ABSTRACT

A blind test case for a compressor rotor (ROTOR 37) was organized by the ASME/IGTI at its 1994 meeting in order to assess the predictive capabilities of the turbomachinery CFD tools. The results from the different CFD codes showed a wide scatter which in part is due to the differences in the turbulence models that were used. In order to systematically isolate the capabilities and limitations of the turbulence models, ROTOR 37 flow is computed from the same numerical platform with three different turbulence models. These include: the Baldwin-Lomax model, the standard k-ε model, and an improved version of this k-ε model. The results from the three models are compared with the experiment. We find that with increasing model complexity the results move closer to the experiment. Several sensitivity studies are carried out to bracket the uncertainty in the computations. These include the effect of: wall boundary conditions for the turbulence models; numerical accuracy of the turbulence solver; and the effect of the inlet boundary condition for turbulence.

INTRODUCTION

Turbomachinery flows pose a major challenge for CFD since they are three dimensional and time dependent. In order to assess the predictive capabilities of turbomachinery CFD tools a blind test case for a rotor compressor (ROTOR 37) was organized by the 1994 ASME/IGTI held at the Hague (Wisler and Denton 1994). A total of 12 groups submitted their predictions for this case, which were then compared with the detailed experimental data. The results can be found in the above reference. The predictions showed a wide scatter when compared with the experimental data. There are probably several reasons for these differences, and it is beyond the scope of this paper to investigate all of these. However, one of these differences could be attributed to the deficiencies in the turbulence models but this was difficult to ascertain because different codes used the same turbulence model but gave different results. Obviously this is related to the different numerical features of the computations (type of grid, number of grid points, discretization schemes, etc.).

From the point of view of isolating the turbulence model performance, the need is to assess different turbulence models from the same numerical platform. If different turbulence models are implemented in a single CFD code and are computed on the same grid, then the differences in the results can be attributed only to the differences in the models. In this way the impact of different models on the prediction of turbomachinery flows can be assessed in a systematic manner. With this background the present study was initiated with the objective of assessing the performance of different turbulence models for turbomachinery flows. The VSTAGE code was selected as the numerical platform to achieve this objective. The predictions of the VSTAGE code, using the Baldwin-Lomax model, for the ROTOR 37 blind test case were presented at the 1994 ASME/IGTI meeting (Celestina 1994). In the present study we will assess the performance of a standard two equation k-ε turbulence model and a CMOTT (Center for Modeling of Turbulence and Transition at the Lewis Research Center) improvement to this model for the ROTOR 37 flow. There are several studies which deal with the use of turbulence models in computing turbomachinery flows. For example Turner and Jennions (1993) investigated the turb-
bulence modeling issues for transonic fans. Dalbert and Wiss (1995) and Chima (1996) consider the ROTOR 37 flow predictions using zero equation as well as two equation turbulence models.

**ROTOR 37 Background**

In order to orient the reader about the flow-field geometry we briefly describe ROTOR 37. NASA stage 37 was designed by NASA Lewis Research Center as a test compressor for a core compressor of an aircraft engine. For the purposes of code evaluation the rotor was tested in isolation, Suder (1994), and is referred to as ROTOR 37. The inlet relative Mach number of the rotor is 1.13 at the hub and 1.48 at the tip. It has an aspect ratio of 1.19 and a radius ratio of 0.70. Details of the aerodynamic design can be found in Reid and Moore (1978). Detailed aero probe data was taken upstream as well as downstream of the rotor. These locations are depicted in Fig. 1 and are referred to as station 1 and station 4. Detailed pitchwise surveys of the rotor wake were also taken at several span-wise positions just behind the rotor (station 3) as well as station 4 using LDV. These are depicted in Fig. 1. The experimental data for ROTOR 37 was released by the ASME Turbomachinery Committee after the deadline for the blind test case submissions. Further details about the experiment and facility can be found in Suder (1994).

