APPLICATION OF COMPUTATIONAL FLUID DYNAMICS
TO TURBINE DISC CAVITIES

G. P. Virr, J. W. Chew, and J. Coupland
Roll-Royce plc
Derby, United Kingdom

ABSTRACT

A CFD code for the prediction of flow and heat transfer in rotating turbine disc cavities is described and its capabilities demonstrated through comparison with available experimental data. Application of the method to configurations typically found in aeroengine gas turbines is illustrated and discussed.

The code employs boundary-fitted coordinates and uses the k-ε turbulence model with alternative near-wall treatments. The wall function approach and a one-equation near-wall model are compared and it is shown that there are particular limitations in the use of wall functions at low rotational Reynolds number. Validation of the code includes comparison with earlier CFD calculations and measurements of heat transfer, disc moment and fluid velocities.

It is concluded that, for this application CFD is a valuable design tool capable of predicting the flow at engine operating conditions thereby offering the potential for reduced engine testing through enhanced understanding of the physical processes.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>b</td>
<td>outer radius of cavity</td>
</tr>
<tr>
<td>Cm</td>
<td>moment coefficient (=2M/p0 b^2)</td>
</tr>
<tr>
<td>c_p</td>
<td>gas specific heat</td>
</tr>
<tr>
<td>C_w</td>
<td>cavity throughflow coefficient (=m/ub)</td>
</tr>
<tr>
<td>C_k</td>
<td>k-ε turbulence model constant (=0.09)</td>
</tr>
<tr>
<td>C_e1</td>
<td>k-ε turbulence model constant (=1.44)</td>
</tr>
<tr>
<td>C_e2</td>
<td>k-ε turbulence model constant (=1.92)</td>
</tr>
<tr>
<td>E</td>
<td>logarithmic wall function constant (=8.8)</td>
</tr>
<tr>
<td>g_{i,j}</td>
<td>grid metric tensor</td>
</tr>
<tr>
<td>G</td>
<td>turbulence energy production rate</td>
</tr>
<tr>
<td>H</td>
<td>stagnation enthalpy</td>
</tr>
<tr>
<td>k</td>
<td>turbulence energy</td>
</tr>
<tr>
<td>L</td>
<td>near-wall turbulence model length scale</td>
</tr>
<tr>
<td>m</td>
<td>cavity mass flow</td>
</tr>
<tr>
<td>M</td>
<td>disc moment</td>
</tr>
<tr>
<td>p</td>
<td>static pressure</td>
</tr>
<tr>
<td>P_j</td>
<td>Jayatillaka term in stagnation enthalpy</td>
</tr>
<tr>
<td>Q</td>
<td>total energy flux</td>
</tr>
<tr>
<td>r</td>
<td>radius</td>
</tr>
<tr>
<td>Re</td>
<td>cavity rotational Reynolds number (=p Db^2)</td>
</tr>
<tr>
<td>R_y</td>
<td>near wall Reynolds number based on turbulence energy</td>
</tr>
<tr>
<td>s</td>
<td>axial gap between discs</td>
</tr>
<tr>
<td>T</td>
<td>static temperature</td>
</tr>
<tr>
<td>u^{i,j}</td>
<td>Reynolds stress tensor</td>
</tr>
<tr>
<td>u^{i}</td>
<td>velocity vector</td>
</tr>
<tr>
<td>v</td>
<td>radial velocity</td>
</tr>
<tr>
<td>w</td>
<td>tangential velocity</td>
</tr>
<tr>
<td>y</td>
<td>distance to the wall</td>
</tr>
<tr>
<td>y'</td>
<td>near wall Reynolds number based on (\tau_{w}=(\rho \tau_{w})^{1/2}y/\mu)</td>
</tr>
<tr>
<td>\delta_{i,j}</td>
<td>Kroenecker delta tensor</td>
</tr>
</tbody>
</table>

