NUMERICAL CALCULATION OF THREE-DIMENSIONAL TURBULENT FLOW IN A TURBINE-STATOR PASSAGE

B. Song and R. S. Amano
Department of Mechanical Engineering
University of Wisconsin-Milwaukee
Milwaukee, WI 53201

ABSTRACT
A numerical study on three-dimensional turbulent flow in a turbine-stator passage is presented in this paper. The standard model with the one equation, near-wall model (SKE) and the Launder-Sharma model (L-S) are used for turbulence computations. The computational results obtained using these models were compared in order to investigate the turbulence effect in the near-wall region. The governing equations in a generalized curvilinear coordinate system are discretized by using the SIMPLEC method with non-staggered grids. The oscillation of pressure and velocity due to non-staggered grids is eliminated by using the interpolation method suggested by Rhie and Chow (1983). The predicted midspan pressure coefficients using SKE and L-S models are compared with experimental data. It was shown that the present results obtained by using both models are satisfactory. Computations were then extended to cover the entire blade-to-blade flow passage, and the three-dimensional effects on pressure and turbulence kinetic energy were evaluated. It was observed that two turbulence models predict similar results for the pressure and velocity but these predict different results in the turbulence kinetic energy.

1. INTRODUCTION
Modern turbomachinery operates under extremely complex three-dimensional flow conditions, and further improvement requires detailed knowledge of the flow structure. In particular, the need to estimate off-design conditions, secondary flows, and turbulence and heat transfer demand that viscous models be examined. Near the hub and tip of a turbine-stator passage, the flow is affected by the interaction between the longitudinal boundary layer (streamwise) and the side wall boundary layer. Although this region is small, its effect on the overall aerodynamic performance can not be neglected. To design a high performance turbine, an engineer has to understand the detailed three-dimensional flow field near the hub and tip. The flow through the midspan of a turbine stator passage is, principally, driven by the viscous process. However, some important characteristics and flow parameters are strongly influenced by the turbulence transport near the solid walls and the wake region behind the airfoils. Under certain operating conditions, the boundary layer development on the blade surface is much enhanced due to the existence of adverse pressure gradients, which will have considerable effects on the following rotor stage. For these reasons, a technique for executing three-dimensional computations becomes more and more essential. In the past decade computational fluid dynamics (CFD) has undergone an impressive evolution in turbomachinery research. So far numerous research studies in three-dimensional CFD in turbomachinery have been reported. Rai (1987) and Rao and Delaney (1990) studied unsteady three-dimensional flows. Adamczyk et al. (1990) calculated three-dimensional viscous flow through multi-stage turbines. Subramanian and Bozola (1987), Chima and Yokota (1988), Choi and Knight (1988), Davis et al. (1988), Hah (1989), Nakahashi et al. (1989), and Weber and Delaney (1991) solved three-dimensional Navier-Stokes equations.

Despite the abundance of higher-order turbulence models, model incorporation into a compressible turbomachinery flow field is still complex. Dawes (1992) compared zero- and one-equation turbulence models. One of the most widely used turbulence models in a compressible aerofoil flow is the one developed by Baldwin and Lomax (1978). Although Amano and He (1993) and Fan and Lakshminarayana (1993) have successfully formulated higher-order turbulence closure models and employed them in turbomachinery blade calculations, these were limited to two-dimensional flows. In many cases, however, these models are still widely used in practical industrial applications for three-dimensional turbulence flows due mainly to their excellent numerical stability, robustness, simplicity, and generally reliable results. Admitting one of the shortcomings of the standard \( \kappa - \varepsilon \) model, e.g., unreliability of the computations in the near-wall regions, a near-wall model (Iacovides and Launder, 1987) has to be employed to correct the error in such a region. Still, these near-wall models usually cause complexity in the formulations for three-dimensional computations. With a low-Reynolds number model proposed by Launder-Sharma (1974), such difficulties are partly overcome and the model has been shown to be convenient for near-wall turbulence computations. The purpose of this study is to predict the three-dimensional turbulence and pressure fields in a turbine-stator...
2. MATHEMATICAL MODELS

