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
DEVELOPMENT OF AN EXPLICIT MULTIGRID ALGORITHM FOR QUASI-THREE-DIMENSIONAL VISCOS FLUIDS IN TURBOMACHINERY

Rodrick V. Chima
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Abstract

A rapid quasi-three-dimensional analysis has been developed for blade-to-blade flows in turbo-
machinery. The analysis solves the unsteady Euler or thin-layer Navier-Stokes equations in a body-
fitting 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 flow-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 predominantly 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 dif-
cult than analysis of strictly axial-flow machines.

Analysis of axial-flow turbomachinery blade rows is usually simplified by modelling a blade section as a flat cascade. The governing equa-
tions for a flat cascade are the same two-
dimensional flow equations that are solved for
isolated airfoils, to cascade analyses often draw heavily on numerical techniques developed for
two-dimensional external flows. Examples of flat
cascade analyses include panel methods,2 potential
theory,3,4,5 finite-difference solutions,5,6 and Navier-
Stokes solutions.5,6

Flows in radial- or mixed-flow turbomachines are inherently three-dimensional, requiring speci-
fication 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
C.H. Wu in 1952.7 In Wu's model the flow is
assumed to follow an axisymmetric stream surface
(Wu's TS surface, Fig. 1). The radius and
thickness of the stream surface are assumed to be known functions of the streamwise distance.
These quantities are usually obtained from an
axisymmetric through-flow or meridional8

1

analysis,9 sometimes coupled with a boundary
layer analysis9 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 wall displacement thicknesses to be
modeled. This is similar to specifying a
tangential plane in boundary-layer theory.
Since the effects of radius change and stream-
surface thickness are modeled in this analysis,
the model is termed "quasi-three-dimensional." Examples of quasi-three-dimensional turbomachinery analyses include panel methods,10 stream-function
methods,11 potential methods,12 and Euler methods.13

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 thick-
nesses. The explicit MacCormack algorithm14 used
in Ref. 6 has been replaced with an explicit two-stage
Runge-Kutta finite-difference algorithm based on
the work of Jameson.15 Efficiency is achieved
by three means: vectorization, use of a variable
time-step, and by use of a multigrid scheme devel-
oped by Selle and modified by Johnson and
Chima.14,16

Governing Equations

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

$$2 \frac{dz}{dr} = 2 \text{ (1)}$$

and the $\theta$-coordinate is defined by:

$$\theta = \theta' - ut \text{ (2)}$$

where $z$ is fixed in space and $\theta$ rotates with the blade row with a constant velocity of $u$ at the
radius $r$ and the stream surface thickness $h$ is taken to be known functions of $r$. In this
coordinate system the dimensionless Navier-Stokes

equations may be written in the following nearly-
conservative form:

$$\partial \theta = \frac{\partial}{\partial z}(r \frac{\partial}{\partial r}(\alpha r)) + \frac{\partial}{\partial \theta}(r \frac{\partial}{\partial \theta}(\alpha r)) = K \text{ (3)}$$

where $\alpha$ is the viscosity and the other

variables are defined in the following table:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta$</td>
<td>Variable of the solution</td>
</tr>
<tr>
<td>$u$</td>
<td>Streamwise velocity</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
</tr>
<tr>
<td>$h$</td>
<td>Stream surface thickness</td>
</tr>
</tbody>
</table>

The equations are solved using the explicit
Two-step Runge-Kutta scheme.

$$2 \frac{dz}{dr} = 2 \text{ (1)}$$

and the $\theta$-coordinate is defined by:

$$\theta = \theta' - ut \text{ (2)}$$

where $z$ is fixed in space and $\theta$ rotates with the blade row with a constant velocity of $u$ at the
radius $r$ and the stream surface thickness $h$ is taken to be known functions of $r$. In this
coordinate system the dimensionless Navier-Stokes

equations may be written in the following nearly-
conservative form:

$$\partial \theta = \frac{\partial}{\partial z}(r \frac{\partial}{\partial r}(\alpha r)) + \frac{\partial}{\partial \theta}(r \frac{\partial}{\partial \theta}(\alpha r)) = K \text{ (3)}$$

where $\alpha$ is the viscosity and the other

variables are defined in the following table:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\theta$</td>
<td>Variable of the solution</td>
</tr>
<tr>
<td>$u$</td>
<td>Streamwise velocity</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
</tr>
<tr>
<td>$h$</td>
<td>Stream surface thickness</td>
</tr>
</tbody>
</table>

