Viscous Analysis of Three-Dimensional Turbomachinery Flows on Body Conforming Grids Using an Implicit Solver

K. F. WEBER* and R. A. DELANEY†
Allison Gas Turbine Division
General Motors Corporation
Indianapolis, Indiana

ABSTRACT
A 3-D Navier-Stokes analysis for turbomachinery flows on C- or O-type grids is presented. The analysis is based on the Beam and Warming implicit algorithm for solution of the unsteady Navier-Stokes equations and is derived from an early version of the ARC3D flow code developed at NASA Ames Research Center. The Navier-Stokes equations are written in a Cartesian coordinate system rotating about the z-axis, and then mapped to a general body-fitted coordinate system. All viscous terms are calculated and the turbulence effects are modelled using the Baldwin-Lomax turbulence model. The equations are discretized using finite differences on stacked body-conforming grids. Modifications made to convert ARC3D from external to internal turbomachinery flows and to improve solution accuracy are given in detail. The body-conforming grid construction procedure is also presented. Calculations for several rotor flows have been made, and results of code experimental verification studies are presented. Comparisons of the solutions obtained on the C- and O-type grids are also presented, with particular attention to shock resolution.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a, a₀</td>
<td>speed of sound</td>
</tr>
<tr>
<td>C₀ₑ</td>
<td>specific heat at constant pressure</td>
</tr>
<tr>
<td>e, E₀</td>
<td>total internal energy</td>
</tr>
<tr>
<td>h, h₀</td>
<td>enthalpy</td>
</tr>
<tr>
<td>j, k, l, n</td>
<td>indices for ξ, η, ζ, and r coordinates</td>
</tr>
<tr>
<td>J</td>
<td>Jacobian of the coordinate transformation</td>
</tr>
<tr>
<td>p, P₀</td>
<td>pressure, total pressure</td>
</tr>
<tr>
<td>r, θ, z</td>
<td>cylindrical polar coordinates</td>
</tr>
<tr>
<td>t, T₀</td>
<td>time</td>
</tr>
<tr>
<td>U, V, W</td>
<td>contravariant velocity components in the ξ, η, and ζ directions, respectively</td>
</tr>
<tr>
<td>u, v, w</td>
<td>absolute velocity components in the x, y, and z directions, respectively</td>
</tr>
<tr>
<td>u, v, w, uₙ</td>
<td>velocity component of rotating coordinate system in x direction = Ωy, velocity component of rotating coordinate system in y direction = Ωz, velocity components in the r, θ, z directions in Eq. 18</td>
</tr>
<tr>
<td>x, y, z</td>
<td>Cartesian coordinates</td>
</tr>
<tr>
<td>γ</td>
<td>ratio of specific heats</td>
</tr>
<tr>
<td>∇, ∆</td>
<td>forward and backward difference operators, respectively</td>
</tr>
<tr>
<td>ξ, η, ζ</td>
<td>general curvilinear coordinates</td>
</tr>
<tr>
<td>ρ, ρ₀</td>
<td>density</td>
</tr>
<tr>
<td>ρ₁</td>
<td>reference density (inlet hub)</td>
</tr>
<tr>
<td>τ, τ₀</td>
<td>transformed time</td>
</tr>
<tr>
<td>ω</td>
<td>mass flow in Figure 4</td>
</tr>
<tr>
<td>Ω</td>
<td>rotational speed of the rotor</td>
</tr>
</tbody>
</table>

INTRODUCTION
Over the past 10 years there has been significant progress in the development of numerical fluid flow analyses for turbomachinery blade rows. This progress has moved us closer to the eventual goal of a time-accurate model of the three-dimensional (3-D) flow through a multistage compressor or turbine. Solving the Reynolds-averaged Navier-Stokes equations, with an accurate turbulence or closure model, probably represents the next achievable goal or milestone in this development. Rai (1987) and Whitfield et al., (1987) have reported unsteady 3-D simulations of single stage turbine configurations and counterrotating propellers, respectively, and Gundy-Burlet and Rai(1989) have reported an unsteady 2-D simulation of a two and one-half stage compressor, but a detailed 3-D unsteady simulation of a multistage machine is probably beyond the capability of existing computers and certainly too costly to be used in the design process. In time-detailed calculations of complete compressor and turbine flow fields will be made.

