An Investigation of Turbulence Modelling in Transonic Fans Including a Novel Implementation of an Implicit $k - \varepsilon$ Turbulence Model

MARK G. TURNER$^1$ and IAN K. JENNIONS$^1$

GE Aircraft Engines
Cincinnati, Ohio

Abstract

An explicit Navier-Stokes solver has been written with the option of using one of two types of turbulence models. One is the Baldwin-Lomax algebraic model and the other is an implicit $k - \varepsilon$ model which has been coupled with the explicit Navier-Stokes solver in a novel way. This type of coupling, which uses two different solution methods, is unique and combines the overall robustness of the implicit $k - \varepsilon$ solver with the simplicity of the explicit solver. The resulting code has been applied to the solution of the flow in a transonic fan rotor which has been experimentally investigated by Wennerstrom. Five separate solutions, each identical except for the turbulence modelling details, have been obtained and compared with the experimental results. The five different turbulence models run were: the standard Baldwin-Lomax model both with and without wall functions, the Baldwin-Lomax model with modified constants and wall functions, a standard $k - \varepsilon$ model and an extended $k - \varepsilon$ model which accounts for multiple time scales by adding an extra term to the dissipation equation. In general, as the model includes more of the physics, the computed shock position becomes closer to the experimental results.

Nomenclature

$C_o, C_D$ $k - \varepsilon$ constants
$C_1, C_2$ $k - \varepsilon$ constants
C.V. control volume
$f, F^1, F^2$ functions of $y^+$
$M$ Mach number
$R$ added term in extended $k - \varepsilon$ model, or ideal gas constant
$\kappa$ surface of control volume, or entropy

$\nu_x, \nu_y, \nu_z$ absolute velocity components
$V$ absolute velocity magnitude
$V$ absolute velocity vector
$Vol$ volume of control volume
$w_x, w_y, w_z$ relative velocity components
$W$ relative velocity magnitude
$W$ relative velocity vector
$x, y, z$ Cartesian position components, $z$ is axial
$y$ is also a generic distance from a wall
$\gamma$ ratio of specific heats, $\gamma = C_p/C_v$
$\varepsilon$ turbulent dissipation rate
$k$ thermal conductivity
$\tau$ shear stress
$\tau_w$ wall shear stress magnitude
$\tau_w$ wall shear stress vector

Introduction

The fan in a commercial or military turbofan engine is a component critical to the successful attainment of the performance and efficiency targets of the entire engine. As

---

$^1$ Engineer, Computational Turbomachinery Aerodynamics
Member ASME

$^2$ Manager, Computational Turbomachinery Aerodynamics
Member ASME
and vectorizing the entire code on the CRAY YMP. The details of the algorithm will be presented in this paper.

The coupling strategy is very simple. The density, velocity and energy field is updated at each time step in the time marching solver. Using this velocity and density field, the turbulence model and law of the wall are applied with a resulting turbulent viscosity and wall shear stress calculated. For the algebraic Baldwin-Lomax model, the turbulent viscosity is simply calculated; whereas for the $k$-$\epsilon$ model, the finite volume form of the $k$ and $\epsilon$ equations are advanced one iteration, and the turbulent viscosity is calculated from the $k$ and $\epsilon$ values. The turbulent viscosity and shear stress in turn are used by the viscous time marching solver.

A transonic fan rotor which has been experimentally investigated by Art Wennerstrom, formerly at Wright-Patterson AFB, has been used to validate and compare the Navier-Stokes code with five variations of the turbulence models. By using the same Navier-Stokes solver, the same grid and same boundary conditions, the effect of the turbulence model on the quality of the flowfield results can be quantified. Except for investigating the effect of the Baldwin-Lomax constants which other researchers have made a function of pressure gradient, it has been the intent to keep the method general and not "tune" the constants based on the experimental results.

The Wennerstrom fan has also been run as a stage with the stator modeled as axisymmetric source terms. These results along with the results of two other fan geometries are described by Jennions and Turner (1992).

## Background to Turbulence Modelling Efforts

To close the Reynolds averaged Navier-Stokes equations requires a model for the turbulent shear stresses. The Boussinesq hypothesis has been used which says that the effective turbulent shear stress can be related to the strain times the turbulent viscosity.

The Baldwin-Lomax turbulence model (Baldwin and Lomax, 1978) is a two layer algebraic model which determines the turbulent viscosity in an inner and outer layer which are subsequently matched. Dawes (1985), Adamczyk et al. (1989) and Rai (1989) have used this model for turbomachinery applications. Granville (1987) and York and Knight (1985) have investigated the effects of pressure gradients on the predictive capability of such a model, and have concluded that the model constants need to be modified in the presence of a pressure gradient.

The two equation $k$-$\epsilon$ model, described by Launder and Spalding (1974), has been developed with fewer approximations than the Baldwin-Lomax model. The turbulent viscosity is a function of the $k$ and $\epsilon$ values. This model is often associated with a pressure-correction solver, and the current code started with the $k$-$\epsilon$ model in a pressure-correction solver which was modified to fit into the framework of the explicit time-marching solver.

Chen and Kim (1987) describe an extension to the $k$-$\epsilon$ model. Basically an attempt is made to model the physical process of the generation of large eddies which break down into smaller eddies and fine scale eddies which are then
dissipated. This is done by using two time scales of the turbulent kinetic energy spectrum rather than just one, making a few assumptions, and adding an extra term to the dissipation rate equation. This additional term has the effect of decreasing the turbulent viscosity which seems to be overpredicted using the standard model when the mean shear is large.