The equations are solved using the explicit
Two-step Runge-Kutta scheme.
\[
\begin{align*}
\begin{bmatrix}
\varphi_0 \\
\varphi_0 \\
\varphi_f \\
\varphi_e \\
\varphi_h \\
\varphi_{h*} \\
\varphi_{h*} + p \rho \\
\varphi_{h*} \left( a + p \right) + m_0 \\
\varphi_{h*} + p \rho \\
\varphi_{h*} + p \rho \\
\end{bmatrix}
&= R = h
\end{align*}
\]

\[\begin{align*}
K_2 &= 0 \\
\rho_1 &= 0 \\
\rho_2 &= 0 \\
\end{align*}\]

\[S = h\] (4)

\[\begin{align*}
K_0 &= (\varphi_0 + p) \frac{\partial \varphi_e}{\partial R} + (\varphi_{h*} + m_0) \frac{\partial \varphi_e}{\partial R}
\end{align*}\]

\[\begin{align*}
\rho_e/R &= 1 - \frac{1}{\varphi_{h*} + p} \\
\rho_{h*}/R &= 1 - \frac{1}{\varphi_{h*} + p}
\end{align*}\]

The viscous terms in the energy equation are:

\[\begin{align*}
\kappa_0 = \frac{1}{(y-1)Pr} \left( \frac{1}{\varphi_{h*} + m_0} \right) \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} \frac{\partial \varphi_e}{\partial R}
\end{align*}\] (5)

\[\begin{align*}
\kappa_0 = \frac{1}{(y-1)Pr} \left( \frac{1}{\varphi_{h*} + p} \right) \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} \frac{\partial \varphi_e}{\partial R}
\end{align*}\]

\[\begin{align*}
a = \sqrt{\kappa_0 \varphi_{h*}}
\end{align*}\]

The shear stress terms are:

\[\begin{align*}
\tau_{11} &= 2\alpha_0 \varphi_e \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} + \varphi_e \[\varphi_{h*} + m_0 \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} \frac{\partial \varphi_e}{\partial R}\]
\end{align*}\]

\[\begin{align*}
\tau_{22} &= 2\alpha_0 \varphi_e \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} + \varphi_e \[\varphi_{h*} + m_0 \frac{\partial \varphi_e}{\partial R} + \varphi_{h*} \frac{\partial \varphi_e}{\partial R}\]
\end{align*}\]

The equations are non-dimensionalized 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 Stok's hypothesis \(k = -2/3\) is valid, and that the effective viscosity may be written as:

\[\mu = \frac{k\varphi_0}{K_1} \varphi_{h*} \text{ turbulent}\]

Equations (2) to (6) are transformed from the \((\varphi, e)\) coordinate system to a general body-fitted \((x, y)\) coordinate system using standard methods. The thin-layer assumption is then used to eliminate viscous derivatives in the streamwise \((x)\) direction, thereby reducing computational overhead while retaining the capability of computing separated flows. The resulting equations are similar to those developed by Katsarou:

\[\begin{align*}
\frac{\partial \varphi_k}{\partial t} + \frac{\partial \varphi_k}{\partial x} &= 0 \\
\frac{\partial \varphi_k}{\partial x} &= 0
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]

\[\begin{align*}
\mu &= \frac{k\varphi_0}{K_1} \varphi_{h*}
\end{align*}\]
and

\[ \kappa = (\kappa_0 + p - \kappa_2 \omega^2) \rho / (p - \rho \omega_0^2) \frac{\partial \tau}{\partial \tau} \]

\[ S_4 = \frac{k}{1 - \mu^2} \left( \frac{\partial}{\partial \tau} \right)^2 \frac{1}{\rho^2} = \frac{1}{\rho^2} \frac{\partial^2 \rho}{\partial \tau^2} \]