Today, the analyses that are being used for multistage calculations are based on the average-passage equations derived by Adamczyk(1986). Using this approach, Kirtley et al.,(1990) have been successful in the development and application of an average-passage multistage analysis to the study of secondary flow in a two-stage turbine. The analysis does not seek to provide unsteady solutions of each blade row, but it does account for the effect of upstream and downstream stages. This approach can be very costly for more than one stage, but it does give insight into the flow phenomena in multistage turbomachinery. The accuracy of this analysis method will be determined by future experiments and advanced analyses that directly account for more of the physics.

The 3-D analysis methods that will impact the day-to-day design of compressor and turbine stages for the next few years are the viscous, 3-D, isolated blade-row and single stage analyses, based on the Reynolds-averaged Navier-Stokes equations. The isolated blade-row analyses have been developed for several years, give accurate solutions, and are efficient enough to allow designers to use them. These analyses do not account for upstream and downstream blade rows, but can be used as a starting point for an interactive stage code. The ability to extend the method to unsteady
flows is important because blade row interactions have been shown to be significant (Rao and Delaney, 1990).

The method of solution chosen for the turbomachinery analysis presented here is based on the implicit approximate factorization algorithm of Beam and Warming (1976). An implicit scheme was chosen over an explicit scheme because for steady-state calculations, implicit methods have less stringent stability criteria, allowing larger time steps to be used to speed up the convergence to the steady state. The CFL stability criteria for explicit methods can become very limiting for the Navier-Stokes equations because of the need to resolve the wall region on highly clustered grids. Accurate shock resolution, requiring small mesh spacing, has also become increasingly important in advanced high Mach number compressors, again making implicit techniques advantageous. Recent work by Merriam (1987,1989) shows that using implicit time advancement reduces the problem of satisfying a fully discrete cell entropy inequality to satisfying a semi-discrete entropy inequality, and that schemes which satisfy a discrete entropy inequality may need to be implicit to achieve second-order accuracy in space.

The choice of coordinate system type is just as important to solution accuracy as the choice of flow solver. In the last few years, there has been considerable progress in the development of body-conforming grids for turbomachinery flow calculations. Compared to the sheared H grids, the body-conforming grids resolve the leading edge flows better without special procedures and without being forced to use points inefficiently in other areas of the solution domain. (Much of the problem with H grids is due to the singular Jacobian at the leading edge branch point.) Increasing accuracy and minimizing error at the leading edge is very important because any errors incurred at the leading edge are convected downstream and adversely affect the solution accuracy over the entire airfoil surface.

This paper describes an efficient 3-D turbomachinery flow analysis that incorporates an approximate factorization scheme based on the Beam and Warming algorithm, and body conforming O- and C-type grids. The method is based on the numerics in the ARCAD code developed at the NASA Ames Research Center. Included in the paper are descriptions of the governing equations for a rotating Cartesian system, the grid generation scheme, and modifications made to negate the metric invariant error. Numerical solutions comparing O- and C-grid shock resolution are presented. Viscous solutions for two compressor rotors are presented and compared with experimental data to demonstrate the predictive capability of the analysis.

GOVERNING EQUATIONS

The differential equations solved in this study are the Reynolds-averaged Navier-Stokes equations for a compressible fluid. The equations are written in a Cartesian (x,y,z) coordinate system rotating with angular velocity \( \Omega \) about the z-axis. The rotation introduces acceleration terms in the \( x \)- and \( y \)-momentum equations. If absolute Cartesian velocity components are retained as dependent variables in a system attached to a rotating or stationary blade row, the 3-D unsteady Navier-Stokes equations can be transformed to an arbitrary curvilinear coordinate system \( \xi, \eta, \zeta \) while retaining strong conservation form; see Vivian (1974) or Vinokur (1974). All of the viscous terms are retained, and the resulting equations are written as follows:

\[
\frac{\partial Q}{\partial t} + \frac{\partial (E + E_1)}{\partial \xi} + \frac{\partial (F + F_1)}{\partial \eta} + \frac{\partial (G + G_1)}{\partial \zeta} = H
\]

where

\[
Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{bmatrix},
E = J^{-1} \begin{bmatrix} \rho U \\ \rho u U + \rho \xi \phi \\ \rho v U + \rho \eta \phi \\ \rho w U + \rho \zeta \phi \\ \rho (U - \rho \Phi) \end{bmatrix},
F = J^{-1} \begin{bmatrix} \rho V \\ \rho u V + \rho \xi \rho V + \rho \eta \phi \\ \rho v V + \rho \eta \phi \\ \rho w V + \rho \zeta \phi \\ \rho (V - \rho \Phi) \end{bmatrix},
G = J^{-1} \begin{bmatrix} \rho W \\ \rho u W + \rho \xi \phi \\ \rho v W + \rho \eta \phi \\ \rho w W + \rho \zeta \phi \\ \rho (W - \rho \Phi) \end{bmatrix},
H = J^{-1} \begin{bmatrix} 0 \\ -\rho \xi \Omega \\ -\rho \eta \Omega \\ -\rho \zeta \Omega \\ 0 \end{bmatrix},
\]

