Navier-Stokes Computations for Turbulent Flow Predictions in Transonic Turbine Cascade Using a Zonal Approach

J.J. Yeuan, A. Ilamed, and W. Tabakoff

Department of Aerospace Engineering and Engineering Mechanics
University of Cincinnati
Cincinnati, Ohio

ABSTRACT
Numerical results are presented for viscous flow through a transonic turbine cascade using different turbulence models and H-type grids. The explicit Navier-Stokes solver used in the solution was developed with an option of conservative zonal approach for interpolation across the periodic boundaries with minimum numerical errors. This approach allows the use of a grid that is more orthogonal and less skewed which leads to higher accuracy in the prediction of turbine blade performance. The results obtained with an algebraic and two equation turbulence models, and with two types of H grids are compared at two different flow conditions.

NOMENCLATURE

\begin{itemize}
\item $c$: Total energy per unit volume
\item $c_n$: Unit vector in m direction
\item $c_\theta$: Unit vector in $\theta$ direction
\item $F$: m-wise flux vector
\item $G$: $\theta$-wise flux vector
\item $h$: Streamtube thickness
\item $J$: Jacobian
\item $k$: Turbulent kinetic energy
\item $M$: Local Mach number
\item $P$: Static Pressure
\item $P_t$: Production of turbulent kinetic energy
\item $P_\epsilon$: Production of dissipation of turbulent kinetic energy
\item $Pr$: Prandtl number
\item $Pr_t$: Turbulent Prandtl number
\item $q$: State Vector
\item $R$: m-wise viscous flux vector
\item $r$: Radius coordinate
\item $Re$: Reynolds number
\item $S$: $\theta$-wise viscous flux vector
\item $t$: Time
\item $V_m$: m-wise absolute velocity
\item $V_\theta$: $\theta$-wise absolute velocity
\item $W_m$: m-wise relative velocity
\item $W_\theta$: $\theta$-wise relative velocity
\item $y_n$: Compressibility corrected normal distance from wall
\item $y^+$: Profile shape, $\rho y^{-1/2}$
\item $\epsilon$: Dissipation of turbulent kinetic energy
\item $\theta$: Tangential coordinate
\item $\xi$: Loss coefficient
\item $\rho$: Density
\item $\mu$: Molecular viscosity
\item $\mu_t$: Turbulent viscosity
\item $\gamma$: Specific heat ratio
\item $\Omega$: Vorticity
\item $\sigma$: Total viscous stresses
\item $\Omega$: Rotational angular velocity
\end{itemize}

INTRODUCTION
In the continuing efforts for increased blade loading, several investigations have been aimed at gas turbines operating in the transonic flow regime. The prediction of transonic turbomachinery flow fields is very challenging because of the complex effects arising from shock boundary layer interactions on the blade surface. These interactions near the trailing edge of the suction surface are especially critical to the performance. In addition, the complex viscous flow in the base region of transonic turbine blades determines the shock structure at the blade trailing edge.
and the associated blade row losses (Sieverding, 1980). The experimental results of Mee et al. (1990) indicate that the mixing losses generated immediately downstream of the trailing edge increase with increased flow velocities and dominate the overall transonic turbine losses at supersonic exit Mach numbers.

Quasi-three-dimensional inviscid flow solvers were proposed as a fast, less expensive and less complicated tool in the design of turbomachinery (Simoneau and Hudson, 1989). The earliest quasi-three-dimensional flow formulation for turbomachinery applications was proposed by Wu (1952). Several different procedures were presented for the numerical solution of inviscid flow on axisymmetric blade-to-blade stream surface (McFarland, 1984, Katsanis, 1969, Farrel and Adamczyk, 1982, and Bertheau et al., 1985). Inviscid solutions, even when combined with a blade surface and end wall boundary layer solutions, cannot adequately model the effects of the separated flow in the case of strong shock wave boundary layer interactions over the blade surface. A thin-layer Navier-Stokes solution using an explicit two-stage Runge-Kutta finite-difference algorithm was developed by Chima (1986) for the numerical solution of the finite-difference equations. Blade-to-blade axisymmetric flow solutions of the full Navier-Stokes equations, based on an explicit fractional step algorithm, were reported by Simandirakis et al. (1989).