In Eq. (8), the overbars denote a rescaling of the metric terms:

\[ \bar{\kappa} = \kappa / \kappa_0 \]

\[ \bar{\eta} = \eta / \eta_0 \]

\[ \bar{S}_4 = \frac{1}{\mu^2} \frac{\partial^2 \rho}{\partial \tau^2} \]

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

\[ J = \rho_0 \eta_0 \tilde{S}_4 \rho_0 \eta_0 - \rho_0 \eta_0 \tilde{S}_4 \]

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

\[ \tilde{S}_4 = \tilde{S}_4 \]

\[ \tilde{S}_4 = \tilde{S}_4 \]

\[ \bar{S}_4 = \bar{S}_4 \]

\[ \bar{S}_4 = \bar{S}_4 \]

\[ \bar{S}_4 = \bar{S}_4 \]

The relative contravariant velocity components \( \bar{u} \) and \( \bar{w} \) along the \( \zeta \) and \( \eta \) grid lines are given by:

\[ \bar{u} = \frac{\bar{u}}{\bar{u}} \]

\[ \bar{w} = \frac{\bar{w}}{\bar{w}} \]

The shear stress terms are found from Eq. (6) by replacing \( \tilde{\eta} \) with \( \bar{\eta}_0 \), and \( 1/\tilde{S}_4 \) with \( \tilde{S}_4 \).

The quasi-three-dimensional equations (Eqs. (7) to (12)) are similar to the two-dimensional equations solved in Ref. 5 except for the source term \( K_2 \), the ripples appearing in the \( \psi \) momentum equation, the rescaled metrics (Eq. (11)), and the relative velocity component \( \tilde{\eta} \) appearing in the convective velocities (Eqs. (8)).

Equations (7) to (12) reduce to the two-dimensional equations for constant \( \kappa \) and \( \eta \), and zero rotation. Note that Eqs. (7) to (12) are independent of the magnitude of the stream surface thickness \( \kappa \), so that any function \( h(n) \geq 0 \) may be used. The equations do depend on the magnitude of the radius \( r \) because of the \( 1/r \) terms scaling \( \bar{\kappa} \) and \( \bar{\eta} \) in Eq. (9).

For turbulent flows the two-layer eddy viscosity model developed by Baldwin and Lomax is used. In the \( \psi \) coordinate system the wall shear \( \bar{\tau}_w \) and vorticity \( \bar{\chi} \) required by the model are given by:

\[ \bar{\tau}_w = \rho \bar{\tau}_w \]

\[ \bar{\tau}_w = \rho \bar{\tau}_w \]

\[ \bar{\tau}_w = \rho \bar{\tau}_w \]

\[ \bar{\tau}_w = \rho \bar{\tau}_w \]

Equations (16) and (17) can be used with the isentropic relations to show that isentropic conditions are constant outside of the blade row. Evaluating Eqs. (16) at some point \( 1/2 \) and using Eq. (17) to eliminate \( a \) gives:
\[ w_2^2 = \left( \frac{n}{p Y_1 p_{21}} \right) w_2^2 \quad (18) \]

Now the isentropic relations and the definition of \( T_2^* \) can be used to evaluate \( \theta_2 \) giving:
\[ \theta_2 = \theta_1 \left( -\frac{V_2}{c_{p2}} \right) \frac{1}{\gamma - 1} \quad (19) \]

Substituting Eq. (19) into Eq. (18) gives:
\[ w_2^2 \left( 1 - \frac{V_2}{c_{p2}\theta_1} \right)^{\gamma - 1} - w_2^2 = 0 \quad (20) \]

Equation (20) is solved for \( w_2 \) at each grid point upstream of the blade using Newton iteration. Other flow quantities are then found using Eq. (17), the known total conditions, and the isentropic relations.

