Development of an Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery

Rodrick V. Chima
Lewis Research Center
Cleveland, Ohio

Prepared for the
Twenty-fourth Aerospace Sciences Meeting
sponsored by American Institute of Aeronautics and Astronautics
Reno, Nevada, January 6-9, 1986
Abstract

A rapid quasi-three-dimensional analysis has been developed for blade-to-blade flows in turbomachinery. The analysis solves the unsteady Euler or thin-layer Navier-Stokes equations in a body-fitted coordinate system. It accounts for the effects of rotation, radius change, and stream-surface thickness. The Baldwin-Lomax eddy-viscosity model is used for turbulent flows. The equations are solved using a two-stage Runge-Kutta scheme made efficient by use of vectorization, a variable time-step, and a flux-based multigrid scheme, which are all described. A stability analysis is presented for the two-stage scheme. Results for a flat-plate model problem show the applicability of the method to axial, radial, and rotating geometries. Results for a centrifugal impeller and a radial diffuser show that the quasi-three-dimensional viscous analysis can be a practical design tool.

Introduction

Turbomachinery intended to produce large amounts of power from a small volume often require use of radial-flow or mixed-flow components, that is, components in which the streamwise velocity is not strictly axial. Radial-flow turbomachines such as centrifugal impellers, radial diffusers, and radial-inflow turbines have a predominately radial flow direction. Mixed-flow turbomachines may be used when restrictions on space prevent a completely radial flow. Complicated geometries, shock waves, and viscous phenomena make analysis of radial- or mixed-flow turbomachines more difficult than analysis of strictly axial-flow machines.

Analysis of axial-flow turbomachinery blade rows is usually simplified by modeling a blade section as a flat cascade. The governing equations for a flat cascade are the same two-dimensional flow equations that are solved for isolated airfoils, so cascade analyses often draw heavily on numerical techniques developed for two-dimensional external flows. Examples of flat-cascade analyses include panel methods, potential methods, Euler solutions, and Navier-Stokes solutions.

Flows in radial- or mixed-flow turbomachines are inherently three-dimensional, requiring specification of the axial, radial, and tangential velocity components to fully specify the flow. A simplification that allows these machines to be analyzed in two dimensions was proposed by E. H. Wu in 1952. In Wu's model the flow is assumed to follow an axisymmetric stream surface (Wu's "S2 surface," Fig. 1). The radius and thickness of the stream surface are assumed to be known as functions of the streamwise distance. These quantities are usually obtained from an axisymmetric through-flow or "meridional" analysis, sometimes coupled with a boundary layer analysis on the hub and shroud.

The equations governing the flow along the stream surface combine the axial- and radial-velocity components into one streamwise component, and are thus two-dimensional. The solution can be resolved into three velocity components since the shape of the surface is known. Specifying the stream-surface thickness allows variable blade heights and end wall displacement thicknesses to be modeled. This is similar to specifying area change in the one-dimensional nozzle equations. Since the effects of radius change and stream-surface thickness are modeled in this analysis, it is termed "quasi-three-dimensional." Examples of quasi-three-dimensional turbomachinery analyses include panel methods, stream-function methods, potential methods, and Euler methods.

In the present work, the Euler and Navier-Stokes code developed for flat cascades in Ref. 6 has been extended to a quasi-three-dimensional analysis. It is thought that this is the first Navier-Stokes analysis to include the effects of rotation, radius change, and stream-surface thickness. The explicit MacCormack algorithm used in Ref. 6 has been replaced with an explicit two-stage Runge-Kutta finite-difference algorithm based on the work of Jameson. Efficiency is achieved by three means: vectorization, use of a variable time-step, and use of a multigrid scheme developed by Johnson and Chima.

Governin' Equations

The axisymmetric (m,e) coordinate system used for the quasi-three-dimensional analysis is shown in Fig. 1. Here the m-coordinate is defined by

\[ 2 \quad 2 \quad 2 \quad 2 \quad \alpha m = \alpha \quad o + \pi \quad (1) \]

and the e-coordinate is defined by:

\[ e = \theta + wt \quad (2) \]

where \( \theta \) is fixed in space and \( e \) rotates with the blade row with angular velocity \( \theta \). The radius \( r \) and the stream surface thickness \( h \) are taken to be known functions of \( m \). In this coordinate system the dimensionless Navier-Stokes equations may be written in the following nearly-conservative form:

\[ \alpha \phi + \alpha m f - \mathrm{Re}^{-2} \phi + \alpha g \left( \mathrm{Re}^{-2} s \right) = K \quad (3) \]
\[
\begin{align*}
\sigma_{11} &= 2 \omega v_m \frac{m}{h} + \frac{\alpha}{r} + V \\
\sigma_{22} &= 2 \omega \left( \frac{v_\theta}{r} + \frac{v_m r}{m} \right) + \frac{\alpha}{r} + V \\
\sigma_{12} &= u \left( r v_\theta - v_\theta r + \frac{1}{r} \right) v_m
\end{align*}
\]

and

\[
\omega - V = \frac{-2}{3} u \left[ a_0 v_\theta + v_m \left( r \frac{r}{m} + \frac{m}{h} \right) + \frac{1}{r} \right]
\]

is the dilatation.

