NUMERICAL COMPUTATIONS OF TURBOMACHINERY
CASCADE TURBULENT FLOWS WITH SHOCKS BY
USING MULTIGRID SCHEME*

Chen Naixing  Zheng Xiaqing  Xu Yanji
The Institute of Engineering Thermophysics
Chinese Academy of Science
P.O. Box 2706, Beijing 100080
China

Abstract

A time-marching explicit finite volume method with multigrid scheme is presented for
solution of steady, two-dimensional, transonic flows in turbomachinery cascades. The tech-
niques for both Euler and Navier–Stokes solvers are the same. A new method for
discretization of viscous terms is suggested. A body-fitted coordinate system is generated by
a simple grid generation technique based on the staffs—springs system analogy method. Nu-
merical solution results for two turbomachinery cascades are presented and compared with
experimental data to demonstrate the accuracy, shock capturing capability and
computational efficiency of the present method.

Introduction

The flows past through turbomachinery are very com-
plex. In most instances, they are incompressible or
compressible, three-dimensional unsteady viscous and fre-
quently separated flows. The flows may be subsonic,
transonic or supersonic, with or without rotation, curva-
ture, heat transfer, phase-changes, leakage and
shock—boundary layer interaction. Because the progress
has been made during the past decade in the area of com-
puter science, numerical computational techniques, turbu-
lence modeling, grid generation technique etc. The numerical
computations provide an efficient tool for
turbomachinery design and analysis. There are various
numerical techniques used for numerical aerodynamics so-
lution of turbomachinery flows. In these techniques
time—marching methods
time—marching methods
time—marching methods
time—marching methods
time—marching methods
in implicit or explicit, are widely used in external and internal flows.
In the present paper are presented only the numerical com-
putations by using the explicit finite volume time—march-
ing method with multigrid scheme. In this paper the tech-
niques for the solution of the Euler and the Navier–Stoker
equations are the same. They are discretized by finite vol-
ume approximation. The discretized algebraic equations
are solved by Runge–Kutta technique. The multigrid
scheme is also used in the present method for acceler-
ating iteration convergence procedure. The various terms of
the full Navier–Stokes equations are expressed and
discretized by a more efficient method compared with pre-
vious ones. In this paper a grid generation technique is util-
ized for obtaining a body-fitted coordinate system. It is a
special method based on the concept similar to that used
for constitution of self—adaptive coordinate system. Calcula-
tions for both turbine and compressor cascades are car-
rried out and compared with experiments.

Governing Equations and Boundary Conditions

The governing equations can be found in most
references and text books. In the present paper the
two-dimensional Reynolds—averaged Navier–Stokes
equations, neglecting body force and heat sources, are
written in finite volume formulation as:

\[
\frac{\partial}{\partial t} \int_V \rho \, \dot{V} + \int_{\partial V} \rho \, \dot{W} \cdot \vec{n} = 0
\]

\[
\int_V \frac{\partial}{\partial t} \rho \, \dot{W} + \int_{\partial V} \rho \, \dot{W} \cdot \vec{n} = \int_{\partial V} \tau \cdot \vec{n} \, ds
\]  

(1)
\[ \int_{\partial V} f_\gamma \, dV + \int_{\partial S} f_\epsilon (\overrightarrow{W} \cdot \overrightarrow{n}) \, ds = \int_{\partial S} \left( \tau \cdot \overrightarrow{W} \right) ds \\
+ \int_{\partial S} \left( \frac{\partial p}{\partial n} \right) \, ds \\
p = (k - 1) \left( \frac{1}{\rho} \frac{d}{dt} \overrightarrow{W} \right)^2 \]

(2)

where: \( V \) denotes a fixed volume with boundary \( S \), \( W \) represents velocity vector, \( \rho, p, \epsilon \) represent density, pressure and total energy per unit volume, respectively. \( k \) is ratio of specific heats. \( \tau \) is the viscous stress acted on the surface with outer normal \( \overrightarrow{n} \).

\[ C = \frac{k}{k - 1} \left( \frac{\mu_l}{P_{rl}} + \frac{\mu_t}{P_{rt}} \right) \]

(3)

\( P_{ri} \) and \( P_{rt} \) denote laminar and turbulent Prandtl numbers, respectively. \( \mu_l \) and \( \mu_t \) denote molecular and turbulent dynamic viscosities, respectively. Molecular viscosity \( \mu_l \) is determined by Sutherland's law. Turbulent viscosity is estimated by turbulence model.