With the blade row \( \theta_2 \) constant, Eq. (27) is replaced with an assumption that the flow angle \( \alpha_2 \) varies linearly through the blade row. A derivation similar to that above gives:
\[ \left( \frac{\theta_2}{\theta_1} \right)^{\gamma - 1} - \frac{V_2}{c_{p2}} \cos \alpha_2 \quad (21) \]

which is solved at each grid point within the blade row. Once the flow conditions are known at the trailing edge, Eq. (28) can be used for the downstream region.

Boundary Conditions
At the inlet, total pressure, total temperature, and all \( \theta_2 \) are specified. For supersonic inflow the governing equations have one negative eigenvalue so that one variable at the inlet must be computed as part of the solution. Here a characteristic relation is used to extrapolate the upstream-running Riemann invariant to the inlet.

The axisymmetric moment equation may be written as:
\[ \frac{\partial}{\partial r} \left( r \frac{\partial}{\partial r} \right) + \frac{\partial}{\partial z} \frac{\partial}{\partial z} = \left( \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{1}{r^2} \right) \frac{\partial}{\partial r} \]
\[ (r+1)\rho \frac{\partial}{\partial r} \left( \frac{\partial}{\partial r} \right) \frac{\partial}{\partial z} = \frac{\partial}{\partial z} \left( \frac{\partial}{\partial z} \right) \frac{\partial}{\partial r} \]
\[ \frac{\partial}{\partial r} \left( r \frac{\partial}{\partial r} \right) + \frac{\partial}{\partial z} \frac{\partial}{\partial z} = \left( \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{1}{r^2} \right) \frac{\partial}{\partial r} \]
\[ (r+1)\rho \frac{\partial}{\partial r} \left( \frac{\partial}{\partial r} \right) \frac{\partial}{\partial z} = \frac{\partial}{\partial z} \left( \frac{\partial}{\partial z} \right) \frac{\partial}{\partial r} \]

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. First-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. The small a radius can cause the flow to be supersonic and to large a radius can cause the velocity to diverge near zero at the boundaries.

Blade surface pressure are found from the normal momentum equation:
\[ (1 + \rho Y_1 \theta_0) \frac{\partial}{\partial z} \left( \frac{\partial}{\partial z} \right) \frac{\partial}{\partial r} \]
\[ (r+1)\rho \frac{\partial}{\partial r} \left( \frac{\partial}{\partial r} \right) \frac{\partial}{\partial z} = \frac{\partial}{\partial z} \left( \frac{\partial}{\partial z} \right) \frac{\partial}{\partial r} \]
\[ (r+1)\rho \frac{\partial}{\partial r} \left( \frac{\partial}{\partial r} \right) \frac{\partial}{\partial z} = \frac{\partial}{\partial z} \left( \frac{\partial}{\partial z} \right) \frac{\partial}{\partial r} \]

where \( \frac{\partial}{\partial z} \) on the surface for viscous flows.

Periodic boundaries are solved like interior points.

**Fine-Grid Algorithm**

On the fine grid an explicit two-stage (first-order) Runge-Kutta algorithm based on the work of Zeman\(^{22}\) is used. It is given below as applied to Eq. (7).
Two-stage Runge-Kutta algorithm

\[
\begin{align*}
q^{(1)} &= q^n - \Delta t \theta q^n \\
q^{(2)} &= q^n - \Delta t \theta (q^{(1)} + q^{(2)})
\end{align*}
\]  \hspace{1cm} (26)

where

\[
\theta = 1.2
\]

\[
\begin{align*}
\theta_q &= \theta_q - \frac{1}{2} (\theta_q + \theta_{\ell}) \\
\theta_{\ell} &= \theta_{\ell} - \frac{1}{2} (\theta_q + \theta_{\ell})
\end{align*}
\]