The equations are nondimensionalized by arbitrary reference quantities (here the inlet total density and critical sonic velocity define the reference state), and the Reynolds number \(Re\) and the Prandtl number \(Pr\) must be specified in terms of that state. These equations assume that the specific heats \(C_p\) and \(C_v\) and the Prandtl number are constant, that Stokes' hypothesis \(a = -2/3 u\) is valid, and that the effective viscosity may be written

\[
\nu = \nu_{\text{laminar}} + \nu_{\text{turbulent}}
\]

Equations (3) to (6) are transformed from the \((m, e)\) coordinate system to a general body-fitted \((\xi, \eta)\) coordinate system using standard methods. The thin-layer assumption is then used to eliminate viscous derivatives in the streamwise \((\xi)\) direction, thereby reducing computational overhead while retaining the capability of computing separated flows. The resulting equations are similar to those developed by Katsman.\(^{20}\)

\[
z \hat{\alpha} + z \hat{\beta} + z_0 (\hat{G} - \text{Re}^{-1} \beta) = \hat{K}
\]

where

\[
\begin{align*}
\sigma_{11} &= 2 \omega v_m \frac{m}{h} + \frac{\alpha}{r} + V \\
\sigma_{22} &= 2 \omega \left( \frac{v_\theta}{r} + \frac{v_m r}{m} \right) + \frac{\alpha}{r} + V \\
\sigma_{12} &= u \left( r v_\theta - v_\theta r + \frac{1}{r} \right) v_m
\end{align*}
\]

and

\[
\omega - V = \frac{-2}{3} u \left[ a_0 v_\theta + v_m \left( r \frac{r}{m} + \frac{m}{h} \right) + \frac{1}{r} \right]
\]

is the dilatation.

The equations are nondimensionalized by arbitrary reference quantities (here the inlet total density and critical sonic velocity define the reference state), and the Reynolds number \(Re\) and the Prandtl number \(Pr\) must be specified in terms of that state. These equations assume that the specific heats \(C_p\) and \(C_v\) and the Prandtl number are constant, that Stokes' hypothesis \(a = -2/3 u\) is valid, and that the effective viscosity may be written

\[
\nu = \nu_{\text{laminar}} + \nu_{\text{turbulent}}
\]

Equations (3) to (6) are transformed from the \((m, e)\) coordinate system to a general body-fitted \((\xi, \eta)\) coordinate system using standard methods. The thin-layer assumption is then used to eliminate viscous derivatives in the streamwise \((\xi)\) direction, thereby reducing computational overhead while retaining the capability of computing separated flows. The resulting equations are similar to those developed by Katsman.\(^{20}\)

\[
z \hat{\alpha} + z \hat{\beta} + z_0 (\hat{G} - \text{Re}^{-1} \beta) = \hat{K}
\]

where

\[
\begin{align*}
\sigma_{11} &= 2 \omega v_m \frac{m}{h} + \frac{\alpha}{r} + V \\
\sigma_{22} &= 2 \omega \left( \frac{v_\theta}{r} + \frac{v_m r}{m} \right) + \frac{\alpha}{r} + V \\
\sigma_{12} &= u \left( r v_\theta - v_\theta r + \frac{1}{r} \right) v_m
\end{align*}
\]

and

\[
\omega - V = \frac{-2}{3} u \left[ a_0 v_\theta + v_m \left( r \frac{r}{m} + \frac{m}{h} \right) + \frac{1}{r} \right]
\]

is the dilatation.

The equations are nondimensionalized by arbitrary reference quantities (here the inlet total density and critical sonic velocity define the reference state), and the Reynolds number \(Re\) and the Prandtl number \(Pr\) must be specified in terms of that state. These equations assume that the specific heats \(C_p\) and \(C_v\) and the Prandtl number are constant, that Stokes' hypothesis \(a = -2/3 u\) is valid, and that the effective viscosity may be written

\[
\nu = \nu_{\text{laminar}} + \nu_{\text{turbulent}}
\]
and

\[ K_2 = \left( \nu p + p - \rho^{-1} \alpha_{12} \right) \frac{m}{r} + \left( p - \rho^{-1} \alpha_{33} \right) \frac{h}{r} \]

\[ S_4 = \frac{k}{(\gamma - 1) \rho} \left( \frac{\alpha_m}{\alpha_n} - \frac{\alpha_n}{\alpha_m} \right) \frac{a_n^2}{\alpha_n^3} + v_m S_2 + v_\theta S_3 \]