2.1 Governing Equations

For the steady-state turbulent flows, the averaged governing equations are given as follows:

Continuity:
\[ \frac{\partial \rho U_i}{\partial x_i} = 0 \]  
(1)

Momentum:
\[ \rho \left( \frac{\partial U_j}{\partial x_j} + \epsilon \frac{u_i}{u_j} f_k \right) = \frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} - \rho u_i u_j \right) \]  
(2)

where \( U \) represents the time-averaged mean velocity, \( u \) the turbulence fluctuation velocity components, and \( f_k \) is Coriolis force. The Reynolds stresses are represented by \(- \rho u_i u_j\). These stresses can be determined from the Boussinesq assumption in which the Reynolds stresses are linearly related with strain tensor, that is:
\[ -\rho u_i u_j = \frac{2}{3} \sigma_{ij} + \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \]  
(3)

where \( u_i \) is turbulence dynamic viscosity and \( k \) the turbulence kinetic energy.

2.2 Standard \( k-\varepsilon \) Model With One-Equation Near-Wall Model

In the standard \( k-\varepsilon \) model (SKE), the turbulence dynamic viscosity is given by:
\[ \mu = C_\mu \rho \frac{k^2}{\varepsilon} \]  
(4)

where \( \varepsilon \) denotes the turbulence energy dissipation rate. Both the turbulence kinetic energy and the turbulence energy dissipation rate are determined from their own transport equations of the forms shown as follows:

Turbulence Kinetic Energy:
\[ \rho U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_k} \right] \frac{\partial k}{\partial x_i} - \rho \left( P_k - \varepsilon \right) \]  
(5)

Turbulence Energy Dissipation Rate:
\[ \rho U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_\varepsilon} \right] \frac{\partial \varepsilon}{\partial x_i} - \frac{\rho \varepsilon}{k} \left( C_{d1} P_k - C_{d2} \varepsilon \right) \]  
(6)

where \( P_k \) is the production term given as:
\[ P_k = -\frac{\partial U_i}{\partial x_j} \frac{\partial U_i}{\partial x_j} \]  
(7)

In the near-wall region, the high Reynolds number version of the above equations do not simulate correctly. Therefore, the one equation model of turbulence by lacovides and Launder (1987) was employed. In this model the dissipation rate and turbulence viscosity near the wall are calculated with the prescribed length scale \( l_k \) and \( l_\varepsilon \), that is:
\[ \mu = C_\mu' \rho \sqrt{k} l_k \]  
(8)

and
\[ \varepsilon = \frac{k^2}{l_\varepsilon} \]  
(9)

where
\[ l_k = \frac{k}{\varepsilon} y \left[ 1 - \exp (-0.263 y^*) \right] \]  
(10)

and
\[ l_\varepsilon = \frac{k}{\varepsilon} y \left[ 1 - \exp (-0.016 y^*) \right] \]  
(11)

where \( y^* \) is the dimensionless distance and is given as:
\[ y^* = \frac{y}{v} \]  
(12)

Here \( y \) represents the normal distance from the wall and \( v \) the molecular kinematic viscosity. In the above equations, \( C_{d1}, C_{d2}, \mu \), and \( k \) are constants whose values are 0.09, 1.45, 1.92, and 0.42, respectively.

2.3 Launder-Sharma Model

For the Launder-Sharma model (Launder and Sharma, 1974) (L-S), turbulence viscosity is damped in the near-wall region. Thus,
\[ \mu_1 = C_\mu f_\mu \rho \frac{k^2}{\varepsilon} \]  
(13)

where \( f_\mu \) is the damping function which is calculated from the following equation:
\[ f_\mu = \exp \left[ -\frac{3.4}{(1 + 0.02 R_k)^2} \right] \]  
(14)

where \( R_k \) is the turbulence Reynolds number. This is given as:
\[ R_k = \frac{k^2}{\mu \varepsilon} \]  
(15)