\( e \) is the total energy, and \( U, V, \) and \( W \) are the contravariant velocity components in the \( \xi, \eta, \) and \( \zeta \) directions, and \( \tau = t \). The contravariant velocity components, written without metric normalization, are given by

\[
U = \xi (u - u_\tau) + \xi (v - v_\tau) + \xi w
\]

\[
V = \eta (u - u_\tau) + \eta (v - v_\tau) + \eta w
\]

\[
W = \zeta (u - u_\tau) + \zeta (v - v_\tau) + \zeta w
\]

where \( u - u_\tau \) and \( v - v_\tau \) are the relative velocity components in the \( x \) and \( y \) coordinate directions, and \( u_\tau = \Omega \xi \eta \) and \( v_\tau = -\Omega \xi \eta \) are the velocity components of the rotating blade row. The inverse Jacobian of the transformation, \( J^{-1} \) in Eq.(1), is defined as

\[
J^{-1} = \begin{bmatrix} \xi & \eta & \zeta \\ \xi & \eta & \zeta \\ \xi & \eta & \zeta \end{bmatrix}
\]

The viscous flux terms, \( E_\tau, F_\tau, \) and \( G_\tau \) in Eq.(1) are given by

\[
E_\tau = J^{-1} \begin{bmatrix} \xi \tau_{xx} + \eta \tau_{yx} + \zeta \tau_{zx} \\ \eta \tau_{xy} + \xi \tau_{yy} + \zeta \tau_{zy} \\ \zeta \tau_{zy} + \xi \tau_{zy} + \eta \tau_{zz} \end{bmatrix}
\]

\[
F_\tau = J^{-1} \begin{bmatrix} \eta \tau_{xx} + \xi \tau_{yx} + \zeta \tau_{zx} \\ \xi \tau_{xy} + \eta \tau_{yy} + \zeta \tau_{zy} \\ \zeta \tau_{zy} + \eta \tau_{zy} + \xi \tau_{zz} \end{bmatrix}
\]

\[
G_\tau = J^{-1} \begin{bmatrix} \zeta \tau_{xx} + \eta \tau_{yx} + \xi \tau_{zx} \\ \eta \tau_{zy} + \zeta \tau_{zy} + \xi \tau_{zy} \\ \xi \tau_{zy} + \eta \tau_{zy} + \zeta \tau_{zz} \end{bmatrix}
\]

where

\[
\tau_{xx} = \lambda (u_x + v_y + w_z) + 2\mu u_{xx}
\]

\[
\tau_{xy} = \tau_{yx} = \mu (u_x + v_y)
\]

\[
\tau_{xz} = \tau_{zx} = \mu (u_x + w_z)
\]

\[
\tau_{yz} = \lambda (u_x + v_y + w_z) + 2\mu v_y
\]

\[
\tau_{zv} = \tau_{yz} = \mu (u_x + v_y)
\]

\[
\tau_{zz} = \lambda (u_x + v_y + w_z) + 2\mu w_z
\]

\[
\beta_x = \gamma \kappa \rho^{\tau - 1} \xi \tau_{xx} + \eta \tau_{yx} + \zeta \tau_{zx} + v_{rxx} + w_{rxy} + \tau_{rxx}\
\]

\[
\beta_y = \gamma \kappa \rho^{\tau - 1} \eta \tau_{yy} + \xi \tau_{xy} + \zeta \tau_{zy} + v_{ryy} + w_{ryy} + \tau_{ryy}\
\]

\[
\beta_z = \gamma \kappa \rho^{\tau - 1} \zeta \tau_{zz} + \xi \tau_{xz} + \eta \tau_{zy} + v_{rzz} + w_{rz} + \tau_{rzz}\
\]

\[
e = e_0 - 5(u^2 + v^2 + w^2)
\]

and where \( \text{Re} \) is the Reynolds number, \( \Pr \) is the Prandtl number, \( \kappa \) is the coefficient of thermal conductivity, \( \mu \) is the dynamic viscosity, and (using Stokes' hypothesis) \( \lambda = -\frac{2\mu}{\kappa} \). For simplicity, the viscous flux terms are given in terms of their Cartesian derivatives. These terms are expanded in the \( \xi, \eta, \zeta \) space via the chain rule to give relations such as

\[
u_{xx} = \xi \nu_{xx} + \eta \nu_{xy} + \zeta \nu_{xz}
\]