In Eq. (8) the overbar denotes a rescaling of the metric terms:

\[ \bar{e}_i = \frac{e_i}{r}; \quad \bar{\epsilon}_n = \frac{\epsilon_n}{r}; \quad \bar{\theta} = \frac{\theta}{r} \]

where \( \lambda \) is the Jacobian of the transformation

\[ J = \frac{\epsilon_m^* - \epsilon_n^*}{(\epsilon_n^* - \epsilon_m^* / \lambda)} \]

and the metric quantities are determined from the grid-point coordinates using central differences and:

\[ \epsilon_m = \epsilon_n; \quad \epsilon_n = -\epsilon_m; \quad \eta_m = -\eta_n; \quad \eta_n = \eta_m \]

The relative contravariant velocity components \( w^c \) and \( w^n \) along the \( \zeta \) and \( \eta \) grid lines are given by:

\[ \begin{align*}
w^c &= \epsilon_m \nu_m + \epsilon_n \nu_n; \\
&= \eta_m \nu_n + \eta_n \nu_m \\
&= \eta_m \nu_n + \eta_n \nu_m
\end{align*} \]

The shear stress terms are found from Eq. (6) by replacing \( \alpha_m \) with \( \alpha_m^* \) and \( 1/\rho m^2 \) with \( \alpha_m^* \alpha_n^* \).

The quasi-three-dimensional equations (Eqs. (7) to (12)) are similar to the two-dimensional equations solved in Ref. 6 except for the source term \( K_2 \), the radius appearing in the \( \alpha \)-momentum equation, the rescaled metrics (Eq. (9)), and the relative velocity component \( w^r \) appearing in the contravariant velocities (Eq. (12)). Equations (7) to (12) reduce to the two-dimensional equations for constant \( r \) and \( h \), and zero rotation. Note that Eqs. (7) to (12) are independent of the magnitude of the stream surface thickness \( h \) so that any function \( h(m) > 0 \) may be used. The equations do depend on the magnitude of the radius \( r \) because of the \( 1/r^2 \) terms scaling \( \epsilon_n \) and \( \eta_n \) in Eq. (9).

For turbulent flows the two-layer eddy viscosity model developed by Baldwin and Lomax21 is used. In the \( (m,c) \) coordinate system the wall shear \( \tau_w \) and vorticity \( \omega \) required by the model are given by:

\[ \tau_w = \alpha_{12} \omega = \omega \left( \alpha_m^* + 1/\rho m^2 \nu_m - \alpha_n^* \nu_n \right) \]

\[ \omega = \frac{1}{2} \left( \alpha_m^* \nu_n - 1/\rho m^2 \nu_m + \alpha_n^* \nu_n \right) \]

**Computational Grid**

Body-fitted grids for this work were generated using the GRAPE code (GRids about Air-foils using Poisson's Equation) developed by Sorenson,22,23 Briefly, the code allows arbitrary specification of inner and outer boundary points, then generates interior points as a solution to a Poisson equation. Forcing terms in the Poisson equation are chosen such that desired grid spacing and intersection angles may be maintained at the inner and outer boundaries.

New inner and boundary subroutines were written for turbomachine geometries. The new inner boundary has constant spacing around blade leading and trailing edges, larger constant spacing over blade surfaces, and exponential stretching connecting the regions. The new outer boundary is composed of the mean-camber line between the blades, a quadratic extension upstream, and a linear extension downstream. The C-shaped grids are periodic over the pitch of the blade. Grids are generated in a Cartesian \((m,\eta)\) coordinate system, where \( \eta \) is some mean radius. The local radius and stream surface thickness are supplied to the Navier-Stokes code at a later time as tabulated functions of \( \eta \), then spline-fit to the grid. The \( m = r_m/r \) and \( h = h_m/h \) in Eqs. (7) and (8) are calculated using central differences and are stored.

**Initial Conditions**

Since a centrifugal compressor can produce pressure ratios of 5:1 or greater it is not generally possible to start a quasi-three-dimensional calculation with constant initial conditions. Instead an analytic solution of the one-dimensional flow equations with area change is used. Turbomachinery blades are usually designed based on desired leading- and trailing-edge velocity triangles, so the relative flow velocity \( W \) and angle \( \alpha \) at the leading edge and the relative flow angle at the trailing edge are used as input for the initial conditions.