In the present investigation, the numerical scheme for solving the Navier-Stokes equations and k-ε equations is based on Runge-Kutta explicit method with the option of two stage and five stage time marching algorithms (Jameson et al., 1981, and Holmes et al., 1985 and 1989). The Navier-Stokes equations solved in this code are formulated in the axisymmetric (m,θ) coordinate system for blade-to-blade flows with a specified stream filament thickness distribution. The finite volume approach is used in this code with the volume and area projections in the quadrilateral mesh calculations based on the inverse Jacobian and metric coefficients in body-fitted coordinate system. The computational results are presented for the flow field in a transonic turbine cascade which has been experimentally investigated in four European wind tunnels (Klock et al., 1986). The computed results are compared for two turbulence models and for two different H-type grids (traditional and nonperiodic). A zonal approach modified from Rai (1986) is used to interpolate the flow conditions across the nonperiodic H-type grids at the periodic boundaries (Yuan et al., 1992). This grid type is used to overcome the problems of skewed mesh in the traditional H-type grid that can occur toward the trailing edge in the transonic turbine blade with high exit flow angle. The effect of the turbulence model on the quality of flow field results is quantified from the differences among the results using the same Navier-Stokes solver, the same grids and the same boundary conditions for the Baldwin-Lomax algebraic and k-ε turbulence models. The Navier-

Stokes solutions from two different grids, one with highly skewed grid and another with near orthogonal grid, demonstrate the effect of grid skewness on the blade surface pressure distribution, flowfield properties and exit flow losses.

GOVERNING EQUATIONS
The quasi conservative form of the dimensionless Navier-Stokes equations and low Reynolds number k-ε equations on a blade-to-blade axisymmetric stream surface are written in meridional coordinates:

\[
\frac{\partial q}{\partial t} + \frac{\partial \left( r_1 Re^{-1} R \right)}{\partial m} + \frac{\partial \left( r_2 Re^{-1} S \right)}{\partial \theta} = Re^{-1} K
\]

where:

\[
\begin{align*}
q &= \begin{bmatrix} p \\ pV_m \\ pV_m^2 \\ pV_m \rho V_m \\ \rho e \\ pV_m - \frac{pV_m^2}{\rho} \\ \rho V_m - \frac{pV_m^2}{\rho} \end{bmatrix},
R &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
S &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
K &= \begin{bmatrix} 0 \\ K_2 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
\end{align*}
\]

\[
\begin{align*}
R &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
S &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
K &= \begin{bmatrix} 0 \\ K_2 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
\end{align*}
\]

\[
\begin{align*}
\frac{\partial \left( r_1 Re^{-1} R \right)}{\partial m} &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
\frac{\partial \left( r_2 Re^{-1} S \right)}{\partial \theta} &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix},
Re^{-1} K &= \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}.
\end{align*}
\]
\[ e = \rho \left[ C_v + 0.5(V_m^2 + V_0^2) \right] \]

\[ P = (\gamma - 1) \left[ e - 0.5 \rho (V_m^2 + V_0^2) \right] \]

\[ K_2 = \left[(\rho V_m^2 + P) \frac{\partial e}{r \partial m} - (P + \rho \sigma_3)^2 \frac{1}{h \partial m} \right] \]

In the above equations, \( W \) and \( V \) are the relative and absolute flow velocity components, \( e \) is the total energy per unit volume, and \( P \) is the static pressure.

The viscous terms in the energy equation are expressed as follows:

\[ R_6 = \left( \frac{\mu_1}{Pr} + \frac{\mu_1}{Pr} \right) \frac{1}{\gamma - 1} \frac{\partial^2 e}{r \partial m} + \frac{1}{r} \frac{\partial^2 e}{\partial \theta \partial m} + \frac{1}{r} \frac{\partial e}{\partial \theta} + \frac{1}{r} \frac{\partial e}{\partial \theta} \]

\[ \sigma_{22} = 2 \mu \frac{\partial V_m}{r \partial m} + \lambda \frac{\partial V_m}{r \partial \theta} \]

\[ \sigma_{33} = 2 \mu V_m \frac{\partial h}{r \partial m} + \lambda \frac{\partial V_m}{r \partial \theta} \]

\[ \sigma_{12} = \mu \left[ \frac{\partial V_m}{r \partial m} - \frac{\partial V}{\partial m} \frac{1}{r} \frac{\partial e}{\partial \theta} + \frac{1}{r} \frac{\partial e}{\partial \theta} \right] \]

\[ \lambda \frac{\partial V_m}{r \partial m} = \frac{2}{3} \mu \left[ \frac{\partial V_m}{r \partial m} \frac{1}{r \partial m} \frac{1}{r \partial \theta} \right] \frac{1}{r \partial \theta} \]

where \( \lambda \) is the local sonic velocity

\[ \sigma^2 = \frac{\gamma P}{p} \]

In the present analysis, the inlet total temperature, total pressure, and critical sonic velocity are used as the reference conditions. The effective viscosity, \( \mu \), is calculated from the sum of the molecular dynamic viscosity, \( \mu_m \), and the turbulent eddy viscosity, \( \mu_t \). The turbulent Prandtl number, \( Pr_t \), is 0.9 for air.