In order to overcome the extremely fine grids required to accurately model boundary layers down to the wall, both turbulence models use wall functions. This is an option when using the Baldwin-Lomax model. However the k-ε solver is a high Reynolds number model, and the wall functions are always applied. It is a gridding requirement to keep the grid from becoming too fine.

The Navier-Stokes Equations

The compressible Navier-Stokes equations for a single blade row in three dimensions are cast in terms of absolute velocity but solved in a relative non-Newtonian reference frame rotating along with the blade about the z axis with angular velocity Ω. The equations are written in a conservative form in Cartesian coordinates (x,y,z). The equations in differential form are:

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = E + \frac{\partial F_v}{\partial x} + \frac{\partial G_v}{\partial y} + \frac{\partial H_v}{\partial z}, \quad (1)$$

where the solution vector U is given by

$$U = \begin{bmatrix} \rho \\ \rho v_x \\ \rho v_y \\ \rho v_z \\ \rho e \end{bmatrix} \quad (2)$$

in which ρ is the density and e is the total internal energy. The absolute velocity vector V = vyj vzk and i, j, and k are the unit vectors in the x, y, and z directions. E accounts for terms due to rotation of the coordinate system:

$$E = \begin{bmatrix} 0 \\ \Omega \rho v_y \\ -\Omega \rho v_x \\ 0 \\ 0 \end{bmatrix} \quad (3)$$

and F, G, and H are the convective fluxes given by:

$$F = \begin{bmatrix} \rho w_x \\ \rho w_x v_x + p \\ \rho w_x v_y \\ \rho w_x v_z \\ w_x(\rho e + p) + \Omega x p \end{bmatrix}, \quad (4)$$

$$G = \begin{bmatrix} \rho w_y \\ \rho w_y v_x + p \\ \rho w_y v_y \\ \rho w_y v_z \\ w_y(\rho e + p) + \Omega y p \end{bmatrix}, \quad (5)$$

$$H = \begin{bmatrix} \rho w_z \\ \rho w_z v_x + p \\ \rho w_z v_y \\ \rho w_z v_z \\ w_z(\rho e + p) \end{bmatrix}$$

The pressure p is determined through an equation of state. The ideal gas law is assumed:

$$p = (\gamma - 1)(e - V^2/2). \quad (6)$$

The static temperature T is determined by:

$$T = \frac{e - V^2/2}{C_v}, \quad (7)$$

where Cv is the specific heat at constant volume which is a property of the fluid and is a constant if the gas is assumed to be perfect. The relative velocity vector W = Wxj Wyj Wzk is related to the absolute velocity through the following equations:

$$w_x = v_x + \Omega y, \quad w_y = v_y - \Omega z, \quad w_z = v_z. \quad (8)$$

The viscous terms Fv, Gv, and Hv are given assuming no internal energy sources or radiation heat transfer:

$$F_v = \begin{bmatrix} \tau_{xx} \\ \tau_{xy} \\ \tau_{xz} \end{bmatrix}, \quad G_v = \begin{bmatrix} \tau_{yx} \\ \tau_{yy} \\ \tau_{yz} \end{bmatrix}, \quad H_v = \begin{bmatrix} \tau_{zx} \\ \tau_{zy} \\ \tau_{zz} \end{bmatrix}, \quad (9)$$

in which the shear stresses are related to the strains by the following linear relationships:

$$\tau_{xx} = 2\mu \left( \frac{\partial v_x}{\partial x} - \frac{\partial v_y}{\partial y} - \frac{\partial v_z}{\partial z} \right), \quad (10)$$

$$\tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial v_x}{\partial y} + \frac{\partial v_y}{\partial x} \right), \quad (11)$$

where μ is the viscosity. The equations for τyy and τzz are similar to Eq. (10), and the equations for τxx, τyx, τxy, and τvy are similar to Eq. (11). The components of work done due to viscous stresses are:

$$\Pi_x = v_x \tau_{xx} + v_y \tau_{xy} + v_z \tau_{xz}, \quad (12)$$

$$\Pi_y = v_x \tau_{yx} + v_y \tau_{yy} + v_z \tau_{yz}, \quad (13)$$

$$\Pi_z = v_x \tau_{zx} + v_y \tau_{zy} + v_z \tau_{zz}, \quad (14)$$

and q accounts for the heat conduction within the fluid:

$$q_x = -\kappa \frac{\partial T}{\partial x}, \quad q_y = -\kappa \frac{\partial T}{\partial y}, \quad q_z = -\kappa \frac{\partial T}{\partial z}, \quad (15)$$

with the thermal conductivity being given by:

$$\kappa = \frac{\mu C_p}{Pr}, \quad (16)$$

in which Cp is the specific heat at constant pressure and Pr is the Prandtl number.
The unsteady Navier-Stokes equations have been presented. These equations fully describe the flow field including the turbulence, but to solve them numerically requires resolving very small spatial and temporal details. To obtain meaningful results with coarser grids, these equations can be averaged over relatively small time periods to produce the Reynolds average form of these equations. This yields equations with apparent stresses due to the time unsteadiness. Boussinesq introduced an hypothesis which says that this apparent stress can be related to the strain times the turbulent viscosity. An effective viscosity, $\mu_{\text{eff}}$, therefore has two distinct parts:

$$\mu_{\text{eff}} = \mu_i + \mu_t,$$

and similarly:

$$\kappa_{\text{eff}} = C_p\left(\frac{\mu_i}{Pr} + \frac{\mu_t}{Pr_t}\right),$$

where the $l$ and $t$ subscripts denote laminar and turbulent quantities respectively. For air, $Pr = 0.74$ is a property and $Pr_t = 0.9$ has been applied. With $\mu_{\text{eff}}$ and $\kappa_{\text{eff}}$ replacing $\mu$ and $\kappa$, the above Navier-Stokes equations remain unchanged in form. Conceptually these are now the deterministic velocities being calculated rather than instantaneous velocities. The laminar viscosity is modeled by Sutherland's law in which $\mu_l$ is a function of the local static temperature. The turbulent viscosity calculation, which is needed to close this system of equations, is discussed below for two turbulence models.