In solution of governing equations specification and enforcement of proper boundary conditions are essential to accurately capture the physics phenomena of the flow in turbomachinery. The unknown variables to be solved are velocity components, pressure, density, temperature or total energy per unit volume, etc. If there is no injection, a no-slip boundary condition is imposed on velocity for the Navier–Stokes equations on the solid walls. For inviscid fluid flow, i.e. for solution of Euler equation, the velocity vectors are parallel to the blade walls. The pressure at the wall can be obtained by assumption of zero normal gradient to the surface or by solution of the normal momentum equation. For temperature, either the blade surface temperature or the normal temperature gradient is given. For the heat transfer condition the temperature at blade surface is derived by heat transfer analysis of blade. For adiabatic wall of the present paper the normal temperature gradient is set to be zero.

The prescription of the conditions at inlet and exit boundaries far enough from the blade row depends on the flow regime, subsonic or supersonic. Here the total temperature and pressure and either the velocity components or the flow angles are given at inlet in the subsonic inlet case. The exit pressure is prescribed for cascade flow and its distribution is assumed to be uniform. The other flow variables are extrapolated from the interion point grid. For upstream and downstream regions enforcement of periodic boundary conditions is used.

**Turbulence Model**

Turbulence model is one of the essential ingredients for for and accurate computation of turbomachinery flows. For turbulent flow the independent variables, excluding velocity components and thermodynamics parameters, in the equations are \( \rho \overrightarrow{W}, \overrightarrow{W}, \rho \overrightarrow{W} ', \epsilon \). They are five additional variables have to be evaluated for two–dimensional flow. The most general transport equations for turbulence are the Reynolds stress equations. Many of the terms in these equations have to be modeled. Although the algebraic eddy–viscosity model, as indicated by many authors, is strictly valid only for two–dimensional flows with a mild pressure gradient, but for simplicity in the present paper is still used the Baldwin–Lomax (14) algebraic eddy viscosity turbulence model.

**Expressions of Viscous Terms**

Though the solution techniques for both N.S. and Euler equations are the same, but the main difference of exponents between the numerical solutions of these two equations is essential to estimate the viscous terms included in the Navier–Stokes equations. For improving solution efficiency a new method to discretize the viscous terms is proposed. It will be explained below.

The shear stress on the surface can be decomposed into two directions, i.e. normal and tangential, as follows:

\[ -\sigma \overrightarrow{W} \]

\[ \tau_n = -p + 2\mu \frac{\partial \overrightarrow{W}}{\partial n} - \frac{2}{3} \mu \text{div} \overrightarrow{W} \]

\[ \tau_\zeta = -\mu \left( \frac{\partial \overrightarrow{W}}{\partial \zeta} + \frac{\partial \overrightarrow{W}}{\partial n} \right) \]

(4)

where \( \overrightarrow{W}_n, \overrightarrow{W}_\zeta \) are the velocity components in normal, \( n \), and tangential, \( \zeta \), directions, respectively.

By setting a new pressure parameter \( p \) to be the sum of \( p + \frac{2}{3} \mu \text{div} \overrightarrow{W} \) and treating it as a separate term similar to the pressure in computation. Then, the remainder of shear stress is

\[ \tau = \mu \overrightarrow{n} \] + \mu \text{grad} \overrightarrow{W}_n \]

(5)

From the Cartesian coordinate system, \( x, y, \) adopted here, i.e.

\[ W_n = W \cdot n = n_x W_x + n_y W_y \]

and

\[ \frac{\partial W}{\partial n} = u \frac{\partial W_x}{\partial n} + v \frac{\partial W_y}{\partial n} \]

(7)

we have the formulas for expressing the gradient of normal velocity component, \( \text{grad} W_n \), and the first derivative of velocity vector on normal direction, \( \rho W' / \partial n \):

\[ \text{grad} W_n = n_x \text{grad} W_x + n_y \text{grad} W_y \]

(8)

\[ \frac{\partial W}{\partial n} = u \left( n \cdot \text{grad} W_x \right) + v \left( n \cdot \text{grad} W_y \right) \]

(9)

Then, the remainder of shear stress can be rewritten as:

\[ \tau = \mu \left( n_x \text{grad} W_x + n_y \text{grad} W_y + u \left( n \cdot \text{grad} W_x \right) + v \left( n \cdot \text{grad} W_y \right) \right) \]
\[ u_j (n \cdot \text{grad} W) \]

