Tools, in Conjunction With More Conventional Finite Element Analysis Techniques," Proc. Inst. Mech. Eng., Part A **218**, pp. 241–255.
- [30] Sun, Z., Chew, J. W., and Hills, N. J., 2008, "Use of CFD for Thermal Coupling in Aero-Engine Internal Air Systems Applications," The 4th International Symposium on Fluid Machinery and Fluid Engineering, Beijing, China, November 24–27.
- [31] Illingworth, J. B., Hills, N. J., and Barnes, C. J., 2005, "3D Fluid-Solid Heat Transfer Coupling of An Aero-Engine Pre-Swirl System," ASME Paper No. GT2005-68939.
- [32] Sun, Z., Chew, J. W., Hills, N. J., Volkov, K. N., and Barnes, C. J., 2010, "Efficient Finite Element Analysis/Computational Fluid Dynamics Thermal Coupling for Engineering Applications," ASME J. Turbomach., **132**(3), p. 031016.
- [33] Amirante, D., Hills, N. J., and Barnes, C. J., 2010, "Thermo-Mechanical FEA/CFD Coupling of an Interstage Seal Cavity Using Torsional Spring Analogy," ASME Paper No. GT2010-22684.
- [34] Ganine, V., Javiya, U., Hills, N. J., and Chew, J. W., 2012, "Coupled Fluid-Structure Transient Analysis of a Gas Turbine Internal Air System With Multiple Cavities," ASME Paper No. GT2012-68989.
- [35] Lapworth, L., 2004, "Hydra-CFD: A Framework for Collaborative CFD Development," International Conference on Scientific and Engineering Computation (IC-SEC), Singapore, June 30–July 2, Vol. 30.
- [36] Martinelli, L., 1987, "Calculations of Viscous Flows With a Multigrid Methods," Ph.D. thesis, Princeton University, Princeton, NJ.
- [37] Moinier, P., Muller, J.-D., and Giles, M. B., 2002, "Edge-Based Multigrid and Preconditioning for Hybrid Grids," AIAA J., **40**(10), pp. 1954–1960.
- [38] Hills, N., 2007, "Achieving High Parallel Performance for an Unstructured Unsteady Turbomachinery CFD Code," Aeronaut. J., **111**, pp. 185–193, available at: <http://aerosociety.com/News/Publications/Aero-Journal/Online/496/Achieving-high-parallel-performance-for-an-unstructured-unsteady-turbomachinery-CFD-code>
- [39] Margason, R. J., 1993, "Fifty Years of Jet in Crossflow Research," In AGARD Symp. on a Jet in Cross Flow, Winchester, UK, Paper No. AGARD CP-534.
- [40] Armstrong, I., and Edmunds, T. M., 1989, "Fully Automatic Analysis in the Industrial Environment," Proceedings of the Second International Conference on Quality Assurance and Standards, NAFEMS, Stratford-upon-Avon, UK.