Presented at the International Gas Turbine and Aeroengine Congress and Exposition
Cincinnati, Ohio May 24-27, 1993
This paper has been accepted for publication in the Transactions of the ASME
Discussion of it will be accepted at ASME Headquarters until September 30, 1993
\( \epsilon \) turbulence energy dissipation rate  
\( \kappa \) constant in near wall turbulence model  
\( \rho \) density  
\( \sigma \) Prandtl number \((=0.7)\)  
\( \sigma_k \) turbulent Prandtl number for \( k \) \((=1.0)\)  
\( \sigma_T \) turbulent Prandtl number for \( H \) \((=0.9)\)  
\( \sigma_e \) turbulent Prandtl number for \( e \) \((=1.3)\)  
\( \tau_{ij} \) total shear stress tensor  
\( \tau_w \) wall shear stress  
\( \Omega \) angular velocity of disc  
\( \mu \) viscosity  
\( \nu_T \) turbulent viscosity  

Subscripts:  
- \( \text{ad} \) adiabatic value  
- \( \text{in} \) inlet value  
- \( w \) wall value

**INTRODUCTION**

The potential benefits of using computational fluid dynamics (CFD) methods to calculate flow and heat transfer in gas turbine engine disc cavities were recognised some years ago. Although early calculations showed some considerable promise (see, for example, Gosman et al, 1976a) progress towards producing validated methods that can be applied with confidence has not been straightforward. Numerical difficulties, lack of suitable experimental measurements, turbulence model limitations and limited understanding of the flow mechanisms involved have all hampered progress. Nevertheless, it is now considered that certain classes of disc cavity flows can be predicted with some confidence, and in this paper a suitable CFD code is described with discussion and examples of validation and application of the method.

It is appropriate to mention some of the most relevant previous work, with particular reference to turbulence modelling, but for a fuller review the reader is referred to Chew (1980). Studies which employed the well known high turbulence Reynolds number \( k-\epsilon \) model with logarithmic wall functions to model the near-wall region are considered first. Gosman et al (1976a and 1976b) used this model with two different numerical solution methods and included comparisons with Hayley and Owen's (1970) torque measurements for a shrouded disc system with radial outflow of air. Some agreement with data was found with Gosman et al (1976b) showing particularly good agreement with measurement. However, at high Reynolds number, Gosman et al (1976a) had to adjust one of the turbulence model constants in order to match the experimental data. The reason for the difference in the results from these two numerical studies is not clear. Chew (1984) had difficulty in repeating some of Gosman et al's results and also found that this turbulence model performed badly when compared to Pincombe's (1983) velocity measurements for radial outflow between co-rotating discs. Morse (1988) subsequently confirmed the latter result. Comparing calculations with Daily and Nece's (1960) data for an enclosed rotating disc, Williams et al (1989) found that reasonable agreement could be obtained provided care was taken in specifying the near-wall mesh spacing. These workers recommended that the minimum rotor non-dimensional near-wall spacing \( y^+ \) should be in the appropriate range for applicability of the logarithmic law. Presumably, \( y^+ \) was not in the appropriate range at other points. In a recent paper near the (1992) compared CFD predictions with data from a model of a typical disc cavity and has concluded that the \( k-\epsilon \) model with wall functions "provides correct qualitative with unsatisfactory quantitative predictions".

Many workers have preferred turbulence models that avoid the use of wall functions by including a low Reynolds number model for the near-wall region. Various low Reynolds number versions of the \( k-\epsilon \) model have been tried with very mixed results (see for example, Chew 1984, Morse 1988, 1991a, 1991b, Morse and Ong 1992, and Roscoe et al 1988). From Morse's comparisons with Pincombe's (1983) and Morse and Ong's (1988) for co-rotating discs with outflow and Daily and Nece's (1960) and Daily et al's (1964) data for the enclosed disc with outflow it is clear that this model can give good results. The non-dimensional near-wall treatments and different numerical solution methods is less clear.

Mixing length models with van Driest (1956) damping near the wall have been used with some success. For example, Chew (1985a) and Chew and Vaughan (1988) found reasonable agreement of calculations with data for co-rotating discs with outflow and the enclosed rotating disc with outflow. The mixing length model used was based on one applied to boundary layer flow over rotating discs and cones by Koosinlin et al (1974). Seeking to combine the best features of the \( k-\epsilon \) and mixing length models was tried with Theofanopoulos (1991) adopted a zonal approach to turbulence modelling, with a near-wall mixing length treatment and a \( k-\epsilon \) eddy viscosity or algebraic stress model away from the walls. These authors again compared with data for the rotating cavity and enclosed disc cases. Further extensions of the zonal approach to include a one-equation model in the near-wall region and comparison with heat transfer data are reported by Iacovides and Chew (1992). From comparison with data and potential for further development, the \( k-\epsilon \) one-equation model was recommended, though some shortcomings in all the models tested were acknowledged.