According to the definition of gradient of a scalar, \( f \),
\[ \text{grad} f \approx \frac{1}{V} \sum_{i=1}^{4} f_i \Delta S_i, \]  

where \( V \) is order of surface of a control volume, \( n \) is number of surfaces, in general speaking, \( n \) equals four for two–dimensional and six for three–dimensional flow problems, respectively. Applying Eq.(11) to the velocity components,
\[ \text{grad} W = \frac{1}{V} \sum_{i=1}^{4} 0 + \left[ u_x (W_x \Delta S_x) + u_y (W_y \Delta S_y) \right], \]  

and denoting
\[ G_i = \frac{1}{V} \sum_{i=1}^{4} (W_i \Delta S_i), \]  

and \( i,j = 1,2 = x,y \),
we have:
\[ \text{grad} W = \frac{1}{V} \sum_{i=1}^{4} G_i u_j \]  

Finally, the shear stress remainder can be expressed by the following expression:
\[ \tau = \mu \frac{n}{\Delta n} + \mu \text{grad} W, \]  

\[ = \sum_{i=1}^{4} \tau_i u_i \]  

\[ = \mu \sum_{i=1}^{4} \left( \sum_{j=1}^{4} G_{ij} \right) u_i \]  

where:
\[ G_{ij} = \frac{1}{V} \sum_{i=1}^{4} (W_i \Delta S_j + W_j \Delta S_i), \]  

now we are coming back to the pressure parameter, \( \hat{p} \). According to the definition of divergence of velocity vector, \( \vec{W} \), we have:
\[ \vec{W} = \lim_{v \to 0} \frac{\int n \cdot \vec{W} ds}{V} \]

\[ = \frac{1}{V} \sum_{i=1}^{4} \left[ (W_x \Delta S_x)_i + (W_y \Delta S_y)_i \right] \]

\[ = \frac{1}{V} \sum_{i=1}^{4} G_i \]  

Then, the pressure parameter can be obtained as:
\[ \hat{p} = p + \frac{1}{V} \sum_{i=1}^{4} G_i \]  

It is enable to summarize up that the computation of shear stress is mainly only the calculation of the \( G \)–parameters.

Grid Generation Technique

In general speaking, grid cell formulation much influences the convergency and accuracy of computations.
In the present paper a noval method (ASSS method), based on analogy of staffs-springs system, for generating a body-fitted coordinate system is proposed. The body-fitted coordinate system generated by this method at the wall approximately keeps to be orthogonal. The first step is to obtain the geometrically central x-coordinate line. Then the x-coordinate lines are formed by proportion of the straight lines connecting the grids points of the central line with the corresponding grid points of both suction and pressure surfaces (see Fig.1). These x-coordinate lines are fixed in the later grid generation procedure. The third step is to generate the x-coordinate lines. On the one of the x-coordinate lines, for example the j-th x-coordinate line, the spaccings between two grid points are replaced by springs. The effect of the approporite grid points at both walls on the position of the grid point of j-th x-coordinate line is similar to the effect of two bended staffs, the other ends of them are fixed at the pressure and section surfaces with the bending angles of \( \phi_{upp} \) and \( \phi_{low} \), respectively. The following balance equation for each grid point we have

\[
-C_{i-1} (S_{i} - S_{i-1}) + C_{i} (S_{i+1} - S_{i}) + M_{upp} \phi(J - j) - M_{low} \phi(j - 1) = 0
\]  

(19)

where: \( C_{i} \) and \( C_{j} \) denote the elasticity coefficients of the \( i-th \) and \( (i+1)-th \) springs, respectively. \( M_{upp} \) and \( M_{low} \) denote moments of force from upper and lower walls, respectively. \( \phi(J - j) \) and \( \phi(j - 1) \) are damping coefficients of upper grid point, \( J \), and lower grid point, \( 1 \). Because the bending angles, \( \phi_{upp} \) and \( \phi_{low} \), and position of \( S_{i} \) are related to each other, iterations for solving the above equations require greater CPU time. It can be simplified by the following approximation.

\[
\phi_{upp} = (S_{i+1} - S_{i}) / D_{upp}
\]

\[
\phi_{low} = (S_{i} - S_{i-1}) / D_{low}
\]

(20)

where \( S_{i+1} \) and \( S_{i-1} \) are the crosssection points between the \( j-th \) x-coordinate line and the normals to upper and lower walls, respectively. Then the Eq.(19) can be rewritten as:

\[
-C_{i-1} (S_{i} - S_{i-1}) + \left[ C_{i-1} + C_{i} \right] + \frac{M_{upp} \phi(j - 1)}{D_{upp}} S_{i-1} = \frac{M_{upp} \phi(j - 1)}{D_{upp}} S_{i+1}
\]

\[
\frac{M_{low} \phi(j - 1)}{D_{low}} S_{i-1} - C_{i} S_{i+1} = \frac{M_{low} \phi(j - 1)}{D_{low}} S_{i+1}
\]

\[
(i = 2, \ldots, J - 1; \quad j = 2, \ldots, J - 1)
\]