Artificial dissipation

\[
\tilde{q}^{n+1} = \tilde{q}^{(1)} + \tilde{q}^{(2)}
\]  \hspace{1cm} (27)

Collect residuals

\[
\tilde{q}^{n+1} = \tilde{q}^{(1)} - \tilde{q}^{(2)}
\]  \hspace{1cm} (28)

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 Jameson and others. A one-dimensional version (z-direction) is given below. In two dimensions the dissipation is applied as a sequence of one-dimensional operators.

\[
D = C \left( \frac{q^{n+1} - q^n}{\Delta t} \right)
\]

where

\[
C = \frac{2}{\Delta t} \sqrt{1 + \nu} \\ \ \ \ \ \ \ \ \ (29)
\]

\[x^2 = 0.1(1)
\]

\[x^4 = 0.1(10)
\]

The terms in the coefficient C balance similar terms in Eq. (26). In smooth regions of the flow the dissipative terms are of third order and thus do not detract from the formal second-order accuracy of the fine-grid scheme. In regions of the flow where the second difference of the pressure is large, the second-difference dissipation becomes locally of first order. Note that in other work including Ref. 15 the term \((\frac{1}{2\nu})\) is commonly divided by an average pressure. This is not done here because pressures through a centrifugal compressor can increase by factors on the order of five, which would decrease the dissipation correspondingly.

Stability Analysis

A stability analysis of the fine-grid algorithm is performed in two parts. The first part examines the model problem considered by Jameson and is used here to choose the parameters in the two-stage scheme. The model problem is the one-dimensional convection equation with third-order artificial dissipation:

\[
q_t + q_x = 0
\]  \hspace{1cm} (30)

Applying the two-stage scheme (Eqs. (26) and (27)) gives:

\[
\begin{align*}
q^{(1)} &= q^n - \frac{1}{2} (q_{x+1} - q_{x-1}) \\
q^{(2)} &= q^n - \frac{1}{2} (q_{x+1} - q_{x-1})
\end{align*}
\]

where \(x = \sigma t \) is the Courant number.

If we consider a Fourier component of the solution

\[
q = q_0 e^{i\sigma t} = q_0 e^{i\lambda x}
\]  \hspace{1cm} (32)

where \(\lambda = \sigma / \nu\) is the wave number, \(\nu = \sigma / \lambda\) is the amplification factor is given by:

\[
\lambda^2 = 1 - 4(1 - \cos \xi) \\ \ \ \ \ \ \ \ \ (33)
\]

\[
x = (1 - \sin x) - \sin x^2 \sin x^2
\]  \hspace{1cm} (34)

\(x = \left(1 - \sin x\right) / \sin x^2\)

\(a\) damping factor \(x\) characteristic polynomial for undamped scheme.
The damping coefficient \( g \) is chosen such that
\[ g = 0 \text{ at } t = 0, \text{ giving } u = 1/16 \text{ in } \frac{2}{1}x. \]

An undamped \( n \)-stage scheme can only be stable to \( x = n -1 \), so \( n \) is taken to be one
while \( a_1 \) is chosen. Figure 2 shows a plot of
Eq. (21) for several values of \( a_1 \), with \( x = 1 \) and \( u = 1/16 \). It can be seen that the two-
stage scheme is stable for 0.5 < \( a_1 \) ≤ 1.2, with
\( a_1 = 1.2 \) giving the best damping over the range of
frequencies. For \( a_1 = 1.2 \) and \( u = 1/4 \), it can be seen that the two-stage scheme is
stable for Courant numbers \( x \leq 1.1 \). 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:
\[ \frac{\Delta t}{CFL} \leq \frac{1}{2} \left[ \frac{\Delta x}{\Delta t} \left( \frac{\Delta y}{\Delta t} \right)^2 \right]^{-1} \]
where
\[ \Delta x = x_0 \text{ and } \Delta y = y_0 \]

which is implemented as:
\[ \frac{\Delta t}{CFL} \leq \left[ \frac{\Delta x}{\Delta t} \left( \frac{\Delta y}{\Delta t} \right)^2 \right]^{-1} \]

