Analysis of Three-Dimensional Turbomachinery Flows on C-Type Grids Using an Implicit Euler Solver

K. F. WEBER, D. W. THOE, and R. A. DELANEY
Allison Gas Turbine Division
General Motors Corporation
P.O. Box 420
Indianapolis, IN 46206-0420

ABSTRACT

A 3-D Euler analysis for turbomachinery flows on a C-type grid is presented. The analysis is based on the Beam and Warming implicit algorithm for solution of the unsteady Euler equations and is derived from the ARC3D code developed by Pulliam at NASA Ames Research Center. Modifications made to convert this code from external flow applications to internal turbomachinery flows are given in detail. These changes include the addition of inflow, outflow, and periodic boundary point calculation procedures. Also presented are the C-grid construction procedures. Finally, results of code experimental verification studies for 3-D compressor cascade and rotor flows are presented.

NOMENCLATURE

a speed of sound
\( a_1 \) reference speed of sound (inlet hub)
\( C_p \) specific heat at constant pressure
\( C_x \) blade section axial chord
\( e \) total internal energy
\( j, k, l, n \) indices for \( \xi, \eta, \zeta \), and \( t \) coordinates, respectively
\( J \) Jacobian of coordinate transformation
\( \Delta \) Mach number
\( M \) pressure
\( \rho \) total temperature
\( \rho \) Mach number
\( \rho_0 \) reference density (inlet hub)
\( \rho_1 \) rotational speed of rotor
\( U, V, W \) contravariant velocity components in the \( \xi, \eta, \zeta \) directions, respectively
\( V_r, V_\theta, V_\phi \) velocity components in the \( r, \theta, \phi \) directions in Eq. 18
\( x, y, z \) cartesian coordinates
\( \gamma \) ratio of specific heats
\( \Delta, \nabla \) forward and backward difference operators, respectively
\( \xi, \eta, \zeta \) general curvilinear coordinates
\( J \) Jacobian of coordinate transformation
\( p \) density
\( p \) speed of sound

INTRODUCTION

Over the past 15 years steady progress has been made in the development of fluid flow analyses for turbomachinery blade rows. The eventual goal of these analyses is a time accurate model of the three-dimensional (3-D) flow through the blade rows. Solving the full Navier-Stokes equations over the entire flowfield is the most complete model. Although Rai (1985, 1987) has obtained time accurate Navier-Stokes solutions for a single stage, and Adomczyk (1986) has developed an average pass multistage analysis that includes viscous effects, a complete model is still too complex and computationally too costly. This is especially true considering that a highly accurate analysis for any arbitrary compressor blade row does not yet exist.

The 3-D analysis methods that have been the most highly developed and have provided the greatest advancements in the turbomachinery field are the time-dependent Euler solvers based on a fully conservative form of the governing equations. They provide a single approach for subsonic, transonic, and supersonic flows, and they inherently provide natural shock capturing capability. In many cases, predicted results from Euler solvers are in good agreement with experimental data and provide accurate information on important flow features such as shock location and static pressure distribution. Most importantly, an accurate and efficient Euler solver forms the basis for an efficient viscous solution procedure. Denton (1974) was the first to develop an Euler solver for turbomachines, and he has since been followed by others, including Van Hove (1984), Shieh and Delaney.
GOVERNING EQUATIONS

The differential equations used in this study are the Euler equations for a compressible fluid. If relative Cartesian velocity components are retained as dependent variables in a system attached to a rotating or stationary blade row, the 3-D unsteady Euler equations can be expressed in strong conservation form as

\[
\frac{\partial}{\partial t} \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix} \rho u u + p \rho u \\ \rho u v + \rho u \frac{e}{\gamma - 1} \\ \rho u w + \rho u \frac{e}{\gamma - 1} \\ \rho w u + \rho w \frac{e}{\gamma - 1} \\ \rho e \end{bmatrix} + \frac{\partial}{\partial y} \begin{bmatrix} \rho v u + \rho v \frac{e}{\gamma - 1} \\ \rho v v + p \rho v \\ \rho v w + \rho v \frac{e}{\gamma - 1} \\ \rho w v + \rho w \frac{e}{\gamma - 1} \\ \rho e \end{bmatrix} + \frac{\partial}{\partial z} \begin{bmatrix} \rho w u + \rho w \frac{e}{\gamma - 1} \\ \rho w v + \rho w \frac{e}{\gamma - 1} \\ \rho w w + p \rho w \\ \rho e \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \frac{\partial e}{\partial x} \end{bmatrix}
\]