Similarly, the metric terms are obtained from chain rule expansion of \( \xi, \eta, \zeta \), etc. and solved for \( \xi, \eta, \zeta \), etc., to give

\[
\xi = J(\theta_{\xi} - \xi \tau_{\xi})
\]

\[
\eta = J(\theta_{\eta} - \xi \tau_{\eta})
\]

\[
\zeta = J(\theta_{\zeta} - \xi \tau_{\zeta})
\]

\[
\nu_{xx} = \xi \nu_{xx} + \eta \nu_{xy} + \zeta \nu_{xz}
\]

\[
\nu_{yy} = \xi \nu_{yy} + \eta \nu_{yy} + \zeta \nu_{yz}
\]

\[
\nu_{zz} = \xi \nu_{zz} + \eta \nu_{zy} + \zeta \nu_{zz}
\]

where \( J = J(\theta_{\xi} - \xi \tau_{\xi}) \)

\[
\eta = J(\theta_{\eta} - \xi \tau_{\eta})
\]

\[
\zeta = J(\theta_{\zeta} - \xi \tau_{\zeta})
\]

\[
\nu_{xx} = \xi \nu_{xx} + \eta \nu_{xy} + \zeta \nu_{xz}
\]

\[
\nu_{yy} = \xi \nu_{yy} + \eta \nu_{yy} + \zeta \nu_{yz}
\]

\[
\nu_{zz} = \xi \nu_{zz} + \eta \nu_{zy} + \zeta \nu_{zz}
\]

(8)
The Cartesian velocity components \(u, v, \) and \(w\) are nondimensionalized with respect to the speed of sound on the inlet of the hub surface, \(a_1\); density, \(\rho\), is referenced to the hub inlet density, \(\rho_1\); and the energy and pressure to \(\rho_1 a_1^2\). From the definition of total energy, \(e\), pressure is expressed as

\[
p = (\gamma - 1) \left[ e - \frac{5}{2} \left( u^2 + v^2 + w^2 \right) \right]
\]

where \(\gamma\) is the ratio of specific heats.

**COORDINATE SYSTEM**

The coordinate systems used in the 3-D analysis are C- and O-type body-conforming systems. Both of these grids naturally provide high resolution of the leading edge region to accurately capture bow shocks and minimize errors that are convected downstream. The construction procedure for both grid systems is similar. As shown in the O grid construction in Figure 1, these systems are constructed by radially stacking 2-D body-conforming grids on surfaces of revolution.

The grid generator models the geometry of compressor blade rows in the following way. The boundary of the physical passage domain is defined by the hub and shroud endwalls and the inlet and exit boundaries as shown in Figure 2. The location and shape of the inlet and exit boundaries may be defined as planar surfaces, but are generally constructed to follow the curved contours of the leading and trailing edges of the blade, respectively. The distances between the blade leading edge and inlet boundary and the blade trailing edge and exit boundary are specified as percentages of the blade chord at a given radial location. The 2-D blade-to-blade surfaces between the hub and shroud are surfaces of revolution. In the grid generation process, a mean radius is calculated for each surface, and the surface is projected onto a cylinder of that radius. Blade-conforming 2-D grids are generated on each surface by solving an elliptic system of partial differential equations (Thompson et al., 1985), and the grids are then projected back onto the surfaces of revolution.

The 2-D elliptic generator incorporates functions to control orthogonality. Grid point locations on the blade surface are determined in part by imposing orthogonality at the surface. All of these steps are performed interactively on a workstation.