**TURBULENCE MODELS**

The three turbulence models tested in the present study include the zero equation Baldwin-Lomax model, the two equation standard $k - \epsilon$ model, and the CMOTT improved $k - \epsilon$ model. As was mentioned earlier, the results from the VSTAGE code using the Baldwin-Lomax model were presented at the 1994 ASME/IGTI meeting and can be found in Celestina (1994). Since this model has already been described and tested in several earlier studies, (e.g. Adamczyk et al. 1990) it is not described here. The second turbulence model used in the present study is the standard $k - \epsilon$ model of Launder and Spalding (1974). This is a high Reynolds number model and, therefore, requires the use of wall functions. This model forms the basis of almost all of the other two equation models which exist in the literature. These studies have either extended the applicability of Launder Spalding model or have removed some of its deficiencies. Using tensor notation the model can be written as

$$\rho \frac{\partial \bar{u}_i}{\partial x_j} = \mu_T \left[ \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \right] - \rho \epsilon \delta_{ij} + \frac{2}{3} \rho \kappa \delta_{ij}$$

$$\mu_T = C_\mu \frac{k^2}{\epsilon}$$

$$U_j \frac{\partial \rho k}{\partial x_j} = \left( \frac{\mu_T}{\sigma_k} \frac{\partial \rho k}{\partial x_j} \right) - \rho \kappa \frac{\partial U_i}{\partial x_j} - \rho \epsilon$$

$$U_j \frac{\partial \rho \epsilon}{\partial x_j} = \left( \frac{\mu_T}{\sigma_\epsilon} \frac{\partial \rho \epsilon}{\partial x_j} \right) - C_{\epsilon 1} \rho \kappa \frac{\partial U_i}{\partial x_j} - C_{\epsilon 2} \rho k$$

The values of the model constants are: $\sigma_k = 1.0; \sigma_\epsilon = 1.3; C_{\epsilon 1} = 1.44; C_{\epsilon 2} = 1.92; \text{ and } C_\mu = 0.09$. This model will be referred to as the SKE model in this paper.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_{REL}$</td>
<td>relative Mach number</td>
</tr>
<tr>
<td>$P_{TOT}$</td>
<td>total pressure ratio</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>mean strain rate tensor, see eqn. (8)</td>
</tr>
<tr>
<td>$S_{i\phi}$</td>
<td>source term in $\phi$ equation ($\phi = k, \epsilon$)</td>
</tr>
<tr>
<td>$S_{\phi}$</td>
<td>positive part of source term $S_{i\phi}$</td>
</tr>
<tr>
<td>$S_{\phi}$</td>
<td>negative part of source term $S_{i\phi}$</td>
</tr>
<tr>
<td>$T_{TOT}$</td>
<td>total temperature</td>
</tr>
<tr>
<td>$U_i$</td>
<td>mean velocity vector ($i = x, y, z$)</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulence kinetic energy</td>
</tr>
<tr>
<td>$m$</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>$u_i$</td>
<td>turbulence velocity vector ($i = x, y, z$)</td>
</tr>
<tr>
<td>$\omega_k$</td>
<td>rotation rate of coordinate axis about $k$ axis</td>
</tr>
<tr>
<td>$\rho$</td>
<td>mean density</td>
</tr>
<tr>
<td>$\tau_w$</td>
<td>wall stress</td>
</tr>
</tbody>
</table>

$\beta$ | absolute flow angle |
$\epsilon$ | adiabatic efficiency |
$\epsilon$ | dissipation rate of turbulence kinetic energy |
$\gamma$ | ratio of the specific heats |
$\Omega_{ij}$ | mean rotation rate tensor, eqn. (8) |
$\mu$ | molecular viscosity |
$\mu_T$ | turbulence eddy viscosity |
$\gamma$ | ratio of the specific heats |
$\rho$ | mean density |
$\tau_w$ | wall stress |