In the present study use has been made of the \( k-\epsilon \) one-equation model and the \( k-\epsilon \) model using logarithmic wall functions. Brief descriptions of the models and the numerical methods used are given in the next section. Comparisons with some of the available data, and results of mesh dependency tests, are then given. Limitations of the wall function approach are also clarified in this section.
Application of the method to engine configurations is illustrated and discussed, and the main conclusions summarised.

**NUMERICAL METHOD AND TURBULENCE MODEL**

The CFD method used is a finite-volume Navier-Stokes solver using a body-fitted orthogonal curvilinear grid and grid oriented staggered velocity components. The same solver has been used for a number of other gas turbine internal flows, see Priddin and Coupland (1987) and Coupland, Fry and King (1991).

The solver uses the k-ε turbulence model combined with alternative near-wall treatments, hybrid differencing for the convection terms and the SIMPLE or AVPI (time dependent) pressure correction schemes (see Manners (1988)).

The orthogonal grids used by the solver are obtained using a Schwartz-Christoffel transformation based on that of Howell and Trefethen (1990). The procedure allows full control over the positioning of grid nodes on the solution domain boundary.

In general tensor notation, using repeated indices to imply summation and the "i,j" notation to denote the spatial covariant derivatives, the Reynolds averaged mass and momentum conservation equations are

\[(\rho U^i)_{,i} = 0\]  

\[(\rho U^i u^j)_{,j} = -g^{ij}(p + \frac{2}{3}\rho k) + \tau^{ij}\]  

where \[\tau^{ij} = (\mu + \nu) \left( g^{ij}k^{i,j} + g^{ik}k^{j,i} \right)\]  

The energy equation, assumed for a perfect gas and with neglect of some compressible effects in the dissipation term, is

\[(\rho U^i H)_{,i} = \left[ \left( \frac{U_T}{T} \right) g^{ij} \left( \frac{1}{T} \right) \right]_{,j} \]  

where the second term on the right hand side of this equation represents the frictional heating i.e. the work done by the spatial covariance of the dissipation rate using a length scale equal to \(K_T\). The turbulent viscosity in the k-ε region is obtained using

\[\mu_T = C_{\mu} k^2 / \epsilon\]  

with \(k\) and \(\epsilon\) being derived from their transport equations

\[(\rho U^i k)_{,i} = [(\mu + \rho) g^{ij} k^{i,j}]_{,j} + G - p\epsilon\]  

\[(\rho U^i \epsilon)_{,i} = [(\mu + \rho) g^{ij} \epsilon^{i,j}]_{,j} + C_{\epsilon} 2\rho k^2 - C_{\epsilon_2} \rho k^2\]  

The turbulent energy production rate is given by

\[G = -\nu_{ij} g_{jn} \mu_{ikn} (g^{kj} u^j_{,k} + g^{ik} u^j_{,k})\]  

and the Reynolds stresses are given by

\[\mu_{ij} = \frac{2}{3} \rho k \epsilon^{ij} - \mu_T (g^{kj} u^j_{,k} + g^{ik} u^j_{,k})\]  

The standard set of constants for the k-\(\epsilon\) model is used, see Launder and Spalding (1974).

The k-\(\epsilon\) region is coupled to the near wall region using either a near-wall low Reynolds number one-equation k-L model or wall functions. In the k-L model, see Iacovides and Chew (1992), the k transport equation is solved normally but with the dissipation rate and turbulent viscosity within the near wall zone set using

\[\epsilon = C_{\mu} 0.75 k^{1.5} / L \left( 1 - \exp(0.48R_y) \right)\]  

\[\mu_T = C_{\mu} 0.75 k^{1.5} / L \left( 1 - \exp(0.029R_y) \right)\]  

The near wall turbulent Reynolds number is defined by

\[R_y = C_{\mu} 0.75 k^{1.5} / \mu\]  

The length scale is defined to be proportional to the distance from the wall using

\[L = k y\]  

with \(k=0.43\). The near wall zone is defined using

\[R_y < 100\]  