where \( \varepsilon \) is defined as:
The kinetic energy equation is the same as Eq. (5). However, the dissipation rate equation has a different form from Eq. (6) which is given as:

$$\rho \frac{\partial \tilde{e}}{\partial t} + \frac{\partial}{\partial x_i} \left[ \mu + \frac{\mu_t}{\sigma_t} \frac{\partial \tilde{e}}{\partial x_i} \right] - \rho \frac{\partial}{\partial x_i} (C_{1f} f_{s1} \rho_k - C_{2f} f_{s2} \rho) = + E + Y_S$$

(17)

where $E$, $Y_S$, $f_{s1}$ and $f_{s2}$ are the correction terms which are given by:

$$E = 2\nu \left( \frac{\partial^2 U_{1}}{\partial x_i \partial x_i} \right)^2$$

(18)

$$Y_S = \text{Max} \left\{ 0.83 \left( \frac{k}{C_{1f} \bar{c}} \frac{\varepsilon}{ar{c}^2} \right)^{\frac{3}{2}} \left( \frac{k}{C_{1f} \bar{c}} \frac{\tilde{e}}{\bar{c}^2} \right)^{\frac{1}{2}}, 0 \right\}$$

(19)

$$f_{s1} = 1$$

(20)

$$f_{s2} = 1 - 0.3 \exp \left( -R_s^r \right)$$

(21)

where $C_f$ is 2.5, $y$ is the normal distance away from the wall, and $\gamma$ the turbulence kinematic viscosity.

The additional source term in the dissipation equation (17) $Y_S$ was originally introduced by Rhie and Chow (1983), in order to prevent the near-wall length-scale from becoming too large in a recirculating flow region. Computations in this study showed that the correction factor, $Y_S$, in the L-S model is needed in order to maintain the stability of the computational process.

2.4 Transformation of Governing Equations

Equations (2), (5), (6), and (17) can be written in the following general form:

$$\text{div} \left[ \hat{U} \phi - G_q \nabla \phi \right] = S_\phi$$

(22)

where $\phi$ is an arbitrary transport variable and $\hat{U}$ is the velocity vector.

Equation (22) is transformed from the Cartesian coordinates $(x,y,z)$ into generalized curvilinear coordinates $(\xi, \eta, \zeta)$. In the new coordinate system, Eq. (22) becomes:

$$\frac{1}{J} \left[ \frac{\partial (p \hat{U} \phi)}{\partial \xi} + \frac{\partial (p \hat{V} \phi)}{\partial \eta} + \frac{\partial (p \hat{W} \phi)}{\partial \zeta} \right] = \frac{1}{J} \frac{\partial}{\partial \xi} \left( \frac{G_q}{J} (j_{1s} \phi + j_{2s} \phi + j_{3s} \phi) \right) + \frac{1}{J} \frac{\partial}{\partial \eta} \left( \frac{G_q}{J} (j_{1s} \phi + j_{2s} \phi + j_{3s} \phi) \right) + \frac{1}{J} \frac{\partial}{\partial \zeta} \left( \frac{G_q}{J} (j_{1s} \phi + j_{2s} \phi + j_{3s} \phi) \right) + S(\xi, \eta, \zeta)$$

where $\hat{U}$, $\hat{V}$, $\hat{W}$ are contravariant velocities. These are given as:

$$\hat{U} = j_{1s} U + j_{2s} V + j_{3s} W$$

(24)

$$\hat{V} = j_{1s} U + j_{2s} V + j_{3s} W$$

(25)

$$\hat{W} = j_{1s} U + j_{2s} V + j_{3s} W$$

(26)

where

$$J = \left| \begin{array}{ccc} x_\xi & x_\eta & x_\zeta \\ y_\xi & y_\eta & y_\zeta \\ z_\xi & z_\eta & z_\zeta \end{array} \right|$$