Greek Symbols
The third model used in this study, developed by CMOTT, is a refinement of the standard $k - \epsilon$ model. In the SKE model the eddy viscosity coefficient, $C_\mu$, is assigned a value of 0.09. This value was obtained from the log-layer of the flat plate boundary layer where the production of turbulence kinetic energy is roughly balanced by its viscous dissipation (i.e. $-\nabla U_i \cdot \nabla U_j = \epsilon$). In the literature this is referred to as the local equilibrium assumption. Although local equilibrium is a correct assumption for the log-layer of a flat plate turbulent boundary layer it is not valid in general cases where the advection and turbulent diffusion make up a non-negligible part of the turbulence kinetic energy budget. In order to overcome this deficiency CMOTT suggested a variable formulation for $C_\mu$ (Shih et al. 1994). In this formulation the value of $C_\mu$ is dependent on a local parameter $(U_*/k)/c$ where $U_*$ represents the inverse of the mean time scale and $k/c$ represents the turbulence time scale. This formulation recovers the value of 0.09 for flows in local equilibrium. The formulation for $C_\mu$ is represented by the following set of equations.

\[
C_\mu = 4.0 + A_\mu U^2 (k/c)
\]

\[
U_* = \sqrt{S_{ij} S_{ij} + \Omega_{ij} \Omega_{ij}}
\]

\[
\tilde{\Omega}_{ij} = \Omega_{ij} - 2 \epsilon_{ijk} \omega_k, \quad \Omega_{ij} = \Omega_{ij} - \epsilon_{ijk} \omega_k
\]

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right), \quad \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right)
\]

\[
A_\mu = \sqrt[6]{6 \cos \phi}, \quad \phi = \frac{1}{3} \arcsin (\sqrt{6} W)
\]

\[
W = S_{ij} S_{jk} S_{ki}/S^3, \quad S = \sqrt{S_{ij} S_{ij}}
\]

In the above equation $S_{ij}$ is the mean strain rate tensor and $\Omega_{ij}$ is the mean rotation rate tensor. Further $\tilde{\Omega}_{ij}$ is the mean rotation rate as viewed in the rotating reference frame and $\omega_k$ is the angular velocity at which the reference frame is rotating. It is a well known fact that the SKE model does not account for the effect of rotation of coordinate axis on the evolution of turbulence. On the other hand the variable $C_\mu$ formulation given above is dependent on the rotation rate tensor, and, therefore is able to respond to rotation by modifying the turbulent eddy viscosity. (For more details see Shih et al. 1994). This model will be referred to as the CKE model in this paper.

Boundary Conditions

All the models were tested in conjunction with a standard wall function implementation. The boundary conditions near the solid surfaces on $k$ and $\epsilon$ were obtained from the following relations (Lauder and Spalding 1974).

\[
k = \frac{\nu^2}{\sqrt{C_\mu}}, \quad \epsilon = \frac{\nu^2}{\kappa y}
\]

where: $C_\mu = 0.09$, $\kappa = 0.41$ (Von Karman constant), and $u_*$ is the friction velocity ($u_* = \sqrt{\tau_w/\rho}$). Note that $y$ is the normal distance from the wall for the first grid point.

At the inlet the incoming turbulence was assumed to be isotropic with a turbulent intensity of 1%. Since $\epsilon$ was not measured in the ROTOR 37 flow, a reasonable estimate of its value has to be made. In the present study it is assumed that the incoming flow is fully turbulent with a turbulent eddy viscosity of 100$\mu$, where $\mu$ is the molecular viscosity. With these assumptions the inlet boundary conditions on $k$ and $\epsilon$ can be obtained as

\[
k = \frac{3}{2} U_{in}^2 (\frac{1}{100})^2, \quad \epsilon = C_\mu \rho k^3 \frac{1}{\mu_T}
\]

It is known that in some flows the computations are very sensitive to the inlet boundary condition for $\epsilon$. Therefore, it is desirable to supplement the above estimate of turbulent eddy viscosity by some other method as well. Since, based on dimensional reasoning, $\epsilon = k^{5/3}/l$, some researchers specify the length scale $l$ to estimate the dissipation rate at inlet. This length scale is taken as some fraction of the characteristic length scale of the geometry. For example Turner and Jennions (1993) in their investigation of transonic fans use $l = L/20$ where $L$ is the annular height of the fan. We can use these relations to estimate $\epsilon$ for the current problem. This can then be used in Eqn. (11) to estimate turbulent eddy viscosity. The value of turbulent viscosity so obtained comes out to be about 50$\mu$ which is of the same order as 100$\mu$. Later we will also look at the sensitivity of results if the inlet turbulent eddy viscosity at the inlet is taken as ten times the molecular viscosity. (Note that a smaller eddy viscosity corresponds to a higher dissipation rate and vice versa.)