**NUMERICAL SCHEME**

**Algorithm Development**

The implicit approximate-factorisation finite-difference scheme used to solve Eq. (1) was developed by Beam and Warming (1976) and was used initially by Steger (1977) and subsequently by Pulliam and Steger. Explicit and implicit dissipation terms were added to attain nonlinear stability, and a spatially variable time step was used to accelerate convergence for steady-state calculations. The diagonal form of the algorithm was used because it allows for the use of fourth-order implicit dissipation and produces a robust and computationally efficient scheme. The development of the method of solution and algorithm are given in detail in papers by Beam and Warming (1976), Steger (1977), Pulliam (1984), and Pulliam and Steger (1985). A brief description is given here.

By first applying implicit time differencing to the inviscid flux terms, and then local time linearisation and approximate factorization as shown by Pulliam (1984), Eq. (1) can be written as

\[
(I + h \delta_t A^n)(I + h \delta_t B^n)(I + h \delta_t C^n)\Delta Q^n = \text{Explicit RHS}
\]

Eq. 10 consists of an implicit (left) side and an explicit (right) side. The spatial derivative terms are approximated with second-order central differences, which yields a left side that has three implicit operators each of which is block tridiagonal. The computational work involved in evaluating Eq. 10 can be reduced by diagonalizing the blocks in the implicit operators as originally shown by Pulliam and Chasseur (1981). The eigensystem of the flux Jacobians \(A, B,\) and \(C\) are used in this development. Because the flux Jacobians have real eigenvalues and a complete set of eigenvectors, the matrices can be diagonalized (Warming et al., 1975; Turkel, 1973), i.e.,

\[
A\tau = T\tau^{-1}AT\tau, \quad B\tau = T\tau^{-1}BT\tau, \quad C\tau = T\tau^{-1}CT\tau \tag{11}
\]

where \(T, T\tau,\) and \(T\tau\) are the matrices whose columns are the eigenvectors of \(A, B,\) and \(C\). Replacing \(A, B,\) and \(C\) in Eq. 11 by their eigensystem decomposition yields

\[
\left[ T\tau T\tau^{-1} + h\delta_t (T\tau A\tau T\tau^{-1}) \right] \left[ T\tau T\tau^{-1} + h\delta_t (T\tau A\tau T\tau^{-1}) \right] \Delta Q^n = \text{Explicit RHS of Eq. 10} = R^n \tag{12}
\]
A modified version of Eq. 12 can be obtained by factoring the $T_\text{eig}$ and $T_\text{q}$ and the blade, are solid surfaces. The other three boundaries are the inlet, exit, and periodic flow boundaries.

At the inlet, an extension of the 2-D procedure used by Chima (1985) that allows for the specification of radial profiles of total temperature, total pressure, and the radial and tangential velocity components is used. The procedure is based on a method of characteristics formulation similar to that used by Jameson and Baker (1983) in which the upstream-running Riemann Invariant $R^*$, based on the total velocity $q$, is extrapolated from the interior to the boundary, i.e.,

$$R^* = \left[ q - \frac{a_0}{\gamma - 1} \right]$$

(16)

where $a_0 = \sqrt{\gamma p}/\rho$ is the speed of sound. For specified total temperature, $T_\text{eig}$, the inertial relations are then used to determine $\mathbf{q}_\text{m}$, the magnitude of the velocity.

$$\mathbf{q}_\text{m} = \sqrt{\gamma - 1} R^* + \sqrt{4(\gamma + 1) C_p T_0 - 2(\gamma - 1)(R^*)^2}/(\gamma + 1)$$

(17)

The axial velocity component is found from trigonometric relations, and pressure and density are determined using inertial relations.

At the exit boundary points, the static pressure is specified, and $\rho$, $u$, $v$, and $w$ are extrapolated from the inner field. The pressure at the hub or shroud is specified and the radial pressure distribution is determined using a simplified version of the radial equilibrium equation, i.e.,

$$\frac{\partial p}{\partial r} = \rho \omega^2 \frac{r}{r_\text{h}}$$

(18)

This equation is integrated at each time step following the extrapolation of the other flow variables. The calculated pressure distribution is then imposed as a boundary condition at the next step. Initially, if the difference between the exit pressure and the inner field pressure is large, the nonreflective procedure of Rudy and Strikwerda (1980) can be used. The pressure is imposed asymptotically, so that there is not a sharp discontinuity in pressure at the exit.

On the hub and shroud endwalls and the blade surface, solid boundary conditions are specified. To the extent that the flow is tangentially tangent to the blading, the normal profile pressure is imposed asymptotically, so that there is not a sharp discontinuity in pressure at the exit.