With wall functions, semi-logarithmic profiles are used to describe the velocity and stagnation enthalpy profiles at the grid nodes adjacent to the wall and are of the form

\[\rho U_{\mu} \frac{1}{k^{1/2}} / \gamma_w = \frac{1}{k} \ln(ER_y)\]  

\[\rho (H_w - H) C_{\mu} \frac{1}{k^{1/2}} / \gamma_w = C_{\epsilon} \frac{1}{k} \ln(ER_y) + P_j\]  

where \(P_j\) is the Jayatillaka term (see Launder (1988)). Modifications are also made to the near wall production term in the turbulence energy equation to characterise the velocity gradient in terms of the semi-logarithmic profile and to fix the near wall dissipation rate using a length scale equal to \(k y\). The stagnation enthalpy wall function is obtained by analogy with the usual wall function for temperature. Whilst this is reasonable for conditions in which frictional effects are small, in general its validity is open to question.
RESULTS

Sealed Rotor-Stator

A suitable test case for comparing turbulence models and examining mesh sensitivity in rotating cavity flows is the sealed rotor-stator studied experimentally by Daily and Nece (1960). Calculations were performed for incompressible flow in a rectangular cross-section cavity in which one disc rotates while the other disc and the shroud are stationary. The conditions chosen correspond to an aspect ratio \( s/b = 0.0637 \) and rotational Reynolds number \( Re = 4.4 \times 10^6 \).

Calculations were performed with the wall function (W-F) and 2-layer (2-L) models using different numbers of mesh lines, and axial mesh expansion ratios away from the surfaces. A useful comparison of results is through the predicted values of the moment coefficient, \( C_m \), on the rotor, see Table 1. Other quantities listed in this table include the non-dimensional axial near-wall mesh spacings \( y/b \), and \( y^* \), at the radial position \( r/b = 0.95 \).

Table 1.
Results for the sealed rotor-stator example.

<table>
<thead>
<tr>
<th>Case</th>
<th>Turb. Model</th>
<th>Mesh Size</th>
<th>Exp. Rat.</th>
<th>Rotor ( y/b )</th>
<th>( R_y )</th>
<th>( y^* )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>W-F</td>
<td>41x41</td>
<td>1.0</td>
<td>-1.809E-3</td>
<td>1.6E-3</td>
<td>163</td>
</tr>
<tr>
<td>1.2</td>
<td>W-F</td>
<td>41x41</td>
<td>1.1</td>
<td>-1.885E-3</td>
<td>5.5E-4</td>
<td>59</td>
</tr>
<tr>
<td>1.3</td>
<td>W-F</td>
<td>41x41</td>
<td>1.2</td>
<td>-1.878E-3</td>
<td>1.7E-4</td>
<td>18</td>
</tr>
<tr>
<td>1.4</td>
<td>W-F</td>
<td>65x65</td>
<td>1.1</td>
<td>-1.942E-3</td>
<td>1.6E-4</td>
<td>18</td>
</tr>
<tr>
<td>1.5</td>
<td>2-L</td>
<td>65x65</td>
<td>1.2</td>
<td>-1.946E-3</td>
<td>1.9E-5</td>
<td>0.6</td>
</tr>
<tr>
<td>Daily &amp; Nece Correl.</td>
<td>W-F</td>
<td>65x65</td>
<td>1.1</td>
<td>-1.826E-3</td>
<td>1.6E-3</td>
<td></td>
</tr>
</tbody>
</table>

Only in case 1.1 was the predicted moment coefficient less than that obtained from the Daily and Nece correlation. In general, the level of agreement is considered good, all the results being within 8% of the correlation.

The wall function results of cases 1.1-1.3 clearly indicate a degree of sensitivity to the near-wall mesh spacing. The use of wall functions means that for a constant near-wall mesh spacing, it is not possible to obtain non-dimensional distances in the correct range for the application of logarithmic wall functions at each radial position. Since the maximum shear stress occurs towards the outer radius, it is important to obtain an appropriate near-wall mesh spacing in this region. The fact that case 1.1 gives the best agreement with Daily and Nece's correlation may be due to the better near-wall mesh spacing towards the inner part of the cavity. However, with other uncertainties such as measurement and correlation inaccuracies, it would be dangerous to read too much into this result.