The Navier-Stokes Numerical Scheme

The equations above would reduce to the Euler equations if $F_v$, $G_v$ and $H_v$ are set to zero. The code and solution method is then the same as described by Holmes and Tong (1985) and Cedar and Holmes (1989), both of which describe a turbomachinery version of the cell centered Runge-Kutta algorithm described by Jameson (1981). Essentially, the equations are integrated about a finite control volume and marched in time using a five-stage Runge-Kutta scheme. This scheme, on which the dissipation is evaluated on the first and second steps only, has dissipative properties that are particularly suited to the multigrid strategy employed to accelerate the solution. Both second and fourth order smoothing are applied with the second order smoothing being tripped by a pressure gradient switch. Because only a steady result is desired, local time stepping is used.

The current algorithm extends this basic strategy. Various Runge-Kutta schemes have been coded to compare the schemes and aid in starting particularly difficult solutions. A two-stage scheme, similar to that of Dawes (1985) where the dissipation is calculated on each step, has proved particularly robust. Residual averaging has been added to enhance stability and robustness. Rather than increase the Courant number ($CFL$), the residual averaging has been used to stabilize solutions using the same $CFL$ number. Essentially, the residuals are smoothed implicitly using an SLOR type strategy. The smoothing parameters used to obtain the results presented here are all unity.

To account for the viscous terms, velocity and temperature gradients are included. A Green's theorem approach is used which is similar to that used by Kallanderis (1987). The volume integral of the gradient can be converted into a surface integral using Green's theorem. The resulting integral can be expressed in terms of face areas and variable values as:

$$\text{Vol} \frac{\partial T}{\partial x} \approx \int_V \int_s T (\mathbf{n} \cdot \mathbf{n}) \, ds \approx \sum C.V.$$

This is applied to a control volume which is offset from the other control volumes as shown in Fig. 1. Appropriate values and averages of these gradients are then used to obtain the viscous fluxes, $F_v$, $G_v$ and $H_v$, about the flux balance control volume.

For a viscous calculation, the no-slip and adiabatic wall boundary conditions are applied. In order to utilize grid points most efficiently, a law of the wall model can be used to evaluate the shear stress and shear work terms at a wall boundary. This model is described below in more detail. At the inlet, the absolute total pressure and temperature are set to zero. The code and solution procedure therefore has two distinct parts:

- For an inviscid calculation, the no-slip and adiabatic wall boundary conditions are applied. In order to utilize grid points most efficiently, a wall model can be used to evaluate the shear stress and shear work terms at a wall boundary. This model is described below in more detail. At the inlet, the absolute total pressure and temperature are set to zero.

- For a viscous calculation, the no-slip and adiabatic wall boundary conditions are applied. In order to utilize grid points most efficiently, a law of the wall model can be used to evaluate the shear stress and shear work terms at a wall boundary. This model is described below in more detail. At the inlet, the absolute total pressure and temperature are set to zero.
The Baldwin-Lomax Turbulence Model

The formulation of the Baldwin-Lomax model is identical to that described by Baldwin and Lomax (1978), but there is much contention as to how it should be coded in practice. Several assumptions, or clarifications of the model which were used are:

- The equations are applied along grid lines rather than normals to solid surfaces. This avoids having to calculate all the normal distances and the interpolation of flow variables.
- Since a law of the wall approach is used in practice, the Van Driest damping terms are dropped.
- There is a limit placed on recrossover in order for it not to occur too far from the wall.
- Beyond the trailing edge, the turbulence model is essentially frozen. Average values of $F_{max}$ and $y_{max}$ are taken from the trailing edge suction and pressure surfaces and applied at downstream locations.
- The downstream running grid line from the blade trailing edge is assumed to be the wake centerline.
- $F_{wake}$ has been formulated to be the smaller of two terms (Baldwin and Lomax, 1978). Only the term $F_{wake} = y_{max} F_{max}$ has been used.
- The turbulent viscosity is limited to 1000 times the laminar viscosity in order not to have spuriously high viscosities in the flow field. The number of nodes which are clipped in this fashion are kept track of in the code.

Granville (1987) has attempted to predict the behavior of two of the constants, $C_{CP}$ and $C_{KLEB}$, for varying pressure gradients. The values used by Baldwin and Lomax are $C_{CP} = 1.6$, and $C_{KLEB} = 0.3$, which lie far outside the curve suggested by Granville. York and Knight (1985) have also suggested different values of these constants in their work. The effects these changes can have on a solution are demonstrated later in the paper.