The Euler equations are used to replace the third
term.
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha + \beta - \gamma}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

Introducing the space and time derivatives and
using backward differencing in space:
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

and finally
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

where
\[ \alpha = \frac{\alpha^0 - \alpha_i}{\Delta x^0 - \alpha^0} \text{ and } \gamma = \frac{\gamma^0 - \gamma_i}{\gamma^0 - \gamma^0} \]

Equation (36) is implemented on a coarse grid with
spacing \( \Delta x \) and \( \Delta t \) and time step
\( \Delta t \). The coarse-grid scheme is
accelerated using the fine-grid algorithm.

The hybrid algorithm originates from \( x = 16 \)
and modified by Johnson and Chiang \( x = 16 \)
to accelerate convergence to steady state.
Equation (36) 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.

The multi-grid algorithm is based on the
acceleration scheme applied to the coarse grid. \( x = 16 \)
and modified by Johnson and Chiang \( x = 16 \) is used
to accelerate the convergence of the fine-grid
algorithm. \( x = 16 \) scheme is basically a one-step Lax-Wendroff
algorithm used to accelerate his own fine-grid Euler scheme. \( x = 16 \)
Johnson adopted \( x = 16 \) method to other fine-grid
schemes including Macromax's scheme. \( x = 16 \) is
also used for viscous flows by demonstrating
that dissipative terms need not be included on
the coarse grids, thus the multi-grid scheme used
here is based solely on the Euler equations. It is
entirely independent of the viscous term, the
turbulence model, and the artificial dissipation
used on the fine grid.

One-step Lax-Wendroff schemes including \( x = 16 \)
scheme require temporal derivatives of the flux
vectors. These terms are computed as the Jacobian
matrix of the flux vector times the temporal
difference 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. \( x = 16 \) This "flux-based"
scheme is considerably simpler than \( x = 16 \) original scheme.

The flux-based multi-grid scheme is derived by
expanding the fine-grid change \( \Delta a^{n+1} \), in a Taylor series,
\[ \Delta a^{n+1} = \Delta a^{n+1} + \Delta a^{n+1} \Delta t + O(\Delta t^2) \]

The Euler equations are used to replace the third
term.
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha \Delta x - \beta - \gamma}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

Introducing the space and time derivatives and
using backward differencing in space:
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

and finally
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

where
\[ \alpha = \frac{\alpha^0 - \alpha_i}{\Delta x^0 - \alpha^0} \text{ and } \gamma = \frac{\gamma^0 - \gamma_i}{\gamma^0 - \gamma^0} \]

Equation (36) is implemented on a coarse grid with
spacing \( \Delta x \) and \( \Delta t \) and time step
\( \Delta t \). The coarse-grid scheme is
accelerated using the fine-grid algorithm.

The hybrid algorithm is based on the
acceleration scheme applied to the coarse grid. \( x = 16 \)
and modified by Johnson and Chiang \( x = 16 \) is used
to accelerate the convergence of the fine-grid
algorithm. \( x = 16 \) scheme is basically a one-step Lax-Wendroff
algorithm used to accelerate his own fine-grid Euler scheme. \( x = 16 \)
Johnson adopted \( x = 16 \) method to other fine-grid
schemes including Macromax's scheme. \( x = 16 \) is
also used for viscous flows by demonstrating
that dissipative terms need not be included on
the coarse grids, thus the multi-grid scheme used
here is based solely on the Euler equations. It is
entirely independent of the viscous term, the
turbulence model, and the artificial dissipation
used on the fine grid.

One-step Lax-Wendroff schemes including \( x = 16 \)
scheme require temporal derivatives of the flux
vectors. These terms are computed as the Jacobian
matrix of the flux vector times the temporal
difference 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. \( x = 16 \) This "flux-based"
scheme is considerably simpler than \( x = 16 \) original scheme.