Figure 1 shows axial profiles of the tangential and radial velocities at \( r/b = 0.765 \) for cases 1.1 & 1.3. The resolution of the boundary layers on the rotor and stator are clearly shown but the peak radial velocity is only adequately resolved for case 1.3. It has been found that for wall functions to give adequate results for rotating cavity flows it is necessary for the near-wall spacing to be sufficiently small to resolve the peak radial velocity as well as being in the preferred range for \( y^* \) of 30-100. An important implication of this is that wall functions are more likely to give valid results at higher rotational Reynolds number for which the peak radial velocity occurs at larger values of \( y^* \).

At high Reynolds numbers, placing the near-wall grid line in the log-law range automatically resolves the radial velocity profile. The agreement between calculated and measured velocities in Figure 1 is comparable to that found in earlier studies.

The zonal modelling approach adopted in the 2-layer model overcomes the difficulty noted above for wall functions but at the expense of extra mesh lines and the corresponding increased computational cost. The 2-layer model is used here with the maximum near-wall \( y^* \) of order 2 or less. Therefore, provided this is achieved at the outer radius it will also be achieved at all radial positions. The velocities predicted by the 2-layer model (case 1.5) agreed well with the wall function results of case 1.3.
Cobbled Disc with Bolt Cover

In order to demonstrate the geometric capabilities of the code, a cobbled disc with bolt cover was analysed, see Figures 2 & 3. The flow regime was radial outflow of air between co-rotating discs and the conditions considered here correspond to a mass flow parameter \( C_w = 1760 \) and rotational Reynolds number \( Re = 1.5 \times 10^5 \). This configuration has been studied experimentally by Farthing and Owen (1988) and Farthing (1988). In the experiments, air left the cavity through a series of discrete holes in the outer circumference, whereas in these 2D calculations, the fluid exits through a narrow slit. A comparison of CFD results with data for this case was reported by Lapworth and Chew (1992) who used a mixing length turbulence model.

Due to the low value of \( Re \) it was known that wall functions were not appropriate and so the 2-layer model was employed. The presence of the bolt cover on the right hand disc enabled two different meshing approaches to be adopted and compared. These consisted of either fitting an orthogonal mesh around the bolt cover or meshing through the bolt cover and subsequently blocking it out.

Figure 2 gives examples of the 2D boundary fitted orthogonal meshes used in the analysis. Note the highly distorted mesh with large variations in near-wall mesh spacing around the bolt cover shown in Figure 2(a). This type of mesh might be expected to give numerical errors due to the distortion and numerical diffusion errors due to the flow deviating markedly from the mesh line directions. The alternative approach shown in Figure 2(b), avoids all problems through mesh distortion and enables a uniform near-wall mesh spacing to be achieved, although numerical diffusion errors may still be present.

The results of a comparison of moment coefficient for different meshing approaches, near-wall mesh spacing etc., are presented in Table 2. For case 2.1, the fitted mesh resulted in excessive near-wall spacing in the corners where the bolt cover joined the right-hand disc. This was corrected in case 2.2 by moving the first 6 grid lines closer to the right hand disc. For case 2.3, the greater mesh expansion ratio resulted in sufficiently small near-wall mesh spacing. In case 2.6, the sensitivity of the near-wall spacing on the blockage was examined by moving mesh lines closer to the blockage.

Figure 3 gives contours of stream function for each meshing approach and shows excellent agreement for the overall mass flow distribution. Both plots clearly show the source region at inlet, the formation of the Ekman-type layers on the rotating surfaces, a core region in the centre of the cavity and a sink region at outlet. On the right hand disc, the interesting phenomenon that the Ekman layer does not separate from the corners of the bolt cover was predicted in both cases. This phenomenon was also observed in Farthing’s flow visualisation for laminar conditions.

The fact that the Ekman layer remains attached to the bolt cover explains why reasonable agreement is obtained for \( C_m \) for each meshing approach. Large diffusion errors are not present in the fitted mesh because the actual flow quite closely follows the mesh. The main problem with this approach is the large variation in near-wall mesh spacing. Comparison of the results of cases 2.1 & 2.2 shows the importance of obtaining a sufficiently small near-wall spacing at all mesh lines, since obtaining the required value in the corners of the bolt cover produced a 10% reduction in the predicted moment.