Pressure on the solid surfaces is found using the normal momentum equation, which is a combination of the three transformed momentum equations given by

$$\rho_\text{u} (\mathbf{g}_\text{u} + \mathbf{g}_\text{v} + \mathbf{g}_\text{w}) = \rho_\text{u} \left[ \mathbf{u}_\text{v} \mathbf{g}_\text{v} + \mathbf{u}_\text{w} \mathbf{g}_\text{w} \right] + \rho \left[ \mathbf{g}_\text{u}, \mathbf{g}_\text{v} + \mathbf{g}_\text{w} \right]$$

(19)

where $\mathbf{n}$ is the direction normal to the $\zeta$ = constant solid surface. In this form, the boundary conditions are applicable to steady or unsteady flow. The density is found using a boundary condition suggested by Barton and Pulliam (1984) and Chima (1985) in which the density, expressed as $S = \rho/\rho^*$, is extrapolated to the body. This condition is very stable, and it keeps the surface and inner field entropy values at or above the inlet entropy, ensuring that the second law of thermodynamics is not violated by decreases in entropy.

**NONLINEAR ARTIFICIAL DISSIPATION MODEL**

One of the important goals in the numerical study of compressor aerodynamics is the ability to capture shocks and predict shock losses. Generally, artificial dissipation terms are added to the numerical scheme for this purpose. MacCormack and Baldwin (1975) used a second-difference dis-
conform to the shape of the leading edge. But the O-grid can also resolve increasing stagger. Both grids resolve the leading edge flow because they

for high stagger blade sections, whereas the C-grid becomes sheared with

or not one grid type offered an advantage over the other. For high Mach

4

F direction is written

analysis is the combined second- and fourth-order model first proposed by

Jameson, et al. (1981). The model expressed in simplified notation for the

\( \xi \) direction is written

\[
\nabla \left( \sigma \frac{\partial J^2}{\partial x^2} + \sigma \frac{\partial J^3}{\partial x^3} \right) \left( \omega \frac{\partial Q_j}{\partial x} - \frac{\partial Q_j}{\partial x} \right) \nabla \left( \omega \frac{\partial Q_j}{\partial x} \right)
\]

with

\[
\begin{align*}
\epsilon_2^{(2)} &= K_2 \Delta t \max(T_{y_{-1}}, T_0, T_{y_{+1}}), \quad T_j = \frac{p_{y_{j+1}} - 2p_j + p_{y_{j-1}}}{p_{j+1} + 2p_j + p_{j-1}} \\
\epsilon_4^{(4)} &= \max(0, K_4 \Delta t - \epsilon_2^{(2)}) \\
\sigma_r &= \frac{\|V\| + \alpha \sqrt{\epsilon_2^2 + \epsilon_4^2 + \epsilon_2^2} + \|V\| + \alpha \sqrt{n_1^2 + n_2^2 + n_2^2} + \|W\| + \alpha \sqrt{\epsilon_2^2 + \epsilon_4^2 + \epsilon_2^2}}
\end{align*}
\]

which is the sum of the spectral radii of the flux Jacobians \( A, B, \) and \( C. \)

For simplicity, only the \( j \) subscripts, which correspond to the \( \xi \) direction, have been presented. The suggested values for the constants are \( K_2 = 1/4 \) and \( K_4 = 1/100. \) The first term is a second-order dissipation model with an extra pressure gradient dependent coefficient to increase its value near shocks. The second term is a fourth-order dissipation model in which the coefficient \( \epsilon_4^{(4)} \) becomes zero when the second-order nonlinear coefficient is larger than the constant fourth-order coefficient. This occurs very near a shock. Near computational boundaries, the fourth order term is modified to maintain a dissipative term. Derivations and comparisons of various boundary treatments for dissipative models are given by Pulliam (1986b).

RESULTS and DISCUSSION

Viscous numerical solution results for two compressor rotors are presented and compared with experimental data. A comparison of shock resolution on C- and O-type grids is also presented and discussed. All solutions were started with the inner field values set equal to the experimental or design inlet values. The static pressure at the exit was adjusted, usually by raising the value at the hub, until the desired flow condition was reached. The final values of the damping coefficients for the viscous solutions were \( K_2 = 1/5 \) and \( K_4 = 1/50, \) which are close to the suggested (Pulliam; 1985a, 1986b) values of \( K_2 = 1/4 \) and \( K_4 = 1/100. \) The solutions were assumed to be converged when the root-mean-square average of the explicit residuals had been reduced more than three orders of magnitude. This is a rule of thumb, and additional convergence criteria were used in some cases. The solutions converged in approximately one thousand iterations; additional iterations may have been required to reach a prescribed mass flow or total pressure ratio.