where

\[
Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e \end{bmatrix}, \quad E = \begin{bmatrix} \rho u u + p \rho u \\ \rho u v + \rho u \frac{e}{\gamma - 1} \\ \rho u w + \rho u \frac{e}{\gamma - 1} \\ \rho w u + \rho w \frac{e}{\gamma - 1} \\ \rho e \end{bmatrix}, \quad F = \begin{bmatrix} \rho v u + \rho v \frac{e}{\gamma - 1} \\ \rho v v + p \rho v \\ \rho v w + \rho v \frac{e}{\gamma - 1} \\ \rho w v + \rho w \frac{e}{\gamma - 1} \\ \rho e \end{bmatrix}
\]

\[
G = \begin{bmatrix} \rho w u + \rho w \frac{e}{\gamma - 1} \\ \rho w v + \rho w \frac{e}{\gamma - 1} \\ \rho w w + p \rho w \\ \rho e \end{bmatrix}, \quad H = \begin{bmatrix} 0 \\ 0 \\ 0 \\ \frac{\partial e}{\partial x} \end{bmatrix}
\]

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

\[
\begin{align*}
U &= \xi (u - u_c) + \xi (v - v_c) + \xi w \\
V &= \eta (u - u_c) + \eta (v - v_c) + \eta w \\
W &= \zeta (u - u_c) + \zeta (v - v_c) + \zeta w
\end{align*}
\]

The inverse Jacobian of the transformation, \(J^{-1}\), is defined as

\[
J^{-1} = \begin{bmatrix} \xi & \eta & \zeta \\ \xi & \eta & \zeta \\ \xi & \eta & \zeta \end{bmatrix}
\]
and the metrics are given by the equations

\[
\begin{align*}
\xi_x &= J(y_n x - x_n y) & n_x &= J(x y_n - x_n y) \\
\xi_y &= J(y_n x - x_n y) & n_y &= J(x y_n - x_n y) \\
\xi_z &= J(x y_n - x_n y) & n_z &= J(x y_n - x_n y)
\end{align*}
\] (10)

The Cartesian velocity components \(u, v,\) and \(w\) are nondimensionalized with respect to the speed of sound at the inlet of the hub section, \(a_1,\) density, \(\rho,\) is referenced to the hub inlet density, \(\rho_{11},\) and the energy and pressure to \(\rho_{11}a_1^2.\) Pressure is defined as

\[p = (\gamma - 1) [e - 0.5p(u^2 + v^2 + w^2)]\] (11)

with \(\gamma\) as the ratio of specific heats.

COORDINATE SYSTEM

The coordinate system for the 3-D analysis is a C-type body conforming system. This grid is particularly attractive because it affords high resolution of the leading edge region to capture bow shocks and minimize the errors that would be convected downstream. As shown in Figure 1, this system is constructed by radially stacking 2-D, C-type grids on surfaces of revolution.

The grid generator is capable of modeling the complete geometry of compressor blade rows including endwall contour and blade twist. 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 intermediate to the hub and shroud are surfaces of revolution. C-type grids are constructed on these surfaces using elliptic techniques. In the grid generation process, a mean radius is calculated for each surface, and the surface is then projected onto a cylinder of that radius (see Figure 2). A grid for each cylindrical section is then found by solving an elliptic system of partial differential equations (Thompson et al., 1985) to produce blade conforming 2-D grids. Controlling functions are introduced to enforce orthogonality. Grid point locations on the blade surface are determined by imposing orthogonality at the surface. All of these steps are performed in an interactive manner, so that the user is able to monitor the generation and alter the grid by varying parameters.

NUMERICAL ALGORITHM

The algorithm used to solve the system of equations (6) is an implicit approximate factorization finite difference scheme. The scheme was developed by Beam and Warming (1976) and was used initially by Steger (1977) and subsequently by Pulliam and Steger (1980). Explicit and implicit artificial dissipation terms are added to attain nonlinear stability, and a spatially variable time step is used to accelerate convergence for steady-state calculations. The diagonal form of the algorithm is used because it allows for the use of fourth-order implicit dissipation and produces a robust, rapidly converging scheme in most cases.