<table>
<thead>
<tr>
<th>Case</th>
<th>Meshing Approach</th>
<th>Mesh Size</th>
<th>Exp. Rat.</th>
<th>( C_m ) Left Disc</th>
<th>( C_m ) Right Disc</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Fit.Mesh</td>
<td>41x65</td>
<td>1.2</td>
<td>-5.512E-3</td>
<td>-6.630E-3</td>
</tr>
<tr>
<td>2.2</td>
<td>Fit.Mesh</td>
<td>41x65</td>
<td>1.2*</td>
<td>-5.527E-3</td>
<td>-5.960E-3</td>
</tr>
<tr>
<td>2.3</td>
<td>Fit.Mesh</td>
<td>41x65</td>
<td>1.3</td>
<td>-5.343E-3</td>
<td>-5.963E-3</td>
</tr>
<tr>
<td>2.4</td>
<td>Fit.Mesh</td>
<td>65x65</td>
<td>1.3</td>
<td>-5.311E-3</td>
<td>-5.880E-3</td>
</tr>
<tr>
<td>2.5</td>
<td>Blockages</td>
<td>55x65</td>
<td>1.2</td>
<td>-5.524E-3</td>
<td>-6.077E-3</td>
</tr>
<tr>
<td>2.6</td>
<td>Blockages</td>
<td>55x65</td>
<td>1.2+</td>
<td>-5.540E-3</td>
<td>-6.017E-3</td>
</tr>
<tr>
<td>2.7</td>
<td>No Bolt Cover</td>
<td>55x65</td>
<td>1.2</td>
<td>-5.593E-3</td>
<td>-5.959E-5</td>
</tr>
</tbody>
</table>

* Mesh as 2.1 but with 6 grid lines moved closer to RH disc.
+ Mesh as 2.5 but with mesh lines moved closer to blockage.

Downloaded From: https://proceedings.asmedigitalcollection.asme.org on 11/17/2018 Terms of Use: http://www.asme.org/about-asme/terms-of-use
use of blockages to model the bolt cover results in a larger number of mesh lines for the same near-wall spacing since all surfaces of the blockage require the same grid spacing as the discs. This approach also leads to large aspect ratio grid cells in the centre of the cavity which may lead to numerical difficulties.

Comparison of the results of cases 2.2 & 2.3 once again shows how significant the near-wall mesh spacing is in rotating cavity flows. The increased mesh expansion ratio, resulting in reduced near-wall mesh spacing, avoids the need to move mesh lines closer to the right-hand disc, but produced a reduction in moment on the left-hand disc. The results of case 2.4 with extra axial mesh lines confirmed the predicted moment coefficients thereby showing a degree of mesh independence.

The results of cases 2.5 and 2.6 were in close agreement with those of case 2.2, for which identical near-wall mesh spacings were used on the discs away from the bolt cover. The differences can be attributed entirely to the different meshing approaches for the bolt cover. Case 2.7 used the same mesh as case 2.6 but with no bolt cover. As identical mesh spacing was used on the disc surfaces, the slight difference of moment coefficient on each disc indicates that for this flow any errors associated with large aspect ratio cells in the centre of the cavity are small for this particular flow. Comparison of cases 2.6 and 2.7 indicates that the addition of the bolt cover gives a 7.5% increase in moment on the disc with the cover, and a slight reduction of about 1% on the opposing disc. However, further comparison with case 2.4 shows that the prediction of such small effects may be subject to numerical errors.

### Rotor-Stator Heat Transfer

Validation of the code for the prediction of local heat transfer was undertaken by comparing with the experimental results of Millward and Robinson (1989). This data is at typical engine operating conditions and has not previously been compared with CFD predictions. The case chosen was a rotor-stator with radial outflow. A diagram of the rig is given in Figure 4 which differs slightly, from that shown by Millward and Robinson, in terms of the pressure balanced seal and geometry of the stator at the outer radius.