Comparison of O and C Grid Solutions

One of the reasons for the development of both O- and C-grid codes was to compare the solutions calculated on both grids and determine whether or not one grid type offered an advantage over the other. For high Mach number compressor calculations, the O-grid solutions provide superior shock resolution over O-grid solutions. This is clearly seen in comparisons of shock structures for transonic rotors such as NASA fan Rotor 67, shown in Figure 1. The rotor was tested at the NASA Lewis Research Center, and initially reported by Pieraga and Wood (1984). The rotor has 22 low aspect ratio (1.28) blades rotating at 16,042 rpm with a relative tip Mach number of 1.38 at the design speed of 1407.2 ft/sec and flow rate of 73.3 lb/sec. Figure 3 compares the shock structure at 90% span obtained from 3-D O- and C-grid solutions of Rotor 67. The O-grid solution exhibits much sharper leading-edge bow and passage normal shocks than the C-grid solution. The shock resolution on the O-grid is superior because the grid is orthogonal, even for high stagger blade sections, whereas the C-grid becomes sheared with increasing stagger. Both grids resolve the leading edge flow because they conform to the shape of the leading edge. But the O-grid can also resolve the trailing edge flow because it conforms to the entire blade shape including the trailing edge.

Code Validation: NASA Rotor 67

The capability of the analysis to predict flows in rotating blade rows was established and the solution accuracy verified by comparing predicted results with experimental data for NASA fan Rotor 67. As Figure 1 reveals, this rotor contains a large amount of twist from hub to tip, producing a highly three-dimensional flow field. The viscous O-grid solver was used to simulate the flow in Rotor 67 at the peak efficiency and near-stall operating points. For these calculations, the blade and hub were treated as viscous surfaces while the shroud was treated as an inviscid surface. The grid used for the predictions had 199 normals and 31 contours on 61 blade section. The grid contours were clustered near the blade surface so that the first points off the blade had a \( y^+ \) value less than five. This was approximately 0.0002 of hub axial chord. The ability of the analysis to predict overall blade row performance at each operating point was determined by comparing the predicted and measured radial distributions of inlet and exit data. The capability to predict details of the flow field was also demonstrated through comparison of predicted and measured Mach number contours within the blade row. The experimental data for these comparisons can be found in the final report by Strazisar et al. (1986).

Figure 4 shows a comparison of the radial distribution of predicted and measured total pressure and temperature ratios across the rotor at the peak efficiency and near stall operating points. The solutions were run to match the spanwise energy-averaged total pressure ratio at each operating point. Following the procedure used in the tests, the total pressure was energy-averaged and the total temperature was mass-averaged. For the near stall operating point, the prediction was made at a slightly lower pressure ratio than the experimental value because higher values of exit pressure caused the solution to become unstable. The good agreement between the predic-
Fig. 4 Comparison of predicted and measured exit survey data for NASA Rotor 67 at peak-efficiency and near-stall conditions.

Fig. 5 Comparison of predicted and measured adiabatic efficiency at the test locations of the nine design streamlines.

tion and the data shown in Figure 4 demonstrates the ability of the analysis to match the distribution once that the overall average has been set.

Figure 5 gives a comparison of measured and predicted adiabatic efficiency for the peak efficiency and near stall operating points. The spanwise locations of the data points were taken from the final report by Strazisar et al. They are the locations of the nine exit survey stations which correspond to the design streamlines. Calculated values were interpolated to these locations in order to make a direct comparison. For the peak efficiency case, the code underpredicted efficiency in the middle span regions with a maximum difference of 4.60% at 70 percent span. For the near stall case, the comparisons are generally closer with a maximum difference of 3.59% occurring at about 60 percent span.
Figure 6 shows the contour plots of measured and predicted relative Mach numbers for the peak efficiency operating point at 30, 70, and 90 percent span, measured from the hub. Figure 7 compares the measured and predicted relative Mach numbers for the near-stall operating point at the same spanwise locations. For a transonic compressor rotor, these comparisons are considered to be fairly good. The comparisons for the near-stall condition are good at each spanwise location. At the peak-efficiency condition fairly good agreement is shown at 30 and 70 percent span, but the agreement at 90 percent span is not good. The simulation does not predict the shock location very well. Although the exact shock location for the test case is difficult to determine, the measured contours clearly show a more oblique bow shock for the peak-efficiency case at 90 percent span. This is consistent with the higher measured mass flow. There is also a mild separation region downstream of the shock/boundary layer interaction in the simulation that may or may not have existed in the test. This is currently a subject of debate. If the objective of the simulation had been to match mass flow, the shock location comparison would be better, but pressure and temperature rise comparisons would not be as good. It is possible that a complete match cannot be obtained without the inclusion of a tip clearance model and an accurate representation of the machine geometry while running.