Using \( (') \) to denote absolute total conditions and \( (')^* \) to denote relative total conditions, the continuity and energy equations are

\[ \dot{m} = \alpha \rho h m^2 \cos \alpha = \text{constant} \]

\[ I = C_p T' - m \nu_\theta = C_p T'^* - 1/2 \rho u'^* = \text{constant} \]

where \( \alpha \) is the blade spacing and \( I \) is the rothalpy.

Outside of the blade row the angular momentum equation gives:

\[ \dot{r} \nu_\theta = \text{constant} \]

Equations (16) and (17) can be used with the isentropic relations to show that total conditions are constant outside of the blade row. Evaluating Eq. (15) at some point \( (')_2 \) and using Eq. (17) to eliminate \( \dot{m} \) gives:
The axisymmetric m-momentum equation may be written as:

\[ a_t R^- + (v_m - a) m R^- = (\gamma^2 + a v_m) \frac{r_m}{r} + a v_m h_m/n \]

where

\[ R^- = v_m - \frac{2a}{\gamma - 1} \]

is the upstream-running Riemann invariant. For steady axisymmetric flow Eq. (22) may be written as:

\[ 3 \gamma R^- = \left( \frac{T}{T_2} \right) \left( \frac{1}{\gamma - 1} \right) \left( v_m + a v_m \right) \frac{r_m}{r} + a v_m h_m/n \]

Equation (23) is backward-differenced and solved for \( R^- \) at the inlet. Then \( v_m \) is found from \( R^- \) and the specified whirl \( rv_0 \) and total temperature \( T^* \) using:

\[ v_m = \frac{\sqrt{(y+1)(4C T^* - 2v_0^2) - 2(y-1)(R^-)^2}}{(y+1)} \]

Density and energy are found using isentropic relations.

At the exit the static pressure is specified and the other flow quantities are found using first-order extrapolation. Zeroth-order extrapolation is not usually sufficient in cylindrical coordinate systems because the radius and thus the velocity may change between grid points. For the same reason the inlet and exit boundaries cannot be placed arbitrarily far from the blades. Too small a radius can cause the flow to be supersonic and to large a radius can cause the velocity to approach zero at the boundaries.

Blade surface pressures are found from the normal momentum equation:

\[ (\nu_m n_m + \nu_0 - \nu_0) \frac{n_p}{n} + (\nu_m^2 + \nu_0^2) \frac{n_p}{n} \]

\[ = -\rho W^2 (n_m \nu_m + \nu_0 - \nu_0 \nu_m) + \nu_0 (n_m \nu_0 - \nu_0 \nu_m) \frac{r_m}{r} \]

where \( W^2 = 0 \) on the surface for viscous flows.

Periodic boundaries are solved like interior points.

Fine-Grid Algorithm

On the fine grid an explicit two-stage finite-difference Runge-Kutta algorithm based on the work of Jameson is used. It is given below as applied to Eq. (7).
Two-stage Runge-Kutta algorithm

\[ q^{(1)} = q^n - a_1 \Delta t q^n \]
\[ q^{(2)} = q^n - a_1 \Delta t q^{(1)} \]

where

\[ a_1 = 1.2 \]

\[ R_{i,j} = -R_{i,j} + \frac{1}{2} \left( \frac{P_{i+1,j} - P_{i-1,j}}{2} \right) \]

Artificial dissipation

\[ q^{n+1} = q^{(2)} + \Delta t q^{(2)} \]

Collect residuals

\[ \Delta q^{n+1} = q^{n+1} - q^n \]

The two-stage scheme given by Eq. (26) has a Courant number limit of one. It is used in preference to a higher-order scheme with a higher Courant number limit because the multigrid scheme used here also has a Courant number limit of one.

Four minor advantages of the Runge-Kutta scheme over the MacCormack scheme are noted:

1. A steady Runge-Kutta solution is independent of the time step while a steady MacCormack solution is not. This is not true when the artificial dissipation is added in a fractional step as in Eq. (27).

2. The centrally-differenced Runge-Kutta scheme produces perfectly symmetric solutions for symmetric problems while the one-sided MacCormack scheme does not.

3. For a two-dimensional problem with centrally-differenced metrics, the Runge-Kutta scheme exactly conserves a free stream while the MacCormack scheme does not. Although the Runge-Kutta scheme is not fully conservative for the quasi-three-dimensional problem because of the source term, it has been found to possess better conservation properties in general.

4. The Runge-Kutta scheme is slightly easier to program than the MacCormack scheme.

Differences in convergence rates between the two schemes are negligible for Courant numbers near one.