By applying implicit time differencing, local time linearizations, and approximate factorization as shown by Pulliam (1984), Eq. (6) can be written as

\[
(1 + hE)A(1 + hE)B(1 + hE)CDE = -h[dE + dE + dE]eH (12)
\]

where \(A = 3E/3Q, B = 3P/3Q,\) and \(C = 3C/3Q\) are the flux Jacobians, each of which has real eigenvalues and a complete set of eigenvectors. The term \(h\) is the spatially variable time step, given as \(dt/(1.4v).\) 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). Eq. (12) consists of an implicit (left) side and an explicit (right) side. The left side has three implicit operators, each of which is block tridiagonal.

The spatial derivative terms are approximated with second-order central differences. The computational work can be decreased by introducing a diagonalization of the blocks in the implicit operators as developed by Pulliam and Chaussee (1981). The eigenvalues of the flux Jacobians A, B, and C are used in this development. Because the flux Jacobians have real eigenvalues and the complete set of eigenvectors, the matrices can be diagonalized (Warming et al., 1975; Turkel, 1973), i.e.,

$$\Lambda_\zeta = T_{\zeta}^{-1}AT_{\zeta}, \quad \Lambda_\eta = T_{\eta}^{-1}BT_{\eta}, \quad \Lambda_\zeta = T_{\zeta}^{-1}CT_{\zeta}$$  \hspace{1cm} (13)

where $T_{\zeta}$, $T_{\eta}$, and $T_{\zeta}$ are the matrices whose columns are the eigenvectors of $A$, $B$, and $C$. Replacing $A$, $B$, and $C$ in Eq. (13) by their eigensystem decomposition yields

$$[T_{\zeta}^{-1}AT_{\zeta}] [T_{\eta}^{-1}BT_{\eta}] [T_{\zeta}^{-1}CT_{\zeta}] = R_n$$  \hspace{1cm} (14)

A modified version of Eq. (14) can be obtained by factoring the $T_{\zeta}$, $T_{\eta}$, and $T_{\zeta}$ eigenvector matrices outside the spatial derivative terms $\delta_\zeta$, $\delta_\eta$, $\delta_\zeta$. The resulting equations are

$$T_{\zeta}[I+h\delta_\zeta A]\{N[I+h\delta_\zeta A]\}P[I+h\delta_\zeta A]T_{\eta}^{-1}\delta_\eta = R_n$$  \hspace{1cm} (15)

where $N = T_{\zeta}^{-1}T_{\eta}$ and $P = T_{\eta}T_{\zeta}^{-1}$.

The explicit side of the diagonal algorithm is the set of steady state finite difference equations and is exactly the same as the original algorithm. In addition, computational experiments by Pulliam and Chaussee (1981) have shown that the convergence and stability limits of the diagonal algorithm are similar to those of the block tridiagonal algorithm. The diagonal algorithm reduces the block tridiagonal inversion to four $5 \times 5$ matrix multiplies and three scalar tridiagonal inversions, with an overall savings in computational work that can be as high as 40% (Pulliam, 1986a).

**CODE DEVELOPMENTS**

The turbomachinery flow code that has been developed is based on the external flow code ARCD3, which was written at the NASA Ames Research Center. The original ARCD3 code is capable of calculating the flow about bodies with bilateral symmetry in an external environment. The steps used to adapt the code and algorithm to turbomachinery flow are outlined in this section.

The first step in this adaptation was the reformulation of the Euler equations in a rotating Cartesian frame of reference. This involved the inclusion of terms to account for velocities and accelerations in the relative system. The second step was the transformation of the equations to generalized curvilinear coordinates. This yielded Eq. (6), which is very similar to the form of the equations shown by Pulliam and Steger (1980) or Pulliam (1986a) where the algorithm is given in more detail. The major difference is the presence of source terms to account for relative accelerations. At this point, it was possible to follow the development shown in detail by Pulliam (1984) and briefly in this paper to arrive at the diagonal form of the algorithm.

The next step involved code modification to account for the relative terms. The form of the code that had been obtained did not account for these terms in a global sense. The changes to the original code were extensive and were implemented in an algorithmic manner. The code defines the velocity of the rotating coordinate system at each point and calculates the relative velocity and acceleration components when required.

Although the changes in a Cartesian system formulation to account for rotation are more extensive than those in a polar coordinate system formulation, the truncation error in a Cartesian formulation is not a function of radius as it is with the polar coordinate formulation. This advantage may become more important as the accuracy required from the analysis increases.

After the equations had been reformulated and the basic code had been modified, the far field external flow boundary point calculations from ARCD3 were replaced by 3-D calculations for the treatment of periodic, inlet, and exit boundaries. With the modification to the boundary procedures and those to account for the relative system of reference, the code was qualified for axial turbomachinery flow computations.