The $k-\epsilon$ Turbulence Model

The $k-\epsilon$ turbulence model is a two-equation model which determines the turbulent kinetic energy $k$, and a macro length scale of the turbulence $l$ from transport equations. The turbulent kinetic energy is defined as

$$ k = \frac{1}{2} u_i u_i $$

where $u_i$ are the components of the unsteady velocity vector. The length scale can be related to $k$ and the isotropic turbulent dissipation rate $\epsilon$ through the following relation:

$$ l = \frac{C_D k^{3/2}}{\epsilon} $$

where $C_D$ is a constant. The dissipation rate is defined as:

$$ \epsilon = \frac{\mu}{\rho} \frac{\partial u_i}{\partial x_i} \frac{\partial u_i}{\partial x_j} $$

The $k-\epsilon$ model for high Reynolds numbers is described by the following equation (Launder and Spalding, 1974) presented in the conservative differential form of Eq. (1). Subscripts of $k-\epsilon$ will indicate the vectors for these equations:

$$ U_{k-\epsilon} = \left[ \begin{array}{c} \rho k \\ \rho \epsilon \\ \end{array} \right] $$

$$ E_{k-\epsilon} = \left[ \begin{array}{c} \text{Gen} - \rho \epsilon \\ \frac{\epsilon}{k} \text{Gen} - \frac{\epsilon}{\rho} \end{array} \right] $$

$$ F_{k-\epsilon} = \left[ \begin{array}{c} \rho w_x k \\ \rho w_y \epsilon \\ \end{array} \right] $$

$$ G_{k-\epsilon} = \left[ \begin{array}{c} \rho w_x k \\ \rho w_y \epsilon \\ \end{array} \right] $$

$$ H_{k-\epsilon} = \left[ \begin{array}{c} \rho w_x k \\ \rho w_y \epsilon \\ \end{array} \right] $$

$$ F_{v_k\epsilon} = \left[ \begin{array}{c} \mu \frac{\partial w_x}{\partial x} \\ \mu \frac{\partial w_y}{\partial y} \\ \end{array} \right] $$

$$ G_{v_k\epsilon} = \left[ \begin{array}{c} \mu \frac{\partial w_x}{\partial x} \\ \mu \frac{\partial w_y}{\partial y} \\ \end{array} \right] $$

$$ H_{v_k\epsilon} = \left[ \begin{array}{c} \mu \frac{\partial w_x}{\partial x} \\ \mu \frac{\partial w_y}{\partial y} \\ \end{array} \right] $$

In Cartesian coordinates, the generation term, $\text{Gen}$, is:

$$ \text{Gen} = \mu_t \left( \frac{\partial w_x}{\partial x} \right)^2 + \left( \frac{\partial w_y}{\partial y} \right)^2 + \left( \frac{\partial w_z}{\partial z} \right)^2 + \left( \frac{\partial w_x}{\partial y} \right)^2 + \left( \frac{\partial w_x}{\partial z} \right)^2 + \left( \frac{\partial w_y}{\partial x} \right)^2 + \left( \frac{\partial w_y}{\partial z} \right)^2 + \left( \frac{\partial w_z}{\partial x} \right)^2 + \left( \frac{\partial w_z}{\partial y} \right)^2 $$

The turbulent viscosity is

$$ \mu_t = \min \left( \frac{C_{\mu} k^2}{\epsilon} + \mu_{limit} \epsilon \right) $$

where the turbulent viscosity is clipped for numerical stability of the time marching solver. A value of $\mu_{limit} = 1000$ is generally used in practice (for cases in which the inlet $\mu_t$ values would be clipped, larger values of $\mu_{limit}$ are used).

The constants applied are

$$ C_{\mu} = 0.09, \ C_1 = 1.44, \ C_2 = 1.92, \ \sigma_k = 1.0, \ \sigma_\epsilon = 1.217. $$

Extended $k-\epsilon$ model

Chen and Kim (1987) describe an extended turbulence model which adds one term to the $\epsilon$ equation and uses a modified set of constants. This new term is in the source term

$$ E_{k-\epsilon} = \left[ \begin{array}{c} \text{Gen} - \rho \epsilon \\ \frac{\epsilon}{k} \text{Gen} - \frac{\epsilon}{\rho} \end{array} \right] $$

where

$$ R = \min(3, \frac{\text{Gen}}{\rho \epsilon}) $$

$C_3$ is a new constant. The value of this constant and the suggested values of the other constants (Chen, 1991) are:

$$ C_\mu = 0.09, \ C_1 = 1.15, \ C_2 = 1.90, \ C_3 = 0.25, $$

$$ R = \min(3, \frac{\text{Gen}}{\rho \epsilon}) $$

$C_3$ is a new constant. The value of this constant and the suggested values of the other constants (Chen, 1991) are:
\[ \sigma_k = 0.75, \quad \sigma_\varepsilon = 1.05. \quad (32) \]

This extra term represents the energy transfer rate from large scale turbulence to small scale turbulence controlled by the production range time scale and the dissipation rate time scale (two time scales as opposed to one in the standard model). If the generation term is large (i.e. the mean shear is strong), then this extra term will increase \( \varepsilon \) and suppress the overshoot of \( k \) (Chen and Kim, 1987). The resulting turbulent viscosity is also decreased.

**k-\( \varepsilon \) boundary conditions**

At the inlet, \( k \) and \( \varepsilon \) must be prescribed. For the results presented, these are specified as uniform for the entire inlet plane. The inlet turbulence intensity, TURBIN defines the level of turbulent kinetic energy given an average relative velocity, \( W \) as:

\[ k_{\text{inlet}} = \frac{3}{2} (\text{TURBIN} \cdot W)^2, \quad (33) \]

where the factor of 3 is needed because there are three components of velocity. It is assumed that \( k \) is proportional to some characteristic length \( L \) of the problem (the inlet annular height for the fan solution) so

\[ \varepsilon_{\text{inlet}} = \frac{C_D k_{\text{inlet}}^{3/2}}{\lambda L} \quad (34) \]

where \( C_D = 1 \) and \( \lambda = 0.05 \) is the proportionality constant. The values of \( k \) and \( \varepsilon \) at the inlet can be specified to be non-uniform, but how the turbulence intensity and length scales vary is very difficult to predict, and therefore has not been attempted.