The case analysed was a plane disc rotating at 13325rpm with a hot side test airflow of 0.2268kg/s at nominal temperature 471K and pressure 345kPa. The conditions correspond to Re=7E6 and Cw=36800 based on an outer radius b=0.233m. In the experiment, the hot side conditions were kept approximately constant. The mass flow rate of the cold side ventilation airflow was varied to alter the heat transfer rate through the rotor. The disc was instrumented with "plugmeters" and thermocouples to measure the heat flux through the disc and the hot side metal temperature at 3 radial positions.

For a given hot side test condition, experiments were carried out for various cold side mass flows, thus varying the thermal boundary conditions on the disc in the test section. Plots of disc temperature against heat flux were produced, as in Figure 5. By plotting lines through the data points, estimates of adiabatic disc temperature were obtained from values for the zero heat flux condition.

In order to reduce the sensitivity of the CFD results to the assumed inlet conditions, the solution domain was extended upstream of the region where the measurements were taken, as indicated by the cross-hatched lines in Figure 4. The geometry of the stator, in the
outer part of the cavity, and the inner region of the rotor, was also simplified slightly. Calculations were performed with both wall functions and 2-layer model and the results compared with measurements. Figure 6 gives details of the mesh used for the calculations, predicted streaklines, contours of stream function and tangential velocity for the wall function calculations. The 2-layer and wall function model calculations were in reasonable overall agreement as highlighted in Table 3.

The wall function approach predicted a higher moment on the rotor but, due to the limitations described earlier, it was not possible to obtain the correct near-wall spacing at the inner radii. The mesh used for the 2-layer model was created from the wall function mesh by placing extra mesh lines in the near-wall regions. In both cases, consistent results for the total air temperature rise were obtained by the separate calculations of work input to the air and solution of the energy equation with frictional heating source terms and adiabatic walls. Although care was taken to ensure appropriate near-wall mesh spacings, guided by experience gained from earlier test cases, exhaustive mesh-dependency tests were not undertaken for this example.

A comparison of the predicted adiabatic wall temperatures with measured values is given in Table 4. The quantity $\alpha^{1/3}(nr)^2/2C_p$, which is the expected frictional temperature rise for a free disc (see, eq. Chew 1985b), is also tabulated. The most notable feature of the results is that the measurements are somewhat lower than all the theoretical results, particularly at the lower radii. Some differences between the two turbulence models are also apparent, although this degree of uncertainty may be acceptable for engineering calculations. The techniques used to deduce the experimental adiabatic disc temperature, from measurements in which the disc was not truly adiabatic, might account for some differences with theory. Examination of measurements for other flow conditions showed adiabatic disc temperatures considerably closer to the free disc result, reinforcing confidence in the theory.

<table>
<thead>
<tr>
<th>Case</th>
<th>Turb. Model</th>
<th>Mesh Size</th>
<th>Cm</th>
<th>Total Air Temp. Rise (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>W-F</td>
<td>41x45</td>
<td>-3.354E-3</td>
<td>33.29</td>
</tr>
<tr>
<td>3.1</td>
<td>W-F</td>
<td>53x57</td>
<td>-3.209E-3</td>
<td>31.84</td>
</tr>
<tr>
<td>3.2</td>
<td>2-L</td>
<td>53x57</td>
<td>32.97</td>
<td>31.83</td>
</tr>
</tbody>
</table>

Table 4.
Results for adiabatic disc temperature.

<table>
<thead>
<tr>
<th>Plugmeter Radius</th>
<th>Experiment $T_{ad-Tin}$ (K)</th>
<th>Predicted $T_{ad-Tin}$ (K)</th>
<th>$\alpha^{1/3}(nr)^2/2C_p$</th>
</tr>
</thead>
<tbody>
<tr>
<td>r/b</td>
<td>W-F</td>
<td>2-L</td>
<td></td>
</tr>
<tr>
<td>0.681</td>
<td>14.0</td>
<td>17.4</td>
<td>22.7</td>
</tr>
<tr>
<td>0.809</td>
<td>22.0</td>
<td>27.7</td>
<td>34.2</td>
</tr>
<tr>
<td>0.937</td>
<td>37.0</td>
<td>38.3</td>
<td>46.1</td>
</tr>
</tbody>
</table>

Figure 5. Measured Disc Heat Flux & Temperature (Cw=36800, Re=7E6)

Figure 6. Rotor-Stator with Radial Outflow (Cw=36800, Re=7E6)
Heat transfer calculations were performed with both near-wall treatments using measured disc temperatures from the case with the maximum cold ventilation flow, i.e. maximum disc heat fluxes. Disc temperatures were only measured at 3 radial positions and so a temperature profile was derived from a spline fit to the measured values. This profile was used in the CFD model with fixed temperature boundary conditions applied on the disc, and with other surfaces specified as adiabatic. The flow field was assumed to remain unchanged with other surfaces specified as adiabatic. The approach appears to overpredict the heat flux from the air to the disc and work done on the air due to frictional drag on the disc are of similar magnitude. Thus, this is quite a sensitive test case.