Artificial Dissipation

Dissipative terms consisting of second and fourth differences are added to prevent odd-even point decoupling and to allow shock capturing. The dissipative terms are similar to those used by Jameson15 and others. A one-dimensional version (c-direction) is given below. In two dimensions the dissipation is applied as a sequence of one-dimensional operators.
The damping coefficient $\gamma$ is chosen such that $g = 0$ at $\lambda = n-1$, so $\lambda$ is taken to be one while $a_1$ is chosen. Figure 2 shows a plot of Eq. (33) for several values of $a_1$, with $\lambda = 1$ and $\mu = 1/(16 \lambda)$. It can be seen that the two-stage scheme is stable for $0.5 < a_1 < 1$, with $a_1 = 1.2$ giving the best damping over the range of frequencies. For $a_1 = 1.2$ and $\mu = 1/(16 \lambda)$ it can be shown that the two-stage scheme is stable for Courant numbers $CFL < 1.1$. In general the two-stage scheme is first-order accurate in time. It is second-order accurate in time only if $a_1 = 0.5$.

The second part of the stability analysis considers the linearized Euler subset of the governing equations (Eqs. (7) and (8)). A Von Neumann analysis shows the stability limit on the time step to be:

$$a t \leq \left[ \frac{1}{\left( \frac{c}{\Delta x} \right)^2 + \left( \frac{\nu}{\Delta y} \right)^2} \right]^{1/2}$$

which is implemented as:

$$a t \leq CFL \Delta t \left[ \frac{1}{\left( \frac{c}{\Delta x} \right)^2 + \left( \frac{\nu}{\Delta y} \right)^2} \right]^{1/2}$$

where

$$\Delta x = 2 \Delta t \left( \frac{c}{\Delta x} \right)$$

$$\Delta y = 2 \Delta t \left( \frac{\nu}{\Delta y} \right)$$

A spatially-variable time-step $\Delta t(i,j)$ is used in the two-stage scheme (Eqs. (26) and (27)) to accelerate convergence to steady state. Equation (34) is used to choose the time step at each grid point such that the Courant number is constant, typically CFL = 0.95. Time steps are calculated based on the initial conditions. They are stored and not updated during the calculations.

**Multigrid Algorithm**

The multigrid algorithm originated by Ni16 and modified by Johnson and Chima17-19 is used to accelerate convergence of the fine-grid algorithm. Ni's scheme is basically a one-step Lax-Wendroff scheme applied on a coarse grid. Ni used it to accelerate his own fine-grid Euler scheme. Johnson adapted Ni's method to other finite-volume schemes including MacCormack's scheme. He also used it for viscous flows by demonstrating that dissipative terms need not be included on the coarse grids. Thus the multigrid scheme used here is based solely on the Euler equations. It is entirely independent of the viscous terms, the turbulence model, and the artificial dissipation used on the fine grid.

One-step Lax-Wendroff schemes including Ni's scheme require temporal derivatives of the flux vectors. These terms are computed as the Jacobian matrix of the flux vector times the temporal differences of the solution vector. Johnson replaced these lengthy computations with a direct temporal difference of the flux vector using the old and new solutions on the fine grid. This "flux-based" scheme is considerably simpler than Ni's original scheme.

The flux-based multigrid scheme is derived by expanding the finite-grid change $\Delta q$ in a Taylor series.

$$\Delta q^n = \Delta q^{n+1} + \Delta t (\Delta q^{n+1})_t + O(\Delta t^2)$$

The Euler equations are used to replace the third term.

$$(\Delta q^{n+1})_t = [\Delta t (R - F_t - G)]_t$$

Intercalating the space and time derivatives and using backward differencing in time gives:

$$(\Delta q^{n+1})_t = \Delta t (R - F_t - G)_t$$

$$= (R^{n+1} - R^n) + (F^{n+1} - F^n) c (G^{n+1} - G^n)_n$$

and finally

$$\Delta q^{n+1} = \Delta q^n + \Delta t [R - (F_t + G)]_n$$

where

$$\Delta q^n = \Delta q^n + \Delta t [R - (F_t + G)]_n$$

Equation (36) is implemented on a coarse grid with spacing $\Delta x$ and $\Delta y$, and time step $\Delta t$, $i = 2, 4, 8, \ldots$, using:

$$\Delta q^{n+1} = \frac{1}{8} \left\{ \begin{array}{c}
[\Delta q + \Delta t (R - F_t - G)]_{i=2,3}
+ [\Delta q + \Delta t (R - F_t + G)]_{i=4,5}
+ [\Delta q + \Delta t (R + F_t - G)]_{i=6,7}
+ [\Delta q + \Delta t (R + F_t + G)]_{i=8,9}
\end{array} \right\}$$

Fine-grid changes $\Delta q^n$ are restricted to the coarse grid by injection. The multigrid algorithm (Eq. (37)) computes coarse-grid corrections $\Delta q^{n+2}$ based on changes in the fine-grid solution. Thus the coarse-grid corrections must vanish if the fine-grid solution converges, thereby retaining fine-grid accuracy.
The coarse-grid corrections are prolonged back to the fine grid using bilinear interpolation and the fine-grid solution is updated. The process may then be repeated on a coarser grid.