The coefficient \( C_{i,j} \) is chosen to be an average value of \( C_{i,j-1} \) and \( C_{i,j+1} \) for smoothness of the coordinate lines generated, i.e.

\[
C_{i,j} = \frac{1}{2} (C_{i,j-1} + C_{i,j+1})
\]

\[
-C_{i,j-1} + 2C_{i,j} - C_{i,j+1} = 0
\]

(22)

The boundary coditions at lower and upper walls of above equation are

\[
C_{1,j} = 1 / (S_{i+1} - S_{i}),
C_{J,j} = 1 / (S_{i+1} - S_{i})
\]

(23)

respectively.

The values of \( C_{i,j} \) are obtained by solution of Eq.(22).

Finally, The x-coordinate lines are obtained by solving Eq.(21). In this method iteration procedure is not required. The final grid system of a turbine blade cascade is shown in Fig.3. It is shown the the x-coordinate lines is closed to the normals.

**Discretization and Solution**

The integral equations, Eq.(1), can be discretized with summation around the four sides (numbered as 1) of the rhombus cell and the variables are defined at the cell center, i.e.

\[
\frac{dp}{dt} = -\frac{1}{V} \sum_{i=1}^{4} p_i (\vec{W} \cdot \Delta S_i)
\]

\[
\frac{d(\rho W^i)}{dt} = -\frac{1}{V} \left[ \sum_{i=1}^{4} (pW^i)^i (\vec{W} \cdot \Delta S_i) + \sum_{i=1}^{4} (\rho \Delta S^i) \right] \frac{\vec{\rho}}{\Delta S_i}
\]

\[
\frac{d(\rho W^i)}{dt} = -\frac{1}{V} \left[ \sum_{i=1}^{4} (pW^i)^i (\vec{W} \cdot \Delta S_i) + \sum_{i=1}^{4} (\rho \Delta S^i) \right] \frac{\vec{\rho}}{\Delta S_i}
\]

\[
\frac{d(e + p)}{dt} = -\frac{1}{V} \left[ \sum_{i=1}^{4} \left( e + p \right) (\vec{W} \cdot \Delta S_i) + \sum_{i=1}^{4} \left( \tau \cdot W^i \right) \Delta S^i \right]
\]

(24)

where \( V \) is cell area, \( \Delta S_i = (\Delta S_{xi}, \Delta S_{yi}) \) is the length vector of cell surface for twodimensional problem. \( W_z \) and \( W_y \) are the velocity components in xy directions, respectively. \( \tau = (\tau_x, \tau_y)^{\tau} \) and \( p \) are viscous stress and pressure parameter, respectively. They will be discussed later.

Eq.(24) can be written in general form as:

\[
\frac{d\Phi}{dt} + Q = 0
\]

(25)

Since the variables are stored at the cell center, their values at cell surfaces are found by simple averaging of the quantities \( \Phi \) from the adjacent cell centers. This kind of finite volume spatial discretization reduces to central difference scheme which is formally second order accurate for smooth grid. In order to suppress its well-known tendency for odd-even points decoupling, to capture shocks and to
minimize pre- and post-shock oscillation, an adaptive dissipation term \( D \) is added to the system, i.e.

\[
\frac{d\Phi}{dt} + Q - D = 0 \tag{26}
\]

The discretization equations, Eq.(25), are solved using the modified multi-stage Runge–Kutta time-marching method \(^{(15)}\). In the present paper the multigrid scheme is also applied to solve the algebraic equations. The auxiliary meshes are formed by doubling the mesh spacing and the informations of the flow variables are transferred to a coarser grid by the following principle:

\[
\Phi_{2h}^{(0)} = \sum_{i=1}^{4} (V_{i}^{h} \Phi_{i}^{h} / V_{2h}) \tag{27}
\]

where the subscripts denote the values of mesh spacing parameter. Then a forcing function is defined as:

\[
P_{2h} = \sum_{i=1}^{4} (Q_{h}^{i} \Phi_{i}^{h} - D_{h} \Phi_{i}^{h}) - (Q_{2h} \Phi_{2h}^{(0)} - D_{2h} \Phi_{2h}^{(0)}) \tag{28}
\]

The final equation, updating on a coarse grid, can be rewritten as

\[
\frac{d\Phi}{dt} + Q - D + P = 0 \tag{29}
\]

The process is repeated on successively coarser grids. Finally, then correction calculated on each grid is passed back to the next finer grid by bilinear interpolations.