(27)

$$j_{1s} = q_{1s}^2 + q_{2s}^2 + q_{3s}^2$$

(28)

$$j_{2s} = q_{1s} q_{2s} + q_{1s} q_{3s} + q_{2s} q_{3s}$$

(29)

$$j_{3s} = q_{1s} q_{1s} + q_{2s} q_{2s} + q_{3s} q_{3s}$$

(30)

$$j_{2s} = q_{1s}^2 + q_{2s}^2 + q_{3s}^2$$

(31)

$$j_{3s} = q_{1s} q_{2s} + q_{1s} q_{3s} + q_{2s} q_{3s}$$

(32)

$$j_{2s} = q_{1s}^2 + q_{2s}^2 + q_{3s}^2$$

(33)

$$q_{1s} = y_{\xi} x_{\xi} - y_{\eta} x_{\eta}$$

(34)

$$q_{1s} = y_{\zeta} x_{\zeta} - y_{\eta} x_{\eta}$$

(35)

$$q_{1s} = y_{\xi} x_{\xi} - y_{\eta} x_{\eta}$$

(36)

$$q_{2s} = z_{\xi} x_{\xi} - z_{\eta} x_{\eta}$$

(37)

$$q_{2s} = z_{\xi} x_{\xi} - z_{\eta} x_{\eta}$$

(38)
3. NUMERICAL MODELS

3.1 Numerical Method

The system of the equations is solved by using the non-staggered finite-volume difference (FVD) method. Comparing the staggered FVD method with the non-staggered one, the non-staggered FVD has several advantages; it is easier to program and it requires less CPU time. However, it is well known that oscillation of pressure and velocity can appear with non-staggered FVD. Fortunately, Rhie and Chow (1983) successfully solved these oscillatory problems by using the interpolation method where the flux flowing through a control volume surface is linked with the pressures at the neighboring nodes. Keeping these observations in mind, a brief summary of the numerical model used in this study is provided here:

- Grids were generated by using the algebraic grid generation technique developed by Maruszewski and Amano (1992).
- The computational domain is discretized, and both scalar and vector variables are located at the common grid position as opposed to staggered grid arrangements. Here, the method of Rhie and Chow (1983) is adopted for the non-staggered grid formulations.
- The linkage between the continuity and momentum equations is carried out through the SIMPLER method by Van Doormaal and Raithby (1984).

3.2 Boundary Conditions

At the inlet, uniform profiles are provided for velocity, turbulence kinetic energy, and turbulence dissipation rate. At the outflow region, standard continuative conditions are used, i.e., zero gradients of the variables. At the side flow boundaries, periodic-boundary conditions are used as:

\[ \phi(x,y,z) = \phi(x,y - \text{pitch}, z) \]  

(43)

At the wall boundary, both the velocity and turbulence kinetic energy are set to zero. With the standard \( k - e \) model, the dissipation rate was evaluated by using the local turbulence length scale, whereas it is given by the form, \( 2\mu(\partial \phi/\partial x)^2 \) with the L-S model with the L-S model.

3.3 Computational Conditions

Present computations are performed in a flow of stator region as shown in Fig. 1. The flow parameters and airfoil geometries are given as:

- Radial length to the midspan, \( R_{mp} = 68.6 \text{cm} \)
- Ratio of radial length of hub to tip, \( R_{hub}/R_{tip} = 0.8 \)
- Pitch at the midspan, \( \alpha = 19.6 \text{cm} \)
- Angle of attack, \( \alpha = 0^\circ \)
- Axial chord length, \( c = 15 \text{cm} \)
- Inlet flow velocity at the midspan, \( V_m = 23 \text{m/s} \)
- Reynolds number, \( Re = 5.9 \times 10^5 \)

In the present work, 71 x 42 x 42 non-uniform grids are employed. Exploratory computations were also performed with 71 x 32 x 32 grids. The results of the pressure coefficient distribution in the midspan showed a difference of about 5 percent between these two sets of computations. The CPU time on a supercomputer (Power Challenge Array) is about the same for both SKE and L-S models. It takes about ten hours for 2500 iterations.