At the outflow the gradients of the $k$ and $\epsilon$ with respect to the flow direction were taken as zero.

**NUMERICAL SCHEME**

The discretization and solution procedure for the mean flow is described in Adamczyk et al. (1990), and therefore only the discretization and the solution procedure for the turbulence equations (3) and (4) will be described here. The turbulence equations were integrated over the cell volume to obtain the finite volume analogue of the differential equations. The convection term in these flux equations was approximated using a first order upwinding scheme. The diffusion term was split into two parts; one involving the normal direction derivative and the other the cross direction derivative. The first was combined with the convection term and treated implicitly. The second was combined with the source term. The source term, $S_\phi$, which appears in
both Eqns. (1) and (2), was linearized by splitting it into a positive and a negative part as $S_x = S_x^p + S_x^\phi$. This was done to enhance the stability of the numerical scheme.

The set of algebraic equations was solved by an ADI scheme where the sweep in each direction required a solution of a tridiagonal matrix. Thomas algorithm was used for solving the tridiagonal matrix.

The grid used in the present study is H-type. It has 132 points in the axial direction, 51 points in the radial direction and 41 points in the pitchwise direction. This is the same mesh as was used by Celestina (1994) in the blind test case calculations.

Tip clearance flow was modeled by imposing periodic boundary conditions across the gap and using a discharge coefficient of 0.5 for the vena contracta. In the present study two grid cells were used across the clearance gap. Several earlier studies have shown that this procedure provides a reasonable approximation for calculating the mass flow through the clearance gap.

The solution procedure consisted of first solving the mean flow equations and then updating the boundary conditions of $k$ and $\epsilon$. After solving the $k$ and $\epsilon$ equations the eddy viscosity was updated. The convergence criteria consisted of monitoring the residuals, as well as the mass flow and the overall pressure ratio. Typically it required at least 3000 iterations to obtain a converged solution. It should be noted that the turbulence equations were solved every fifth iteration. Because of this the $k - \epsilon$ model required only 12% extra CPU time over the Baldwin-Lomax model.

As an illustration Fig. 2 shows the convergence history of mass flow and overall pressure ratio for CKE model for the first 3000 iterations. The case shown corresponds to $m/\dot{m}_{choke} = 0.984$. We note that mass flow at inlet is about equal to the mass flow at exit after about 1000 iterations. From this point onwards both the mass flow and overall pressure ratio seems to be settling down.

RESULTS

The results from the computations are compared with data from the Rotor 37 experiment (Suder 1994). In the following comparisons, all the results for the BL model are taken from Celestina (1994) and are reproduced here for comparison purposes. The overall measured and predicted performance of the rotor is shown in Fig. 3 in the form of speedlines for pressure ratio and adiabatic efficiency. We note that all the models slightly overestimate the pressure ratio and underestimate the efficiency. The CKE model reproduces the pressure ratio better than the BL and the SKE models.

Now we will compare the flow details from the computations with the data in the form of axisymmetric profiles of mean flow quantities. In the present paper all of these comparisons correspond to the operating condition of $m/\dot{m}_{choke} = 0.984$. Figure 4 shows the axisymmetric profile of the normalized total pressure at station 4. We note that the BL and SKE models give similar results in the midspan region. Near the tip, BL is closer to the data whereas near the hub SKE is closer to the data. We note that CKE is closer to the experiment than both BL and SKE although it overpredicts the data in both the tip and the hub regions.

The improved prediction of the total pressure profile by the CKE model was the reason for improvement shown in Fig. 4 over the others. This underlines the fact that the flow details must be predicted to a greater degree of accuracy in order to obtain just a slight improvement in the prediction of the overall performance.

Figure 4 also shows the axisymmetric profile for normalized total temperature. Again we find similar trends for the three models with CKE closer to the experiment. CKE model is also able to reproduce the two dips in the temperature profile.