**Vectorization**

The explicit Runge-Kutta and multigrid algorithms used here have been highly vectorized for the Cray 1-S at NASA Lewis Research Center. Indeed the Runge-Kutta computations were clocked at about 40 million floating point operations/sec (40 mflops) for an Euler solution on a 113 by 25 grid. The efficiency of the multigrid computation: decreases as the grid gets coarser and the vectors get shorter.

The code was redimensioned for each grid size run and required 260 K words of memory for the largest grid (161 by 33). The quasi-three-dimensional code requires about 20 percent more storage than the two-dimensional code.  

**Results**

Results are presented for the following problems: a cascade of thin flat plates with round leading edges, a centrifugal impeller, and a vaned radial diffuser. Euler and Navier-Stokes results are presented.

To aid in developing the quasi-three-dimensional code and to illustrate the capabilities of the analysis, a model problem was developed representing a cascade of thin flat plates with round leading edges. The plate has unit chord, four percent thickness, and a pitch of 0.7. Figure 3 shows the computational grid around the plate. The inviscid flow grid had 113 by 25 points with 5° spacing around the leading edge circle and a normal spacing of 1.8x10^-2. The viscous flow grid had 113 by 33 points with a normal spacing of 5x10^-5. The grid shown in Fig. 3, and most of the subsequent grid and contour plots in this paper, are drawn in a transformed plane in which the absissa is \( r = \frac{h}{r} \) and the ordinate is \( s \). This transformation preserves angles and is discussed in Ref. 10.

Grids are generated in an \((m, \theta)\) coordinate system where \( r \) is some mean radius, and are independent of the local radius \( r \) and stream-surface thickness \( h \). Values of \( r \) and \( h \) are supplied to the quasi-three-dimensional code later and can be varied to simulate different geometries.

A flat cascade was simulated by setting \( r = \) constant and \( h = \) constant. Figure 4 shows static pressure contours for an Euler solution with an inlet Mach number of 0.33. (Titles on the figures give nominal values of Mach number, Reynolds number, and flow angle based on the leading-edge velocity triangles given as input. Actual inlet conditions may change as the solution develops.) The contours show the stagnation region and a mild acceleration due to blockage. Identical results were obtained using the two-dimensional code.  

The solution was run 5000 cycles with two multigrid levels and took 165 sec on the Cray. The residuals were reduced eight decades so the solution could have been stopped at about 2000 cycles with a three-decade reduction in the residuals.

A radial duct with constant cross-sectional area was simulated by setting \( r = m \) and \( r \times h = \) constant. The solution is identical to the flat cascade results shown in Fig. 4.

A radial diffuser with 36 blades was simulated by setting \( r = m \) and \( h = \) constant. Figure 5 shows static pressure contours for an Euler solution with an inlet Mach number of 0.38 and an exit Mach number of 0.27. The plot is shown in polar coordinates. The contours show the diffusion of the flow due to the area change. The solution was run 2500 cycles with two multigrid levels and took 92 sec on the Cray. The residuals were reduced 6 decades so the solution could have been stopped at about 1250 cycles with a three-decade reduction in the residuals.

A centrifugal impeller with 36 blades was simulated by setting \( r = m \), \( h = \) constant, and a rotation rate of 1000 rad/sec (9550 rpm). Figure 6 shows Mach number contours for a turbulent solution with an inlet Mach number of 0.5 and an exit Mach number of 0.41. The impeller produces a total pressure ratio of 1.05. The plot is shown in polar coordinates with rotation upwards. The inlet whirl is zero but because of the rotation the inlet relative flow angle is 10.6°. This incidence desymmetrizes the flow and produces a pressure loading on the blade. At the exit this loading conflicts with the specified exit pressure and is responsible for the kinks in the contours there.

Figure 7 shows convergence histories for the previous example with various levels of multigrid. Convergence is taken to be a three decade drop in the maximum residual to 2x10^{-9}. For this example: two grids are 1.64 times faster than the fine grid and three grids are 3.12 times faster than the fine grid. CPU times are included on the figure.

The remaining results are for a nominally 6:1 total-pressure ratio centrifugal impeller and a matching vaned-radial diffuser. These components were designed by the Air Force for use in an auxiliary power unit and were modified and tested at NASA Lewis Research Center. The compressor has a diameter of 16.1 cm. It has 19 blades and was designed for a total pressure ratio of 5.9:1 with a mass flow of 1.033 kg/sec at 68 384 rpm. The diffuser has 27 blades and a diameter of 25.1 cm. Further details concerning the components can be found in Ref. 25.