The flux-based multi-grid scheme is derived by
expanding the fine-grid change \( \Delta a^{n+1} \), in a Taylor series,
\[ \Delta a^{n+1} = \Delta a^{n+1} + \Delta a^{n+1} \Delta t + O(\Delta t^2) \]

The Euler equations are used to replace the third
term.
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha \Delta x - \beta - \gamma}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

Introducing the space and time derivatives and
using backward differencing in space:
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

and finally
\[ \left( \frac{\Delta a^{n+1}}{\Delta t} \right)_i = \left[ \frac{\alpha}{\Delta x} + \frac{\beta}{\Delta t} \right] \left( \alpha \text{ and } \gamma \right) \]

where
\[ \alpha = \frac{\alpha^0 - \alpha_i}{\Delta x^0 - \alpha^0} \text{ and } \gamma = \frac{\gamma^0 - \gamma_i}{\gamma^0 - \gamma^0} \]

Equation (36) is implemented on a coarse grid with
spacing \( \Delta x \) and \( \Delta t \) and time step
\( \Delta t \). The coarse-grid scheme is
accelerated using the fine-grid algorithm.

The hybrid algorithm is based on the
acceleration scheme applied to the coarse grid. \( x = 16 \)
and modified by Johnson and Chiang \( x = 16 \) is used
to accelerate the convergence of the fine-grid
algorithm. \( x = 16 \) scheme is basically a one-step Lax-Wendroff
algorithm used to accelerate his own fine-grid Euler scheme. \( x = 16 \)
Johnson adopted \( x = 16 \) method to other fine-grid
schemes including Macromax's scheme. \( x = 16 \) is
also used for viscous flows by demonstrating
that dissipative terms need not be included on
the coarse grids, thus the multi-grid scheme used
here is based solely on the Euler equations. It is
entirely independent of the viscous term, the
turbulence model, and the artificial dissipation
used on the fine grid.

One-step Lax-Wendroff schemes including \( x = 16 \)
scheme require temporal derivatives of the flux
vectors. These terms are computed as the Jacobian
matrix of the flux vector times the temporal
difference 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. \( x = 16 \) This "flux-based"
scheme is considerably simpler than \( x = 16 \) original scheme.
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 at NASA Lewis Research Center. Indeed the Runge-Kutta computations were clocked at about 40 million floating point operations/second (40 miflops) for an Euler solution on a 113 by 25 grid. The efficiency of the multigrid computations 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 (184 by 32). The quasi-three-dimensional code requires about 50 percent more storage than the two-dimensional code.\(^5\)

Results

Results are presented for the following problems: a cascade of skin-flat plates with round leading edges, a centrifugal impeller, and a variable-diffusion 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 by representing a cascade of 10 flat plates with round leading edges, a centrifugal impeller, and 3 variable-diffusion. Both 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 by representing a cascade of 10 flat plates with round leading edges. The plane 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 111 by 25 points with 5 grids spacing around the leading edge circle and a normal spacing of 1.82 in. The viscous flow grid had 121 by 33 points with a normal spacing of 3.21 in. 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 abscissa is the flow and the ordinate is 1/x. This transformation preserves angles and is discussed in Ref. 10.

Grills are generated in an (m,n) coordinate system with some mean radius, and are independent of the local radius r and stream-surface thickness b. Values of r and b 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 b = constant. Figure 4 shows static pressure contours for an Euler solution with an inlet Mach number of 0.33. (Shade on the figures gives nautical values of Mach number. Yellow is the highest Mach number, and blue is the lowest Mach number.) The leading-edge velocity triangles given as input. Actual inlet conditions are not shown as the solution develops. The contours show the stagnation region and a mild acceleration due to blockage. Linearized results were obtained using the two-dimensional code.\(^5\)

The solution was run 5000 cycles with two multigrid levels and took 515 sec on the Cray. The residuals were reduced eight decades. 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 x = constant and r = constant. The solution is identical to the flat cascade results shown in Fig. 4.