Considering the above mentioned difficulties regarding the adiabatic wall temperature, a direct comparison of measured and predicted surface heat fluxes was preferred to use of Nusselt numbers. The model predictions are 22% below the experimental value, while at the other two points, agreement is within 1%. The discrepancies at the outer point may be associated with the extrapolation of temperature measurements in defining boundary conditions for the calculation. Any inaccuracies introduced in this manner may well be most severe towards the outer radius of the disc.

Differences between the wall function and 2-layer models for heat transfer prediction are considerably greater than the differences in moment coefficient. This may be associated with the stagnation enthalpy wall function which, as noted previously, is subject to some uncertainty. Although the wall function approach appears to overpredict the heat transfer levels, it may still prove to be useful in cases where the number of mesh points must be restricted. With improvements to the stagnation enthalpy wall function or for less sensitive conditions, closer agreement between the two models might be expected.

APPLICATIONS

An example where the methods described in this paper have been applied to a high bypass ratio gas turbine is given in Figure 8. This composition, from a number of separate calculations, shows streamline plots of the predicted flows in the cavities surrounding the low pressure turbine discs. The classes of disc flow calculated include co-rotating discs, rotor-stator systems and differentially rotating discs. In these calculations, wall functions were used and maximum take-off operating conditions were assumed. These CFD predictions have provided detailed information which would otherwise have been extremely difficult and costly to obtain by engine testing.

Use of wall functions enabled the overall mesh size to be kept to a minimum, but in these 2D calculations, it would have been equally possible to employ the 2-layer model without excessive increase in computing cost. For 3D calculations, the increased mesh size inherent in the 2-layer model is more likely to limit use of the code. The greater generality of the 2-layer model must be weighed against the more demanding mesh requirements for individual applications.

The demonstrated geometric capabilities of the code clearly enable a wide variety of engine cavities to be examined. This study, and earlier work, has shown that typical turbine cavities can be successfully calculated and important design information obtained. The validity of the method for some classes of disc flow is however unproven. A good example of this is the cavity between co-rotating discs with no imposed radial throughflow, as is often found in compressors. In these cavities, the convective flows are driven by buoyancy effects in the centripetal force field. Clearly, care is needed both in identifying which flows are suitable for this type of analysis and in obtaining accurate results.

CONCLUSIONS

The benefits of applying CFD methods to calculate flow and heat transfer in gas turbine engines are now being realised. Careful numerical testing and evaluation against experimental data enables predictions to be made with some confidence. Wider use of CFD is expected in the future with three dimensional flows and integration of CFD methods in the disc thermal analysis receiving more attention.

Two near-wall turbulence model treatments have been compared in this study. Conventional wall functions were found to give acceptable results for velocity and moment predictions, provided the Reynolds number was
sufficiently high, and care was taken in specifying the near-wall mesh spacing. For heat transfer, further consideration of the stagnation enthalpy wall function is perhaps needed. The one equation near-wall model has greater generality than wall functions with the disadvantage that it requires more mesh points. Note, however, that the one equation treatment requires less near-wall mesh points than the low Reynolds number $k-\varepsilon$ models used by several other workers.

Present results for the sealed rotor-stator and cobbled disc examples are consistent with expectations from earlier studies. Comparison with Millward and Robinson's (1989) heat transfer measurements for a rotor-stator system with radial outflow gives further confidence in the use of CFD. Although these experiments are not ideal for CFD validation, they are more representative of engine conditions than most of the available data.

ACKNOWLEDGEMENTS

The assistance of our colleagues John Millward and Peter Robinson, in supplying and interpreting their experimental data is gratefully acknowledged.

REFERENCES