In addition to the major modifications for turbomachinery flow calculations, some algorithm modifications or upgrades were also implemented. First, the mixed second- and fourth-order damping scheme of Jameson (1981) was extended from one to all three coordinate directions. This scheme is explained in the section on the nonlinear artificial dissipation procedure, described by Pulliam and Steger (1980), can be used to ensure that the metric invariants are exactly zero. Basically, this is a weighted averaging that computes the metrics in a finite volume manner. The alternative is to subtract out the error term from the coordinate transformation as a source term on the explicit right-hand side. This is not the same as the approach of Pulliam and Steger for maintaining the free stream. However, it is the approach that contributed most toward improving the 3-D solutions.

**BOUNDARY CONDITIONS**

As with the ARCD3 flow code, the dependent variables are updated explicitly, which means that there is a first-order error in time at the boundaries. Because the boundary procedures are a modular element of the code, they can be altered or replaced without interfering with the implicit algorithm. The far field boundary procedures in ARCD3 were replaced with 3-D turbomachinery boundary procedures. For 3-D turbomachinery calculations, there are six boundaries. Figure 3 is a schematic diagram of a 2-D C-grid section. Three of the boundaries, the hub and tip section and the blade, are solid surfaces. The other three boundaries are the inlet, exit, and periodic boundaries.

At the inlet, an extension of the 2-D procedure used by Chima (1985) that allows for the specification of total temperature, total pressure, and the radial and tangential velocity components is used. Nominal radial distributions of all of these properties can be specified. The procedure uses a characteristic boundary condition similar to that used by Jameson and Baker (1983) where the upstream-running Riemann invariant $R^n$, based on the total velocity $q$, is extrapolated from the interior to the boundary, i.e.,
Pressure is found using the normal momentum relation, which is a combination of the three transformed momentum equations given by

\[ q_{in} = \left[ (\gamma-1)R - \sqrt{\gamma(\gamma+1)}C_pT_o - (\gamma-1)R^{-2} \right] / (\gamma+1) \]  

where \( y \) is the normal direction to the \( \xi = \text{constant} \) solid surface. In this form, the boundary conditions are applicable to steady or unsteady motion. Surface densities are found using a boundary condition suggested by China (1985) and by Barton and Pulliam (1984). The entropy expressed as \( S = p/\rho \gamma \) is extrapolated to the body and used to find the density. This condition is very stable and conserves total pressure or entropy better than a boundary condition in which rothalpy or total enthalpy is specified.

**NONLINEAR ARTIFICIAL DISSIPATION MODEL**

One of the important aspects of compressor aerodynamics is the ability to capture shocks and predict shock losses. MacCormack and Baldwin (1975) used a second-difference dissipation operator for the solution of the Navier-Stokes equations for flow with shocks. More recent work by Jameson, et al. (1981) and Pulliam (1986a, 1986b) shows that a mixed second- and fourth-order dissipation model with appropriate coefficients should give a central difference scheme good shock capturing capability. The model used in this 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

\[ \phi_j = \sigma_j \phi_{j-1} \phi_j \phi_{j+1} - \sigma_j \phi_{j-1} \phi_j \phi_{j+1} \]

\[ \sigma_j = \left| \phi_{j-1} \phi_j \phi_{j+1} \phi_{j+2} \right| + \left| \phi_{j-2} \phi_{j-1} \phi_j \phi_{j+1} \right| + \left| \phi_{j-1} \phi_{j} \phi_{j+1} \phi_{j+2} \right| + \left| \phi_{j-2} \phi_{j-1} \phi_{j} \phi_{j+1} \right| + \left| \phi_{j-2} \phi_{j-1} \phi_{j} \phi_{j+2} \right| 
\]

where the coefficients for the constants are \( K_2 = 1/4 \) and \( K_3 = 1/100 \).
The first term is a second-order dissipation model with an extra pressure gradient coefficient to increase its value near shocks. The second term is a fourth-order model where the logic to compute $e^{\chi^4}$ switches it off 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 dissipation term is modified to maintain a dissipative term. A derivation and analysis of various boundary treatments for dissipation models is given by Pulliam (1986b).

RESULTS AND DISCUSSION

Numerical solution results for a compressor cascade and an isolated rotor are presented and compared with experimental data. The solutions were started from uniform inlet hub conditions with the hub exit static pressure set at the desired steady-state value. The final values of the damping coefficients for the rotor solution were $K_2 = 1/3$ and $K_4 = 1/50$, which were above the suggested (Pulliam; 1986a, 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 right-hand side residual had been reduced more than three orders of magnitude; however, this is more a rule of thumb than an absolute criterion.