4. DISCUSSION

Figure 2 shows the pressure coefficient along the surface of the aerofoil in the midspan with the standard \( k - e \) model (SKE) and the Lauder-Sharma model (L-S). It is shown that the computations with both models agree with the experimental data by Dring et al. (1982). Further, the difference in the pressure calculations between these two models are negligibly small (less than 2 percent). Near the leading edge of the blade, however, the computed pressure depicts a high peak and sudden drop which does not appear in the experimental data. Careful examination on this trend indicated that the peak was generated in the computations due to the discontinuity of the grid arrangement in this region. A similar trend
was also observed near the trailing edge of the blade. This type of discontinuity may be corrected by employing different types of grids such as O- or C-grids. However, since this study is intended for the comparison of turbulence models for the near-wall regions, formulations of different grids were not attempted.

\[ C_p = \frac{(P - P_{ref})}{(P_t - P_{ref})} \]

Figure 2. Pressure Coefficient along Blade Surface in Midspan Plane.

- - - - - Experimental data (Dring et al.);
- - - - - SKE model; — LS model.

Figure 3 shows the variations of the pressure at three different planes: hub, midspan, and tip. The pattern of the pressure distribution varies from hub to tip showing higher pressure near the tip than that near the hub. However, differences between the two models (SKE and L-S) are almost negligible.

These differences in the pressure variations in the YZ plane are shown in Fig. 4. In this figure it is clear that near the hub, the low pressure region near the suction side is larger and decreases along the direction from hub to tip. It is also noted that the pressures near the tip and hub are quite different from those in the midspan. An overview of the above mentioned pressure structures is given in Fig. 5.

A comparison of the calculated kinetic energy between the two models is shown in Figs. 6 through 9. Figure 6 shows the turbulence distributions of the four different positions in the midspan. It is shown that in the region between \( x/c = 1.0 \), the turbulence kinetic energy calculated by using two models has a similar distribution. However, behind the trailing edge, since there is no wall effect, the turbulence kinetic energy is damped very quickly in the computations with the L-S model; that is, the turbulence kinetic energy drops down quickly.

Figure 7 presents the three-dimensional effects on the turbulence kinetic energy. In the midspan, the difference mainly appears in the region downstream from the trailing edge of the aerofoil. Along the tip and hub, the wall effects with the L-S model are greater than those with the SKE model. However, near the pressure side along tip and hub, the level of the turbulence kinetic energy is still very low compared with that along the
Figure 5. Pressure Contour in Pressure-Side and Tip Surface. (a) SKE model; (b) L-S model.

Figure 7. Turbulence Kinetic Energy Contour in Hub, Midspan and Tip Planes. (a) SKE model; (b) L-S model.

Figure 6. Turbulence Kinetic Energy Distribution along Blade-to-Blade Direction in Midspan. (a) $x/c_x = 0$; (b) $x/c_x = 1$; (c) $x/c_x = 1.3$.
--- SKE model; --- LS model.
suction side since the flow is accelerated along the blade forcing the turbulence level to be depressed. These behaviors can also be seen in Fig. 8 where the turbulence kinetic energy contours in three different planes (leading edge, middle section, and trailing edge) in three YZ planes. Figure 9 shows the turbulence kinetic energy contours in the pressure side and tip surface. Figure 9. Turbulence Kinetic Energy Contours in Pressure-Side and Tip Surface. (a) SKE model; (b) L-S model.

4. Given the results of this study, it is recommended to keep the correction factor in the L-S model in order to maintain a more stable computational process.

6. ACKNOWLEDGMENTS
The authors are grateful for the financial support from STI for this project. The computations were made on SGI Power Challenge Array at NCSA, Illinois. The fund is provided by NSF under grant NSF CTS960022N. Thanks are also go to Ms. Karen Steldt who prepared the manuscript.

7. REFERENCES
Figure 10. Secondary Velocity ("W") Contour in YZ Plane.
(a) SKE model; (b) L-S model.