A radial diffuser with 36 blades was simulated by setting x = m and b = constant, and a rotation rate of 1000 radians (360 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 0.61. The impeller produces a total pressure rise of 1.05. The plot is shown in polar coordinates with rotation upwards. The inlet Mach is zero but because of the rotation the inlet relative flow angle is 10°. This incidence degrades 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 convergence.

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 0.12%. For this example two grids are 1.6 times faster than the fine grid and three grids are 3.2 times faster than the fine grid. CPU times are included in the figure.

The remaining results are for a nominally 6:1 total pressure ratio centrifugal impeller with matching universal diffuser. These components were designed by the Air Force for use in an auxiliary power unit and were manufactured and tested at NASA Lewis Research Center. The compressor has a diameter of 10 in. and was designed for a total pressure ratio of 5.91 with a mass flow rate of 0.016 lb/sec. The impeller 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 33 shown) with a normal spacing of 3.21 in. for inviscid flows and 0.06 in. for viscous flows. The leading edge is rounded to a radius of 0.065 cm and the grid points are 7° apart.

Figure 9 shows normalized radius (RSP = r), stream-surface thickness b, and the product (bR = r) versus fractional impeller chord. These values were taken from a meridional analysis.\(^5\)

Euler and Navier-Stokes solutions were run for 1000 cycles with two multigrid levels, reducing the maximum residual eight decades. The Euler solution took 137 sec and the Navier-Stokes solution took 185 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 1.0. At the leading edge the flow has 0.90 of the path. This produces a large supersonic bubble with a peak Mach number of 1.90 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 edge. 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 edge. The effect of the relative edge is to accelerate the flow in the suction surface and decelerate the flow on the pressure surface as can be seen.

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

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 1.17 and the peak pressure-surface Mach number to just over 1.0. Both surfaces have small leading-edge separation bubbles that are barely resolved in this print. 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 losses to be lower than the inviscid levels in Fig. 10. The rotation energizes and diminishes the boundary layers on the radial portions of the blades.

Here again the relative edge sweeps the flow off the trailing edge and 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 vanes. The grid has 245 by 43 points [245 by 17 shown] with a normal spacing of 2.0x10^-3 cm for inviscid flows and 6.0x10^-5 cm for viscous flows. The round leading edge has a radius of 0.025 cm and the grid points are 0.02 apart. At the trailing edge the actual vane is cut off at constant radius to 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 experimental/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 super- sonic 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 supersonic 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 0.16 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 slip- to-laminar flows in turbomachines. The analysis solves the thin-layer Navier-Stokes
equations written in general coordinates for an axisymmetric stream surface, and accounts for the effects of blade-row rotation, radius change, and stream-surface thickness. It is believed that this is the first Navier-Stokes solution to include these effects.

The solution technique is a two-stage Rung-Kutta scheme based on the work of Jameson. 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 alone 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.
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 5. - Static pressure contours for inviscid radial diffuser model.

Figure 6. - Mach number contours for turbulent centrifugal impeller model.
Figure 1. - Multigrid convergence histories for turbulent centrifugal impeller model.

Figure 8. - Computational grid for 6:1 pressure ratio centrifugal impeller.
Figure 9. - Radius (RMS) 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 (RMS) and stream surface thickness (IESP) 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.
Title and Subtitle
Development of An Explicit Multigrid Algorithm for Quasi-Three-Dimensional Viscous Flows in Turbo-
machinery

Author(s)
Rodrick V. Chima

Performing-Organization Name and Address
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

Sponsoring-Organization Name and Address
National Aeronautics and Space Administration
Washington, D.C. 20546


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.

Key Words (suggested by Author(s)
Euler equations, Navier-Stokes equations, Multigrid, Inviscid flow, Viscous flow, Turbomachinery

Unclassified - unlimited
STAR Category 01

Security Classification of this report
Unclassified

Security Classification of this page
Unclassified

For sale by the National Technical Information Service, Springfield, Virginia 22161