From the axisymmetric profile of adiabatic efficiency calculated by the three models (Fig. 4), we find that the BL model is closest to the data. The absolute flow angle is reproduced reasonably well by all three models. It should be noted that the adiabatic efficiency shown above is calculated using the total pressure and total temperature profiles. This raises the question that if the total pressure and total temperature profiles from the CKE model are closer to the data then why is it not reflected in the adiabatic efficiency profile? This can be answered by looking at the equation for the adiabatic efficiency which is given by

$$\eta = \frac{(P_{T1}/P_{T0})^{\gamma/\gamma - 1} - 1}{(T_{T1}/T_{T0}) - 1}$$

It just so happens that for the SKE model even though the numerator and denominator of the above equation are not as close to the data as the CKE model, their ratio is such as to give a value closer to the data.

The wake profiles, in the form of radial Mach numbers, at stations 3 and 4 for three different spanwise locations are shown in Fig. 5. These wake profiles are for the operating point $m/\dot{m}_{choke} = 0.984$. The wakes from the three models are deeper than the data. It is also interesting to note that the shape of the wakes at station 4 from all three models are very similar to that measured. It should be pointed out that no attempt was made to slide the wakes in order to match the wake minimum of the computations to the measured wake minimum. On the overall we note that the three models perform more or less the same, predicting wakes which are too deep at locations close to the trailing edges of the blades.

Several sensitivity studies were carried out in order to establish the accuracy of the above results. Except one, all of these sensitivity studies were done with the CKE model
since it performed better than the other two models. Because of space limitations only the profiles for the total pressure and total temperature at station 4 will be presented.

First the sensitivity of wall boundary conditions on $k$ and $\varepsilon$ is presented. As was mentioned before, a standard wall function procedure was used for the boundary conditions at solid surfaces. These boundary conditions assume the law of the wall. This implies that the first grid point off the solid surface should be at least at a distance of $y^+$ of 25. In a three dimensional complicated flow like ROTOR 37 it is not always possible to insure this. As a result there are always some regions in the flow field where $y^+$ is less than this value ($y^+$ could be as low as 10). In order to find out the sensitivity of the above results to the wall boundary conditions, the following procedure was implemented. When ever $y^+$ was less than 11 the wall shear stress was computed directly from the velocity profile (i.e. $\tau_w = \mu \partial U / \partial y$) and the boundary conditions on $k$ and $\varepsilon$ were obtained from the following relations (Shih and Lumley 1993)

$$k = 0.25u_r^2, \quad \varepsilon = 0.25 \frac{u_r^4 \rho}{\mu}$$

(12)

as opposed to using Eqs. (10) and (11). Note that this set of boundary conditions allows integration of mean and turbulence equations to the wall. The results for the axisymmetric total pressure and total temperature are shown in Fig. 6 and we notice that the results are almost identical to the baseline calculation.

The sensitivity of the results to the order of the upwind discretization scheme was also carried out. The results are shown in Fig. 7 and we find that the results are insensitive to the order of the upwinding scheme.

The issue of grid resolution in the pitchwise direction and its effect on wake evolution was also studied by unpacking the grid in this direction. This unpacking was achieved by reducing the grid stretching parameter. It was found that the wakes remained unchanged at 30% and 50% spanwise locations at both station 3 and station 4 and, therefore, are not shown. The maximum change occurred at 90% span, and the results are shown in Fig 8. Notice that the wake at station 3 has become slightly deeper than that associated with the baseline grid, and has moved away from the data. On the other hand at station 4 the wake profile moves toward the data.

Earlier in the paper the issue of specifying the turbulent dissipation rate at the inlet was discussed. It was pointed out that sometimes the computation of a flow field is sensitive to this inlet boundary condition. It was also mentioned that the estimate used in the present study is of the same order as the one obtained by specifying the length scale as a fraction of geometrical height at the inlet. However, it is still beneficial to study the effect of change of inlet value of this quantity on the computation.