Controlled Diffusion Airfoil

The accuracy of the 3-D code was tested during its development by comparison of solutions with experimental data. The solutions were started from uniform inlet hub conditions with the hub exit static pressure set at the desired steady-state value. The final values of the damping coefficients for the rotor solution were $K_2 = 1/3$ and $K_4 = 1/50$, which were above the suggested (Pulliam; 1986a, 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 right-hand side residual had been reduced more than three orders of magnitude; however, this is more a rule of thumb than an absolute criterion.

Controlled Diffusion Airfoil

The accuracy of the 3-D code was tested during its development by comparison of solutions with experimental data. The initial verification was carried out using data for a rectilinear cascade of supercritical airfoils. The cascade data were used to validate the turbomachinery boundary point calculations.

The rectilinear cascade, shown in Figure 4, was tested by Stephens and Hobbs (1979) over a range of inlet Mach numbers and incidence angles. Sample experimental midspan airfoil Mach number distributions are presented in Figure 5. 2-D and 3-D flow predictions were made for the design operating condition with inlet Mach number $M_1 = 0.735$ and incidence $i = 0$ deg. The 3-D analysis accounted for the endwall boundary layer blockage by linearly contracting the endwalls by the axial velocity density ratio (AVDR) of 1.17, whereas the 2-D analysis, determined using the same numerical scheme, did not account for blockage. As shown in Figure 5, the 3-D predictions more closely match the experimental data. The oscillations in the solution near $X/C_x = 1.0$ result from the use of a C-type grid that approximates the trailing edge geometry as a wedge.

NASA Fan Rotor 67

The capability of the code 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, shown in Figure 6. This rotor provided a rigorous test for the grid generator and the flow solver. The highly loaded rotor was tested at the NASA Lewis Research Center and
reported by Pierzga and Wood (1984). The rotor has 22 low aspect ratio (1.56) 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. As Figure 6 reveals, this rotor has a large amount of twist from hub to tip, which produces a highly 3-D flow field. The C-type grid contains 121 normals, 21 contours, and 17 sections hub to tip. The solutions converged three orders of magnitude in approximately 500 iterations and required 10 minutes of CPU time on a CRAY X-MP vector computer.

The analysis was used to simulate the flow at the peak efficiency and near stall operating points.

Figure 7 shows predicted surface relative Mach number and experimental near surface relative Mach number distributions for the peak efficiency operating point. The comparisons are made at 30, 70, and 90% span, measured from the hub. The predicted static pressure contours at the same spanwise locations are presented in Figure 8. Although the agreement is better at 90 and 70% span, overall agreement is good. The solution was obtained by adjusting the hub exit static pressure to obtain the best overall agreement. At peak efficiency, this was 1.4% higher than the reported hub exit static pressure. Figure 8 clearly shows the shock structure within the blade row at the near-tip sections.

Fig. 7. Predicted and experimental Mach number distributions for NASA Rotor 67 at the peak efficiency operating condition.

Fig. 8. Static pressure contour plots for NASA Rotor 67 at 30, 70, and 90% span at the peak efficiency operating condition.
Results for the near stall operating point are presented in Figures 9 and 10. Figure 9 compares the predicted surface and experimental near surface relative Mach number distributions at 30, 70, and 90% span. Figure 10 shows the predicted static pressure contours at 30, 70, and 90% span. Starting with the peak efficiency solution, the exit static pressure was raised until the best overall comparison was obtained for the near stall operating point. In this case, the hub exit static pressure was 2% higher than the reported hub exit static pressure. Overall agreement is fairly good. The discrepancies between the predicted results and data on the pressure surface near the leading edge may be due to inadequate mesh density in that...
high gradient region. The static pressure contour plots at 70 and 90% span show that at the near stall operating point, the passage normal shock has been driven upstream and has combined with the leading edge bow shock.

SUMMARY

An efficient 3-D turbomachinery flow analysis method has been presented. The method, 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 Euler equations with body conforming C-type grids. Explicit and implicit artificial dissipation terms were added to attain nonlinear stability. The grids were constructed by stacking 2-D C-type grids on surfaces of revolution. Numerical solution results for two 3-D compressor flows have been presented. The solution for a cascade of supercritical airfoils and for NASA fan rotor 67 are compared with experimental data to demonstrate the accuracy of the analysis method.

REFERENCES