The boundary conditions for \( k \) and \( \varepsilon \) at the wall are based on the wall-function method discussed in the next section and more fully explained by Launder and Spalding (1974). A normalized distance, \( y^+ \), is defined by:

\[ y^+ = \frac{y V_{\text{ref}}}{\mu W}, \quad (35) \]

where the \( w \) subscript denotes values at the wall. At the node closest to the wall, the value of \( y^+ \) is determined from

\[ y^+ = \frac{\mu \sqrt{k} C_{\text{u}}^{1/4} \delta y}{\mu_\varepsilon} \quad (36) \]

which is based on \( \tau / \rho = C_{\text{u}}^{1/2} k = \text{const.} \). The value of \( \varepsilon \) is proportional to \( k^{3/2} / y \) so

\[ \varepsilon_{\text{wall}} = \frac{C_{\text{u}}^{3/4} k^{3/2}}{\sqrt{\varepsilon} y}. \quad (37) \]

The value of \( k \) is determined from the transport equation applied at the near wall point. However several modifications of the equation need to be applied. First, there is no flux of \( k \) across the wall. Second, based on Eq. (37) the \( \varepsilon \) term in the \( k \) equation is applied as

\[
\begin{align*}
\int \int \int \rho C_{\text{u}}^{3/4} k^{3/2} C_{\text{u}}^{1/4} \delta y \cdot V \, dV = \begin{cases} 
\rho C_{\text{u}}^{3/4} k^{3/2} C_{\text{u}}^{1/4} \delta y \cdot V \, dV & \text{if } y^+ \leq 11.63 \\
\rho C_{\text{u}}^{3/4} k^{3/2} C_{\text{u}}^{1/4} \ln(E_{\text{u}} \delta y) V \, dV & \text{if } y^+ > 11.63
\end{cases}
\end{align*}
\quad (38)
\]

where the constants \( E_{\text{u}} = 9.793 \) and \( \kappa_{\text{u}} = 0.4187 \) are the coefficient of roughness and the Von Karman constant respectively. Third, the generation term includes the stresses which along the wall are not related to the strains, but are determined from the law of the wall. The generation term for the faces parallel to the wall is

\[ \text{Gen} = \tau_w \frac{\partial W}{\partial y}, \quad (39) \]

where \( \tau_w \) is obtained from the law of the wall.

**k-\( \varepsilon \) discretization approach and solution method**

The k-\( \varepsilon \) model is discretized about the same flux balance control volumes used by the explicit solver (Fig. 1), and the \( k \) and \( \varepsilon \) values are stored at the cell center nodes. The approach is based on writing a general form of the transport equations as is common in a pressure-correction scheme, which involves the local cell center, \( P \), plus the six neighbor cell centers \( N, S, E, W, D, \) and \( U \):

\[ a_p \phi_p = \sum_{i=N,S,E,W,D,U} a_{i} \phi_{i} + (S_p)_{p}. \quad (40) \]

The variable \( \phi \) is either \( k \) or \( \varepsilon \). The term \((S_p)_{p}\) includes the original source term in the equation plus any additional terms which are not made up from the seven grid points. As explained by Patankar (1980), the following condition must be satisfied to ensure stability:

\[ |a_p| \geq \sum_{i=N,S,E,W,D,U} |a_i|. \quad (41) \]

This is achieved by using a hybrid scheme which upwinds the convective terms if the diffusion terms are too small. The convective term is just the mass flux through a face times the average of the cell center values.

Eq. (40) is solved using an ADI scheme on a cross flow plane. A tridiagonal solver couples the blade-to-blade direction and the spanwise direction. The equations are not solved implicitly in the streamwise direction. The equations are solved on a cross flow plane starting at the inlet and marching downstream to the exit.

The diffusion term discretization is based on summing \( \rho C_{\text{u}}^{3/4} \delta y / (\nabla \cdot \vec{u}) \) over the six faces of the control volume. This term can be related to the derivatives along the grid lines as follows:

\[ (\nabla \cdot \vec{u}) \, ds = A_x \frac{\partial \phi}{\partial x} + A_y \frac{\partial \phi}{\partial y} + A_z \frac{\partial \phi}{\partial z} \quad (42) \]

\[ = (A_x \frac{\partial \phi}{\partial x} + A_x \frac{\partial \phi}{\partial y} + A_x \frac{\partial \phi}{\partial z} + A_y \frac{\partial \phi}{\partial y} + A_y \frac{\partial \phi}{\partial z} + A_z \frac{\partial \phi}{\partial z}) \quad (43) \]

\[ + (A_x \frac{\partial \phi}{\partial x} + A_y \frac{\partial \phi}{\partial y} + A_z \frac{\partial \phi}{\partial z} + A_y \frac{\partial \phi}{\partial y} + A_z \frac{\partial \phi}{\partial z} + A_z \frac{\partial \phi}{\partial z}) \quad (44) \]

\[ + (A_x \frac{\partial \phi}{\partial x} + A_y \frac{\partial \phi}{\partial y} + A_z \frac{\partial \phi}{\partial z} + A_z \frac{\partial \phi}{\partial y} + A_z \frac{\partial \phi}{\partial z} + A_z \frac{\partial \phi}{\partial z}) \quad (45) \]

If this face projection is in the \( \xi^1 \) direction as shown by \( A^1 \) in Fig. 2, then the grid derivatives

\[ \frac{\partial \phi}{\partial \xi^1} = \frac{A_x}{V_{ol}} \frac{\partial \phi}{\partial y} \frac{A_x}{V_{ol}} \frac{\partial \phi}{\partial z} \quad (46) \]
The Law of the Wall

The law of the wall model uses empirical data of how the wall shear stress relates to the distance away from the wall. White (1974) shows that this data collapses to a log-linear curve for $y^+ > 30$ when $u^+$ and $y^+$ are plotted. The inner-law velocity variable $u^+$ is defined as:

$$u^+ = \frac{u}{\sqrt{\nu/R_\infty}}$$

and $y^+$ is defined by Eq. (35). By using this $u^+ - y^+$ relation, the grid does not have to be fine enough to resolve the entire boundary layer to get the correct shear stress. The grid does have to be fine enough to ensure the log-linear relation is valid which depends on the pressure gradient. The maximum valid $y^+$ at the wall lies between 150 and 1000.