As was pointed out before the inlet value for the turbulent dissipation rate was obtained from Eqn. (11) by assuming $\mu_T / \mu = 100$. In this section we show sensitivity to results if $\mu_T / \mu = 10$ at inlet. Figures 10 and 11 show the total pressure and total temperature profiles at station 4 for the SKE and CKE model respectively. Note that this case corresponds to $\dot{m}/\dot{m}_{\text{chke}} = 0.984$. We note that the results of the SKE model are somewhat sensitive to the inlet value of turbulent dissipation rate, with the results corresponding to $\mu_T / \mu = 100$ closer to data. On the other hand the CKE model shows a very little sensitivity to the inlet value of turbulent dissipation rate.

**SUMMARY AND CONCLUSIONS**

The ROTOR 37 flow field was computed using the standard two equation $k - \varepsilon$ model and a CMOTT improved version of it. The results from these, along with the results from the Baldwin-Lomax model, Celestina (1994), were compared with the data from the ROTOR 37 experiment. Since all the models were calculated using the same numerical platform, the study provides an objective assessment of the three models. In addition, several sensitivity studies were also carried out. These included: changing the wall boundary conditions for the turbulence models; changing the order of the upwind discretization scheme; unpacking the grid in the pitchwise direction; and the effect of inlet boundary condition for turbulence. It is found that the CKE model performed better than the SKE and BL models for the total pressure and total temperature profiles. It was also found that the CKE was a lot less sensitive to the changes in the inlet boundary conditions for turbulence than the SKE model. It is noted that there are still deficiencies in predicting the experiment. For example the flow near the hub, and the wakes which were predicted too deep by all the three models. Work on addressing these deficiencies is underway.

As was pointed out before the only difference between the SKE and CKE model is the value of the eddy viscosity coefficient $C_\mu$. The SKE model uses a constant value of this coefficient which is obtained using the local equilibrium assumption. In the CKE model this coefficient is dependent on local flow parameters. For flows in local equilibrium CKE model gives the same value of this coefficient as the SKE model. The improved performance of the CKE model is attributable to this eddy viscosity coefficient formulation. This formulation captures the non-equilibrium regions of the flow field, such as shock wave boundary layer interaction, better than the SKE model. This in turn leads to the improved prediction of the overall flow-field.

**ACKNOWLEDGMENTS**

The idea of assessing different models for ROTOR 37 was brought to our attention by J.J. Adamczyk and we
want to thank him for his support for this work. We thank R.A. Mulac and T.A. Beach for their help in grid generation. We also acknowledge K. Suder and A. Strazisar for providing us experimental data from their unpublished work. A. Shabbir and J. Zhu would also like to thank Institute for Computational Mechanics in Propulsion and the Office of Advanced Subsonic Technology for their financial support of this work.

REFERENCES


Fig. 1. A meridional view of the ROTOR 37 flow field showing locations where experimental data was acquired. (Taken from Suder 1994.)
Fig. 2. Convergence history of a CKE model run showing the change in mass flow and the overall pressure ratio across the rotor.

Fig. 3. Speedlines for the pressure ratio and the adiabatic efficiency for the different models.

Fig. 4. The axisymmetric profiles of normalized total pressure, normalized total temperature, adiabatic efficiency, and absolute flow angle at station 4. (Figure continues on next page.)
Fig. 5. The wake profiles at stations 3 and 4 for various spanwise locations for the high flow case \((m/m_{\text{choke}} = 0.98)\). No attempt was made to slide the wakes for matching the wake minimum of the computations with the wake minimum of the experiment. (Figure continues on next page)
Fig. 6. The sensitivity to the wall boundary condition for turbulence model on the axisymmetric profiles of normalized total pressure and normalized total temperature at station 4. The results shown are for the CKE model. Note that the results for the standard wall boundary condition are the same as shown in Fig. 1.

Fig. 7. The sensitivity to the order of upwind discretization on the axisymmetric profiles of normalized total pressure and normalized total temperature at station 4. The results shown are for the CKE model. Note that the results for the 1st order scheme are the same as shown in Fig. 1.
Fig. 8. Effect of unpacking the grid in the pitchwise direction on the wake evolution at 90% span.

Fig. 9. Effect of inlet boundary condition on turbulence on the total pressure and total temperature profiles at station 4 for SKE model.

Fig. 10. Effect of inlet boundary condition on turbulence on the total pressure and total temperature profiles at station 4 for CKE model.