Figure 8 shows the computational grid used for the impeller. The grid has 161 by 33 points (161 by 17 shown) with a normal spacing of 3x10^{-4} cm for inviscid flows and 5x10^{-4} cm for viscous flows. The leading edge is round with a radius of 0.045 cm and the grid points are 7° apart.

Figure 9 shows normalized radius \((RMS = r)\), stream-surface thickness \((BESP = h)\), and their product \((R^2B = rh)\) versus fraction of impeller chord. These values were taken from a meridional analysis.

Euler and Navier-Stokes solutions were each run 2000 cycles with two multigrid levels, reducing the maximum residual three decades. The Euler solution took 117 sec and the Navier-Stokes solution took 165 sec on the Cray.
Surface static pressure distributions for the impeller are compared in Fig. 10. Three solutions with identical mass flows are shown: an Euler solution (dashed), a Navier-Stokes solution (solid), and a panel solution (circles). The Euler and Navier-Stokes solutions were run with various exit pressures until the desired mass flow was obtained. The shapes of the three pressure distributions are similar but the panel solution has higher pressure levels since it is loss-free. The Euler solution has strong shock losses which lower the pressure levels. The Navier-Stokes solution has weaker shocks due to viscous smoothing of the leading edge, but blockage effects decrease the pressure levels overall.

Figure 11 shows relative Mach number contours for the Euler solution. The dashed line is the sonic line. At the inlet the relative Mach number is 0.91. At the leading edge the flow has 10° of incidence. This produces a large supersonic bubble with a peak Mach number of 1.92 on the suction (upper) surface. The bubble terminates with a normal shock that is smeared due to grid shearing in this region. There is also a tiny supersonic bubble on the pressure (lower) surface which is not visible at this scale.

Some interesting flow phenomena evident in Fig. 11 can be explained by the concept of a relative eddy. The flow through an impeller is predominantly inviscid and tends to remain irrotational. The blade row in Fig. 11 is rotating downward and so adds clockwise vorticity to the flow. To remain irrotational the flow develops a counterclockwise circulation within the passage. Thus the flow can be modeled as a superposition of a through-flow component and a component rotating counter to the blade row called a relative eddy. The effect of the relative eddy is to accelerate the flow on the suction surface and decelerate the flow on the pressure surface as can be seen in Fig. 11.

A more dramatic effect of the relative eddy is to sweep the unconfined flow beyond the trailing edge up and away from the blade in a spiral, with a slip line leaving the trailing edge. As a common example of slip from rotating machinery, consider the flow of water from a lawn sprinkler as viewed rotating with the sprinkler. The flow spirals up and opposite to the rotation of the sprinkler. It is emphasized that this is strictly an inviscid phenomena.

Figure 12 shows relative Mach number contours for a Navier-Stokes solution for the impeller. Here viscous effects reduce the peak suction-surface Mach number to 1.72 and the peak pressure-surface Mach number to just over 1.0. Both surfaces have small leading-edge separation bubbles that are barely resolved on this grid. The pressure surface quickly develops a thick boundary layer and the suction-surface boundary layer thickens after the shock. This blockage causes the viscous pressure levels to be lower than the inviscid levels in Fig. 10. The rotation energizes and diminishes the boundary layers on the radial portions of the blade.

Here again the relative eddy sweeps the flow off the trailing edge in a spiral. Since the flow does not follow the grid lines the thin-layer assumption may be invalid and diffusion across the wake may not be properly accounted for. This is one shortcoming of the present analysis. However, since the trailing-edge slip is an inviscid phenomena it is felt that the character of the solution is correct.

Figure 13 shows the computational grid for the radial diffuser vane. The grid has 145 by 33 points (145 by 17 shown) with a normal spacing of 2.5x10^-4 cm for inviscid flows and 6x10^-5 cm for viscous flows. The round leading edge has a radius of 0.025 cm and the grid points are 7° apart. At the trailing edge the actual vane is cut off at constant radius and the duct turns axially. For this analysis the trailing edge was sharpened and the duct was extended radially.

Figure 14 shows the normalized radius and stream-surface thickness versus fraction of vane chord. These values were taken from a combined meridional/boundary-layer analysis. Although the vanes have constant height, boundary layer blockage decreases the flow area by nearly 50 percent.

An Euler solution was run 4000 cycles with two multigrid levels, taking 214 sec on the Cray. A Navier-Stokes solution was run 2000 cycles with three multigrid levels, taking 179 sec. In each case the maximum residual was reduced three decades.

Surface static pressure distributions for the diffuser vane are compared in Fig. 15. Again, panel, Euler, and Navier-Stokes solutions are compared. The shapes of the pressure distributions are similar but the panel solution has higher pressure levels since it is loss-free. The Euler and Navier-Stokes solutions each have small supersonic bubbles terminated by normal shocks near the leading edge, and the losses lower the pressure levels. Additional blockage effects cause the viscous pressure levels to be even lower than the inviscid levels.