The force of the shear stress on the wall is applied parallel to the velocity vector and has a magnitude of the shear stress times the area magnitude.

Law of the wall with the Baldwin and Lomax model

The $u^+ - y^+$ relation used with this model is a combination of Spalding's model presented in White (1974) blended with the following log-linear curve:

$$\ln(1.01 + 9 y^+) = 1.0 + 0.435 y^+$$

The cell Reynolds number is

$$Re_c = \left( \frac{\rho u \delta y}{\mu_i} \right)_{c}$$

At the wall $\delta y = y$ so $y^+ = Re_c/u^+$. The $u^+ - y^+$ relation defines $u^+ = f(y^+)$ so the cell Reynolds number

$$Re_c = y^+ u^+ = f(y^+)$$

This relation has been curve fit to give

$$y^+ = f^2(Re_c)$$

where the cell Reynolds number is known. Once $y^+$ is known, $u^+$ is determined from the $u^+ - y^+$ relation and the wall shear stress is known.

Law of the wall with the $k-\epsilon$ model

The law of the wall has already been assumed in prescribing the boundary conditions for the $k$ and $\epsilon$ equations. The $y^+$ values are calculated using Eq. (36), and the $u^+ - y^+$ relation used is:

$$u^+ = \begin{cases} 
    y^+ & \text{if } y^+ \leq 11.63 \\
    \frac{1}{\kappa_{st}} \ln(E_{tow} y^+) & \text{if } y^+ > 11.63
\end{cases}$$

where $E_{tow} = 9.793$ and $\kappa_{st} = 0.4187$. The way this equation is applied is that for $y^+ \leq 11.63$,

$$\tau = \frac{\mu}{\delta y} \frac{dW}{dy}; \quad \tau = \frac{\mu_1}{\delta y}$$

and similar expressions can be written for $(A^2)_3$ and $(A^2)_4$. Terms such as this are made part of the source term $(S_b)_c$ in Eq. (40). The other term for this face and the other faces are calculated similarly.

The use of the projected areas is very similar to using the inverse of the Jacobian matrix to obtain the grid derivatives which is a more common approach (Anderson et al., 1984). It was used here because the areas had already been allocated memory storage for the explicit solver and the inverse of the Jacobian did not need to be calculated.
Figure 3: Meridional view of the grid with a blowup of the tip gap.

For \( y^+ > 11.63 \),

\[
\tau = \mu \frac{dW}{dy}, \tag{59}
\]

From this equation, the equation for viscosity, Eq. (28), and the average dissipation over a control volume, Eq. (38), the shear stress equation is

\[
\tau = \frac{\rho C_{\mu}^{1/4} k^{1/2} \kappa_{lw} k y}{\ln(E_{lw} y^+)} \frac{dW}{dy}. \tag{60}
\]

Therefore,

\[
\frac{\tau}{\rho} = C_{\mu}^{1/4} k^{1/2} \kappa_{lw} \frac{1}{\ln(E_{lw} y^+)}. \tag{61}
\]

Results

The Navier-Stokes solver has been used to solve the flowfield about a transonic fan rotor which was designed at GE Aircraft Engines under a USAF contract and has been experimentally investigated by Wennerstrom at Wright Patterson AFB. This is the fourth rotor in a sequence of experimental fan rotors. Solutions have been obtained using three variations of the Baldwin-Lomax turbulence model and two variations of the \( k-e \) model. Each of the five cases used the same grid, the same smoothing constants, and the same boundary conditions.

The grid used is shown in Figs. 3 and 4. The grid near the walls was spaced to be valid for a turbulence model with wall functions. It has 49 grid points within the blade passage and 37 grid points spanwise (5.855 inch span at the leading edge) of which 4 are within the tip gap (0.025 inch). Upstream of the leading edge, there are 16 axially spaced grid points; there are 65 grid points along the blade surface axially, and there are 16 grid points downstream of the trailing edge for a total of 97 axial grid points. This is a total of 175,861 grid points.

The inlet boundary conditions which have been specified are a uniform absolute total pressure and total temperature. The test inlet total pressure and temperature were 9.85 psi and 545 degrees Rankine respectively. All measurements have been corrected to standard conditions, so the calculation ran with an inlet total pressure of 14.696 psi and an inlet total temperature of 518.688 degrees Rankine. The tip speed is 1500 ft/sec at a tip radius of 8.5 inches. The tangential velocity is zero, and the meridional flow angle, \( \arctan(V_c/V_t) \), was determined from a through-flow calculation. The inlet turbulence intensity for the \( k-e \) solutions have been specified as 2%. At the exit, the static pressure at the casing was specified, and simple radial equilibrium was applied to determine the static pressure at the rest of the span. The pressure at each radial grid line is treated to be uniform tangentially (i.e. \( \frac{dp}{dr} = 0 \)). At the wall, the shear stress is determined from the velocity gradient or from the wall function if the wall functions are used. These shear stresses are then multiplied by the velocity of the wall in the absolute frame to determine the shear work terms of Eqs. (12)-(14). This means there is a shear work contribution from the rotating hub and blade surfaces, but it is zero at the casing.