**Results**

Numerical solution method for transonic turbine and supersonic compressor cascades are presented and compared with experimental data. The computational grids are generated by the use of ASSS method.

![Fig.5. Mach number distribution on suction and pressure surfaces for the case of \( M_{2h} = 0.985 \)](image5)

![Fig.6. Density contour calculated (88 \( \times \) 24)](image6)

**VKI–LS59 turbine cascade** \(^{(16)}\)

The first example solution is for a transonic turbine cascade shown in Figs.3. This blade cascade was tested in four European wind tunnels \(^{(16)}\). Its relative pitch is 0.71, inlet flow angle is \(-30^\circ\). The inlet total temperature, total pressure and turbulence intensity are 290K, 103360 Pa and 1\%, respectively. The outlet isentropic Mach number are about 0.8 and 0.985, respectively. The grid number is 124 \( \times \) 32. The comparisons between the Mach number distributions on suction and pressure surfaces, calculated and tested, for isentropic outlet Mach numbers, 0.8 and 0.985, are
plotted in Fig.4 and Fig.5. The density contours of the Mach number of 0.85 for grid points $88 \times 24$ and $124 \times 32$ are shown in Fig.6 and Fig.7, respectively. The body-fitted coordinate lines in the vicinity of the trailing edge are shown in Fig.8. The velocity vector scheme in the same region as Fig.8 is plotted in Fig.9. It is seem that the flow deviates from the geometric outer angle. There is a pair of vortexes just behind the trailing edge. The large one is located on the left side and the small one on the right side. The convergence histories, shown in Fig.10, have shown the rapid convergence of using multigrid scheme.

Supersonic inlet flow compressor cascade

The second example is a supersonic inlet flow compressor cascade tested in the supersonic compressor wind tunnel of the Institute of Engineering Thermophysics, Chinese Academy of Sciences. The inlet Mach number is 1.3. The inlet flow angle and blade stagger angles are 61.9 and 57.75 deg., respectively. The pressure ratio is 1.463 and the Reynolds number is $3.4 \times 10^6$. The grid system, shown in Fig.11, is also generated by the ASSS method. The grid number is $112 \times 48$ for both Euler equation and Navier–Stokes solutions, respectively. Fig.12 shows the
Fig. 12. Calculated Mach number distributions on pressure and suction surfaces in comparison with experiment.

Fig. 13. Density contour calculated by Euler solver.

Fig. 14. Density contour calculated by Navier–Stokes solver.

Fig. 15. Schlieren picture of supersonic inlet flow compressor cascade.

The actual passage shock presents a λ-type shape. Due to the shock wave separations are appeared on either side of the blade passage. Because of shock–boundary layer interaction starting from point S of the suction surface the pressure have greatly increased (see, Fig. 12) and separation has occurred (see, Fig. 16). Fig. 12 apparently shows the picture of separation. On the pressure surface boundary layer has slightly separated from point $S_1$ and reattached at point $R_1$. At a distance from first reattachment pressure is increased again, then the slight separation and reattachment are reapplied. It can be found from the enlarged velocity vector scheme (Fig. 17).

Isentropic Mach number distributions of both solutions in comparison with experimental data. Figs. 13, 14 are the calculated density contours of both solutions. Fig. 15 is a schlieren picture taken by Professors Yu Shen and Chen Jiagang of the Institute of Engineering Thermophysics. The numerical solution of the Navier–Stokes solver agrees more better with the experiment than of the Euler solver. The shock of the Euler solver locates near trailing edge of suction surface and hits on pressure surface in a post–mid position. The calculated and the experimental results are looking fairly similar. As shown in Figs. 12, 14 and 15 the
Conclusions

A time-marching explicit finite volume method with multigrid scheme has been presented for solving both Euler and Navier–Stokes equations with application to steady two-dimensional transonic flows in turbomachinery cascades. A new method expressing viscous terms of the Navier–Stokes equations has been also proposed. The method is more simple in comparison with previous methods. Based on analogy of staffs–springs system a rapid grid generation technique of a body–fitted coordinate system with approximately normal coordinate lines to the walls has been suggested. The discretized algebraic equations with multigrid scheme are solved by Runge–Kutta method.

Numerical solution results for two turbomachinery cascades have been presented and compared with experimental data to demonstrate the accuracy and computational efficiency of the analysis method. The method has good capability of capturing shock waves.

For simplicity, the presentation of the numerical method was applied to planar, two–dimensional cascade flows. The method, however, can be applied to three–dimensional turbomachinery flows.

References

(1) Lax, P.D. and Wendrot, B., "Difference Schemes for Hyperbolic Equations with High


(9) Nii, R.H., "A Multiple Grid Scheme for Solving