Figure 16 shows Mach number contours for the Euler solution. The inlet Mach number is exactly 1.0 but the radial component is only 0.29 so the flow is subsonic in character. Because of the increasing radius the total Mach number drops to about 0.83 near the leading edge. The flow has a slight positive incidence at the leading edge and the flow accelerates to a peak Mach number of about 1.5 on the suction surface. The tiny supersonic bubble (visible only as a black dot at this scale) terminates with a shock at about 0.26 percent chord. The pressure surface develops an even smaller supersonic bubble. The diffusion through the passage is evident in Fig. 16.

Figure 17 shows Mach number contours for the Navier-Stokes solution. Here viscous effects at the leading edge entirely suppress the formation of supersonic bubbles. The thick boundary layers that develop in the diffuser are obvious, but the flow remains attached on both surfaces.

Concluding Remarks

A quasi-three-dimensional Euler and Navier-Stokes analysis technique has been developed for blade-to-blade flows in turbomachinery. The analysis solves the thin-layer Navier-Stokes
scheme. The multigrid scheme typically reduces efficiency is achieved through use of vectorization, a spatially-variable time-step, and a multigrid scheme based on Johnson's revisions of Ni's scheme. The multigrid scheme typically reduces the CPU time required by the fine grid scheme by a factor of about three, for both inviscid and viscous flows.

Results for a model problem show the analysis to be viable for a variety of axial, radial, and rotating geometries. Results for a centrifugal impeller and a radial diffuser vane show that the analysis can predict a number of phenomena that are not accounted for in previous analyses. These phenomena include: leading-edge stagnation points, leading-edge separation, supersonic regions and shocks, blade-surface boundary layer growth, and trailing-edge slip lines.

It is thought that the ability to predict these phenomena rapidly for general geometries could make the quasi-three-dimensional analysis a useful tool for turbomachinery design. Furthermore, the quasi-three-dimensional analysis can provide insight into both physical and numerical problems that can be expected with fully three-dimensional problems in the future.

References


Figure 1. - Quasi-three-dimensional stream surface and coordinate system for a centrifugal compressor.

Figure 2. - Amplification factor for two-stage Runge-Kutta scheme.

\[ \lambda = 1.0 \]
\[ \mu = 1/(16\lambda) \]
Figure 3. - Computational grid for flat plate model problem.

Figure 4. - Static pressure contours for inviscid flat plate cascade model.
Figure 5. - Static pressure contours for inviscid radial diffuser model.

Figure 6. - Mach number contours for turbulent centrifugal impeller model.
Figure 7. - Multigrid convergence histories for turbulent centrifugal impeller model.

Figure 8. - Computational grid for 6:1 pressure ratio centrifugal impeller.
Figure 9. - Radius (RMSP) and stream surface thickness (BESP) for 6:1 centrifugal impeller.

Figure 10. - Static pressure distribution for 6:1 centrifugal impeller.
Figure 11. - Mach number contours for inviscid flow in 6:1 centrifugal impeller.

Figure 12. - Mach number contours for turbulent flow in 6:1 centrifugal impeller.
Figure 13. - Computational grid for 6:1 pressure ratio radial diffuser.

Figure 14. - Radius (RMSP) and stream surface thickness (BESP) for 6:1 radial diffuser.
Figure 15. - Static pressure distribution for 6:1 radial diffuser.

Figure 16. - Mach number contours for inviscid flow in 6:1 radial diffuser.
Figure 17. - Mach number contours for turbulent flow in 6:1 radial diffuser.
Development of an Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbomachinery

Rodrick V. Chima

National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Technical Memorandum


A rapid quasi-three-dimensional analysis has been developed for blade-to-blade flows in turbomachinery. The analysis solves the unsteady Euler or thin-layer Navier-Stokes equations in a body-fitted coordinate system. It accounts for the effects of rotation, radius change, and stream-surface thickness. The Baldwin-Lomax eddy-viscosity model is used for turbulent flows. The equations are solved using a two-stage Runge-Kutta scheme made efficient by use of vectorization, a variable time-step, and a flux-based multigrid scheme, which are all described. A stability analysis is presented for the two-stage scheme. Results for a flat-plate model problem show the applicability of the method to axial, radial, and rotating geometries. Results for a centrifugal impeller and a radial diffuser show that the quasi-three-dimensional viscous analysis can be a practical design tool.

Euler equations, Navier-Stokes equations, Multigrid, inviscid flow, viscous flow, turbomachinery

Unclassified - unlimited

STAR Category 01