Solutions have been obtained using the following turbulence models:

1. The Baldwin-Lomax model with standard constants and no wall functions.
2. The Baldwin-Lomax model with standard constants and wall functions.
3. The Baldwin-Lomax model with a set of constants modified for pressure gradient (\( C_{CP} = 1.0 \) and \( C_{KEB} = 0.64 \)) and wall functions. These were chosen to lie on the curve predicted by Granville (1987) in the adverse pressure gradient region.
4. The standard \( k-e \) model.
5. The extended \( k-e \) model.

The general procedure to run these complex solutions is to start using the more robust two-stage Runge-Kutta scheme and continue with the faster five-stage scheme. The
For a very favorable pressure gradient. With no wall functions, the flow is not separated; however it can be as high as 1000 standard deviation should be less than 300. The upper bound of 300 is valid for any pressure gradient as long as the flow is not separated; however it can be as high as 1000 for a very favorable pressure gradient. With no wall functions, the \( y^+ \) values should all be near 1. The values for case 1 are much too large for the solution to be meaningful. It is presented to demonstrate that a quality solution is obtained only when the grid resolution matches the model being used.

Table 1: Calculated Mass Flow Compared to Experiment.

<table>
<thead>
<tr>
<th>case</th>
<th>wall functions</th>
<th>calculated mass flow (difference from experiment)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baldwin-Lomax</td>
<td>no</td>
<td>+3.5%</td>
</tr>
<tr>
<td>standard consts</td>
<td>yes</td>
<td>+2.5%</td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>yes</td>
<td>+2.3%</td>
</tr>
<tr>
<td>standard consts</td>
<td>yes</td>
<td>+1.7%</td>
</tr>
<tr>
<td>Baldwin-Lomax</td>
<td>yes</td>
<td>+1.4%</td>
</tr>
<tr>
<td>modified consts</td>
<td>yes</td>
<td></td>
</tr>
<tr>
<td>k-e standard</td>
<td>yes</td>
<td></td>
</tr>
<tr>
<td>extended k-e</td>
<td>yes</td>
<td></td>
</tr>
</tbody>
</table>

The experimental mass flow is 1.372 kg/sec/blade passage with an accuracy of ±0.8%.

Table 2: \( y^+ \) Statistics for the Standard k-\( \varepsilon \) Model Solution.

<table>
<thead>
<tr>
<th></th>
<th>min</th>
<th>max</th>
<th>avg</th>
<th>stand. dev.</th>
</tr>
</thead>
<tbody>
<tr>
<td>hub</td>
<td>16.5</td>
<td>173.0</td>
<td>84.08</td>
<td>17.67</td>
</tr>
<tr>
<td>tip</td>
<td>16.2</td>
<td>73.10</td>
<td>44.56</td>
<td>10.28</td>
</tr>
<tr>
<td>pressure surface</td>
<td>8.36</td>
<td>384.0</td>
<td>86.17</td>
<td>67.06</td>
</tr>
<tr>
<td>suction surface</td>
<td>8.61</td>
<td>347.0</td>
<td>77.47</td>
<td>59.67</td>
</tr>
</tbody>
</table>
Figure 5: The $y^+$ values on the blade surface at midspan.

The pressure rise is wrong when wall functions are not used. The Baldwin-Lomax solution with standard constants and wall functions predict the pressure rise to be too far aft. Again, the standard $k$-$e$ solution and the Baldwin-Lomax model with modified constants are very similar and close to the test data. The extended $k$-$e$ model moves the shock slightly forward compared with the standard $k$-$e$ model, and a little closer to the test data.

Figs. 6 and 7 show that the modified constants which were chosen for case 3 produce a response which was based on the pressure gradient. However, the constants chosen for this Baldwin-Lomax run were somewhat arbitrary and may not be suitable in other flow situations. Also, it is difficult to generalize and code an algorithm which determines the values of the constants to use. The standard $k$-$e$ model however clearly already contains this pressure gradient effect.

Fig. 8 shows the hub-to-case efficiency profiles calculated at the trailing edge plane compared with experimentally derived data. The measurements were taken downstream of a stator and a data-match has been used to separate out the stator losses. It can be seen that all but the Baldwin-Lomax solution without wall functions are in good mutual agreement and reasonably match the data.

The previous discussion focused mainly on the comparison of the casing static pressures. This is because the fan is being modeled as an isolated rotor, but was run as a stage (a stage simulation has been run by Jennions and Turner (1992)). The efficiency, therefore, has to be calculated assuming a stator loss. Also, the shock position is more sensitive for the different turbulence models. Getting the shock position correct using the experimental downstream static pressure means the flow rate, efficiency and pressure rise should be correct and consistent. The comparison of shock position can therefore be a sensitive measure of the prediction capability of the code. The shock position compared well for the Baldwin-Lomax solution with the modified constants and both $k$-$e$ solutions. The flow rate for both the $k$-$e$ solutions was closer to experiment than any of the Baldwin-Lomax solutions.

Even with the $k$-$e$ models, the flow rates are still high relative to the accuracy of the measurement. There are several physical phenomena which are still missing in these solutions which might explain this. First, the inlet total pressure profile was specified to be flat because the incoming casing boundary layer was not measured. However, even though the experiment was set up to minimize the boundary layer thickness, it would still not be zero. Second, the fan was run as a coupled stage. The assumption that the relative deterministic flow is steady is not 100% valid. This effect could be modeled by adding the terms which Adamczyk (1985) recommends or using a time accurate solver for both the rotor and stator. Third, the turbulence has been assumed to be locally isotropic. With such a complicated flowfield, especially in the tip, this is probably not the case, although no measurements of detailed turbulence have been measured in the tip region of a transonic fan. And fourth, the running tip clearance is not accurately known because it is very small and hard to measure, and the flow rate is very sensitive to the size of the tip clearance modeled.