The compressible low Reynolds number \( k-\epsilon \) turbulence model of Nichols (1990) is used in the present investigation, since it simulates the effect of compressibility and is validated through comparison with experimental data for shock boundary layer interactions (Nichols, 1990). In addition, an algebraic turbulence model is also implemented that is based on the original Baldwin-Lomax model (Baldwin and Lomax, 1978) in the blade passage and the Thomas formulation (Thomas, 1979) beyond the trailing edge in the unbounded flow regions. Only the maximum vorticity is required in the Thomas formulation, thus eliminating the need to determine the wake centerline in the calculations.

**NUMERICAL METHOD**

The difference equations are obtained from the volume integration of the governing equations as follows:

\[ \frac{\partial q}{\partial t} = \frac{\partial}{\partial \theta} \left[ \int (F - Re^{-1}R)nds e_m \right] \]

\[ -r \left[ \int (G - Re^{-1}S)nds e_n + K \right] \]

where \( \delta t \) is time step, \( \delta v \) is the control volume, \( n \) is the normal vector normal to the control volume surface, \( s \) and \( e_m \) and \( e_n \) are unit vectors in the \( m \) and \( \theta \) directions respectively.

The numerical solution is advanced in time using Runge-Kutta schemes. Both two stage and five stage cell centered Runge-Kutta schemes (Jameson, 1981, and Holmes et al, 1985, 1989) are coded in the present solver. Second and fourth difference smoothing are applied with the switching of second difference smoothing controlled by second derivative pressure differences in the Navier-Stokes and \( k-\epsilon \) equations. Local time stepping with a constant Courant number, 0.95, is used to obtain the steady solutions.

**BOUNDARY CONDITIONS**

At inlet, the total pressure, total temperature, and swirl \( rV_s \) are specified while the density and energy are calculated from isentropic relations. A Riemann invariant is used to update the inlet meridional velocity (Chima, 1986). The \( k \) and \( \epsilon \) terms are specified according to experimental turbulence levels. At the exit plane, the static pressure is specified and other flow quantities are obtained using first order extrapolation. The normal momentum equation based on first order accurate one sided differencing is used to calculate the blade surface pressure, and the \( k \) and \( \epsilon \) terms are set equal to zero at the blade surface.

The periodic boundary conditions require special treatment in the nonperiodic \( H \)-type grid to insure that the interpolation is conservative. Rai (1986) proposed an
interpolation scheme that satisfies the global conservation properties of the flux vectors across the zonal boundaries for Euler equation based on the contravariant flux vectors. In the present study, a similar scheme was developed for the explicit Runge-Kutta integration schemes. Since the flux vectors are generally not perpendicular to the control volume boundaries in the present investigation, the interpolation scheme is applied directly to the variation of the state vector, $\delta q$, to preserve the local conservation properties. The renewed state vector, $q$, at grid points on the coarse side is obtained from interpolation along the zonal boundaries. A cubic spline-fit of the coarse side state vector, $q$, along the boundaries is used to obtain the state variable, $q$, at grid points on the dense side boundaries. This treatment which is applied here in the viscous flow calculations has been previously verified for inviscid flow calculations (Yeuan, Tabakoff and Hamed, 1992).

RESULTS AND DISCUSSIONS
Numerical solutions were obtained for the flow in the transonic turbine cascade for which the experimental results were obtained by Kiock et al. (1986), at inlet turbulence level of 1%. The design parameters and flow conditions for this cascade are summarized in Tables (1) and (2). The computations were performed for two types of H grids with the same number of mesh points (222x51). The traditional H-type grid mesh shown in Fig. (1) has 236 points on blade surface. The nonperiodic H-type grid mesh shown in Fig. (2) has 139 points on the blade suction surface and 61 points on the blade pressure surface. Both grids were generated using an elliptic technique (Thompson et al., 1985). The first grid point above blade surface was at $Y^*$ between 0.5 to 2.5 over most of blade surface except in the leading edge region where it reaches values between 3 to 4.5. The convergence criteria was based on local

Table 1

| Spacing / Chord (S/C):               | 0.71 |
| Exit Blade Angle (arc cos(S/C)):    | 67.8 |
| Inlet Blade Angle:                  | 33.3 deg. |
| Inlet Flow Angle:                   | 30.0 deg. |
| Chord Length:                       | 60.0 mm |

Table 2

<table>
<thead>
<tr>
<th>Transonic</th>
<th>Subsonic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Flow Angle: 30.000</td>
<td>30.000</td>
</tr>
<tr>
<td>Inlet Mach Number: 0.282</td>
<td>0.270</td>
</tr>
<tr>
<td>Exit Mach Number: 0.937</td>
<td>0.782</td>
</tr>
<tr>
<td>Exit Reynold Number: $8.8 \times 10^8$</td>
<td>$8.2 \times 10^8$</td>
</tr>
</tbody>
</table>