A fully viscous solution for NASA Rotor 67 without tip clearance has also been calculated on the 199x31x61 grid. For this solution, the blade, hub, and the shroud were all treated as viscous surfaces. Figure 8 shows a blade surface oil flow simulation picture for the rotor at the near stall operating point. On the pressure side of the blade, there is no flow separation and very little radial flow migration. On the suction side of the blade, downstream of the shock/boundary layer interaction where the flow separates or loses almost all axial momentum, there is almost complete outward radial migration of the flow near the blade surface. On the pressure surface near the tip, there is a trace pattern that indicates vortex rollup at about 90% span. This example demonstrates the capability of the analysis to predict complex viscous flow phenomena.

**Advanced Concept Compressor Rotor**

The viscous solver was also used to study new design concepts for the first rotor of an advanced multistage compressor. The compressor rotor, whose
flow path is shown in Figure 9, was designed and tested at the Allison Gas Turbine Division. The rotor has 16 blades rotating at 19,622 rpm and a relative tip Mach number of 1.67. This rotor was a challenging test case because of the highly three-dimensional geometry. Figure 9 also shows a schematic representation of the grid on the blade and hub surfaces. The actual grid used for the predictions had 191 normals and 21 contours on 19 blade sections. The blade surface was treated as a viscous surface, but the endwalls were treated as inviscid surfaces because of the computer storage limitations.

Figure 10 shows a comparison between the predicted and measured span-wise distribution of normalized total pressure ratio across the rotor at the design operating point. The predictions were obtained by adjusting the hub exit static pressure until the total pressure ratio, mass averaged over the span of the passage, equaled the design total pressure ratio. This was significantly higher than the total pressure ratio for Rotor 67. The agreement from zero to eighty percent span is fairly good, particularly near the hub. The agreement over the last twenty percent span is not good. Some of the discrepancy very near the tip (95-100% span) can be attributed to the lack of a tip clearance model. The predicted total pressure ratio distributions across the rotor passage for the pre-spill and spill points are also given in Figure 10. The spill point was defined as the point at which the rotor could no longer maintain the choking mass flow. At this point, the rotor reaches its maximum total pressure ratio, and any increase in exit pressure pushes the shock system out of the blade passage (beginning at the tip) and produces an unstable stall condition.

**SUMMARY**

A 3-D viscous turbomachinery flow analysis method has been presented. The method, which is based on the implicit approximate-factorization finite-difference scheme of Beam and Warming, combines Pulliam's diagonal form of the algorithm for solution of the 3-D time-dependent Navier-Stokes equations with body conforming C- or O-type grids. Numerical solution results for two 3-D compressor rotors have been presented. The solution for NASA Rotor 67 and the first rotor of an advanced compressor are compared with experimental data to demonstrate the ability of the analysis to predict complex turbomachinery flow fields, and overall blade row performance. Calculated blade-to-blade Mach number contours presented for Rotor 67 at two
Fig. 8 Oil flow simulation for NASA Rotor 67 at the near-stall operating point.

Fig. 9 Flow path and surface grid for the first rotor of an advanced concept compressor.

Fig. 10 Comparison between predicted and measured spanwise total pressure ratio distributions downstream of the first rotor of an advanced concept compressor.

Operating conditions show good agreement with those derived from LDV measurements. A comparison of the predicted shock structures on C- and O-grids has also been presented to show the superior predictive capability achievable with O-type grids.

Acknowledgement

The authors wish to acknowledge the substantial time and effort given by Dr. Dale Thoe for his assistance in generating the 3-D grids and for his technical assistance. Permission to publish this paper by Allison Gas Turbine Division, General Motors Corporation, is gratefully acknowledged. The computer resources provided by the NASA Ames Research Center under the NASA Numerical Aerodynamic Simulator (NAS) program are also acknowledged.

REFERENCES