The contours of ideal Mach number (a normalized static pressure assuming no loss) for the standard $k$-$e$ model solution are shown in Figs. 9 and 10 for the pressure and suction surfaces respectively. These clearly show the calculated shock locations on the blade surfaces.

Contours of an entropy function ($\exp(-\Delta s/R)$) are shown in Fig. 11 at the trailing edge grid surface for the standard $k$-$e$ model solution. Entropy is a much better quantity for observing loss than total pressure because the system is rotating. Values of one indicate no loss. There are large losses associated with the tip clearance represented by the low level of this entropy function (0.60) associated with the tip clearance vortex. The losses are much larger on the suction surface than the pressure surface due to the higher shock losses.

To demonstrate how different the $k$-$e$ and Baldwin-Lomax models are, a contour plot of the quantity $\mu_L/\mu_t$ is shown for the Baldwin-Lomax solution with standard constants and wall functions (Fig. 12) and the standard $k$-$e$ solution (Fig. 13) at mid span. In Fig. 12, the turbulent viscosity goes to zero in the middle of the passage, whereas in the standard $k$-$e$ model solution, the turbulent viscosity at the inlet is 400 times the laminar value based on the specified inlet turbulence intensity and length scale. Fig. 14 shows how the turbulent viscosities are reduced using the extended model.

Concluding Remarks

An explicit Navier-Stokes solver, which has the option of using either a Baldwin-Lomax algebraic turbulence model or an implicit $k$-$\varepsilon$ turbulence model, has been written. Five solutions have been presented for a transonic fan rotor using modifications of these two turbulence models while keeping the grid, smoothing and boundary conditions identical. Several conclusions can be drawn from the current work; these are:

1. Coupling an implicit $k$-$\varepsilon$ solver with the explicit Navier-Stokes solver has proven very successful.
2. Unless the Baldwin-Lomax constants are modified for the pressure gradient, the shock position does not compare as well with experiment as the $k-\varepsilon$ solutions.

3. The flow rate for the $k-\varepsilon$ solutions is closer to experiment than any of the Baldwin-Lomax solutions.

4. An extended $k-\varepsilon$ model, which is a simple modification to the standard $k-\varepsilon$ model, produces a shock position and mass flow rate which agree slightly better with experimental data than even the standard $k-\varepsilon$ model. Further cases need to be run to determine the generality of the extended model.

5. The $k-\varepsilon$ code, which solves two partial differential equations, actually runs faster than the algebraic Baldwin-Lomax code. Coupled with the $k-\varepsilon$ solutions converging better than the Baldwin-Lomax solutions, the overall run times are less.

6. The $k-\varepsilon$ equations are very stiff and the practical success of the model can be attributed to solving these equations implicitly. Using this approach to solve other stiff systems, such as the chemical reaction equations might prove very useful.

Based on these conclusions, designers at GE Aircraft Engines are using this code with the $k-\varepsilon$ turbulence model on a day-to-day basis. Many different turbomachinery components are being analyzed and designed with the help of this code at both design and off design conditions.

Additional solutions using this code with the standard $k-\varepsilon$ model are presented by Jennions and Turner (1992) for two other fan geometries and the fan presented in this paper as a stage. These results show similar agreement of shock position and flow rate with experimental results as those presented in this paper, and rotor operating characteristics are computed which include tip clearance effects.

Acknowledgements

The authors wish to thank Mark Braaten at GE CR & D for first suggesting the idea to use an existing $k-\varepsilon$ solver to couple with the Navier-Stokes code. The help, advice and support of the following people is also greatly appreciated: Chung Shin, Zee Moussa, Roy Smith, Art Wennerstrom, Aspi Wadia, and Chander Prakash.

References


R.D. Cedar and D. G. Holmes. The calculation of the three-dimensional flow through a transonic fan including the effects of blade surface boundary layers, part-span shroud, engine splitter and adjacent blade rows. ASME 89-GT-325, 1989.


Figure 6: The casing static pressure contours comparing five numerical results with experimental Kulite measurements. The numerical contour intervals are 0.5 psi.
Figure 7: Circumferentially averaged static pressures at the casing. Comparison between experiment and five numerical results.

Figure 8: The adiabatic efficiency hub-to-case profile at the trailing edge plane comparing the five numerical results with experimentally derived values.

Figure 9: Ideal Mach number on the pressure surface for the standard k-ε solution. Contour intervals are 0.05.

Figure 10: Ideal Mach number on the suction surface for the standard k-ε solution. Contour intervals are 0.05.
Figure 11: Entropy function contours at the trailing edge grid surface for the standard $k$-$\varepsilon$ solution. Contours are of $\exp(-\Delta s/R)$ with 0.04 intervals.

Figure 12: Contour plot of $\mu_s/\mu_l$ at the midspan of the blade for the Baldwin-Lomax solution using standard constants and wall functions. Contour intervals are 10 in the lower passage and 100 in the upper passage.

Figure 13: Contour plot of $\mu_s/\mu_l$ at the midspan of the blade for the standard $k$-$\varepsilon$ solution. Contour intervals are 100.

Figure 14: Contour plot of $\mu_s/\mu_l$ at the midspan of the blade for the extended $k$-$\varepsilon$ solution. Contour intervals are 100.