Fig. 1. The Traditional H-Type Grid.
maximum pressure error of $1.0 \times 10^{-4}$ and RMS residual of $10^{-5}$. Approximately 1500 iterations were sufficient to achieve this level of convergence in all viscous solutions. The computed results are presented in Figures (3) through (6) for the transonic case and in Figures (7) and (8) for the subsonic case. Comparing Figures (3) and (4) for the Mach number contours, one can observe the improved flow resolution over the second half of the blade suction surface and downstream of the blade achieved with the nonperiodic grid. This is especially clear in capturing the shock near the suction surface trailing edge which is very crisp in the predictions with the nonperiodic H-type grid. The thicker boundary layers and wakes in Fig. (3) in the calculation are due to numerical dissipation in the highly skewed traditional grid in these regions. This is also confirmed by the grossly overpredicted losses in comparison to the experimental results as will be discussed later in connection with Fig. (9).

The results for the surface isentropic Mach number are
Fig. 4. The Mach Number Contours Obtained From Nonperiodic H-Type Grid for Transonic case. (Mmax = 1.14, Mmin = .14, Increment = 0.02)

Fig. 5. The surface isentropic Mach number. (Transonic case, traditional H-type grid).

Fig. 6. The surface isentropic Mach number. (Transonic case, Nonperiodic H-type grid).

Fig. 7. The surface isentropic Mach number. (Subsonic case, traditional H-type grid).

Fig. 8. The surface isentropic Mach number. (Subsonic case, Nonperiodic H-type grid)
presented and compared with the experimental results for the transonic case in Figures (5) and (6), and for the subsonic case in Figures (7) and (8). Much better agreement between the experimental and computed results are obtained in the transonic case with the nonperiodic type grid. The differences between the results for the two types of grids are less noticeable in the subsonic case (Figures (7) and (8)). One difference is in the improved predictions of the local peak in the isentropic Mach number on the suction surface at \( x/c = 0.61 \) in the case of nonperiodic grid. Figures (5) through (7) demonstrate that the k-\( \varepsilon \) turbulence model does not improve the predictions over the algebraic turbulence model. The kink in the surface Mach number distributions on the pressure side at 80% axial chord in Figures (5) through (8) appears to be due to the blade surface configuration, since it is also reported in the results of other investigations using totally different schemes and grids by Hwang and Liu (1992).

The predicted exit flow angles and loss coefficients for the transonic and subsonic cases are shown in Figures (9) and (10) together with the experimental data. Both are obtained at an axial distance of 27 mm (0.5 chord length) downstream of trailing edge. According to Fig. (9), the computed exit flow angles for the subsonic and transonic cases are in close agreement with the experimental results. The largest differences are seen in the case of transonic flow prediction with the Baldwin-Lomax model and traditional grid where the flow exit angle is underpredicted. Fig. (10) shows that the predicted loss coefficient using the nonperiodic type grid are generally much closer to the experimental results. The loss predictions with the nonperiodic type grid are in very close agreement with the experiments and are within 10% in the transonic case with the k-\( \varepsilon \) model. In both subsonic and transonic cases, the computations based on the traditional type grid grossly overpredict the loss coefficient by a factor of more than 2. While the k-\( \varepsilon \) model slightly reduces the value of the predicted loss coefficients, it is still much higher than the experimental data. The overprediction of the loss coefficient in the case of the traditional type grid can be attributed to the numerical diffusion associated with the highly skewed grid over the blade suction surface and downstream of the trailing edge, and to the inferior modelling of the shock boundary layer interactions over the blade surface.

CONCLUSIONS
Computational results are presented for a subsonic and a transonic flow cases in transonic turbine cascade based on an explicit Navier-Stokes code using traditional and nonperiodic H-type grids. Several conclusions can be drawn from the present results using two different turbulence models.

1. Grid skewness in the traditional H-type grid adversely affects the prediction of loss coefficient, and smear the large flow gradient in the aft part of the suction surface and shock near the trailing edge.

2. The solutions obtained with the nonperiodic H-type
grid and k-ε model are the closest to the experimental results in predicting the losses and exit flow angles.

3. The flow predictions using Baldwin-Lomax turbulence model can be very competitive to those using k-ε turbulence model in the case of nonperiodic H-type grid.

ACKNOWLEDGEMENT
The computations were performed on the CRAY-YMP of the Ohio Supercomputer Center. The authors would like to acknowledge S.H. Shih’s contributions through discussions of coding k-ε model and artificial dissipations.

REFERENCES