A COMPOSITE FINITE-DIFFERENCE SCHEME FOR SUBSONIC/TRANSONIC FLOWS

A. Heydari, N. Amani-Fard and M. Golafshani

Department of Mechanical Engineering
Sharif University of Technology
Tehran, Iran

Abstract This paper presents a simple and computationally-efficient algorithm for solving steady two-dimensional subsonic and transonic compressible flow over an airfoil. This work uses an interactive viscous-inviscid solution by incorporating the viscous effects in a thin shear-layer. Boundary-layer approximation reduces the Navier-Stokes equations to a parabolic set of coupled, non-linear partial differential equations. The resulting system of partial differential equations is then solved using an efficient implicit finite difference scheme. A nonuniform mesh is used and the eddy viscosity concept models the turbulent Reynolds stress terms. The solution for the steady subsonic and transonic Euler equations is obtained using an upwind finite-volume scheme. The scheme is based on artificial viscosity in the governing equations to provide the necessary dissipation for numerical stability. The system of equations is linearized by a Newton method and the resulting fully coupled system of algebraic equations is solved. Convergence of the method is demonstrated to be robust, taking very few iterations to reach machine accuracy. Shock-Capturing methods extend the applicability of the scheme to situations with shocks. The two schemes are coupled and an iterative procedure is used to link the results of the inviscid and viscous flow fields. Computations are made for a series of flows. Results for NACA 0012 airfoil flows are presented and compared with experimental data and other computational results.

Key Words Fluids Flow, Boundary-layers, Finite Difference Methods, Subsonic-Transonic Flows

INTRODUCTION

Much progress has been made in computational fluid dynamics over the last 20 years toward the development of algorithms for flows over lifting bodies. It is the goal of computational fluid dynamics to eco-
nomically predict the viscous flow about three-di-

dimensional bodies. In doing so, two distinct approaches

have evolved. One approach has to do with solving
the Reynolds-averaged Navier-Stokes equations over
the entire flow. In the other approach the boundary-

layer equations along with an inviscid system simul-
taneously describe the viscous-inviscid interactions.
Currently, the Navier-Stokes approach offers the
advantage of being more general, while the viscous-
inviscid interaction approach can be computationally
more efficient. Investigations conducted by Cebeci et
al. [1] have shown an efficient prediction of the incom-
pressible flow over the airfoils at high and low
Reynolds number using a viscous-inviscid iterative
procedure.

In body fitted coordinates, the boundary-layer
approach removes the tangential stress term and
neglects the normal component of momentum equa-
tion. To retain a second order accurate solution, one
must independently prescribe the distribution of pres-

sure over the edge of the boundary-layer.

The inviscid steady subsonic and transonic flow
over the boundary layer is solved using an upwind
finite-volume scheme of the Euler equations. The
resultant distribution of the pressure is then used
interactively for describing the pressure distribution
within the boundary layer.

The following outlines the methodology used in
obtaining the interactive solution of the flow field
using the transonic inviscid-viscous interactive shock-
capturing method.

Viscous Flow Field
The boundary-layer flow for steady, two-dimen-
sional, compressible coupled flow over a two-di-

mensinal body is governed by the following

\[ \frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0 \]  (1)

\[ \rho \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} = \rho u \frac{du}{dx} + \frac{\partial}{\partial y} \left( \mu + \rho \frac{\partial u}{\partial y} \right) \]  (2)

\[ \rho \frac{\partial H}{\partial x} + \rho v \frac{\partial H}{\partial y} = \left( k + c_{p} \frac{\partial H}{\partial y} \right) \frac{dT}{dy} + u \left( \mu + \rho \frac{\partial u}{\partial y} \right) \]  (3)

subject to boundary conditions:

\[ y = 0, \ u = 0, \left( \frac{H}{\partial y} \right)_{w} = 0 \]  (4)

\[ y = \delta, \ u = u_{c}(x), \ H = H_{c} \]

The above conservation equations written in terms
of the Falkner-Skan variables, along with an eddy-

viscosity formulation for the transition and turbulent
regions of the flow yield a system of differential

equations. As equation 2 demonstrates, the velocity
profile at the outer edge of the viscous-layer needs to
be defined. This is obtained by solving the inviscid
flow-field, as described in the next section.

Here a mixing-length and eddy-viscosity formu-
lization based on that developed by Cebeci and Smith
[2] are used to model the Reynolds shear-stress
terms.

The solution of the resulting equations is obtained
using the Keller’s two-point, finite-difference method
described in Reference 3. In this method after trans-
forming the equations 1-3 into a system of first-order
differential equations, the derivatives are approxi-
mated by centered difference quotients and averages
centered at the midpoints of the calculation cells. In
this way, with the use of nonuniform meshes, second
order accuracy of the solution is obtained. The non-
linear system of algebraic equation is then linearized
by Newton’s method and solved using an efficient
block-tridiagonal factorization technique.

Inviscid Flow Field
The inviscid flow is governed by the Euler equation,
which in its two dimensional form is:

\[
\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = 0
\]  

(5)

Here \( Q \) is the vector of unknown variables, \( E \) is the flux vector of variables in the \( x \)-direction and \( F \) is the flux vector in \( y \)-direction, given as follows:

\[
Q = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho e_\parallel \end{bmatrix}
\]

(6)

\[
E = \begin{bmatrix} \rho u \\ \rho u^2 + P \\ \rho uv \\ (\rho e_\parallel + P) u \end{bmatrix}
\]

(7)

\[
F = \begin{bmatrix} \rho v \\ \rho uv \\ \rho v^2 + P \\ (\rho e_\parallel + P) v \end{bmatrix}
\]

(8)

Solution of the above equations starts by transforming the physical coordinates to the computational domain. The equations in their transformed form are:

\[
\frac{\partial \tilde{Q}}{\partial \tilde{t}} + \frac{\partial \tilde{E}}{\partial \tilde{\zeta}} + \frac{\partial \tilde{F}}{\partial \tilde{\eta}} = 0
\]

(9)

Where:

\[
\tilde{\zeta} = \zeta(x, y)
\]

(10)

\[
\tilde{\eta} = \eta(x, y)
\]

(11)

\[
\tilde{E} = \frac{1}{J}\left[ \zeta, E + \zeta, F \right]
\]

(12)

\[
\tilde{F} = \frac{1}{J}\left[ \eta, E + \eta, F \right]
\]

(13)

In which, \( J \) is Jacobian of the transformation. Using an algebraic mapping of the outer flow field enables us to solve the above system of equations in the transformed geometry.

The coordinates of the transformed domain is:

\[
\zeta = x/L
\]

(14)

\[
\eta = \frac{y - y_s(x)}{T - y_s(x)}
\]

(15)

Where \( y_s(x) \) describes the physical geometry of the airfoil and \( L \) and \( T \) are the length and height of the computation zone, respectively.

**NUMERICAL METHOD OF SOLUTION**

Here a quasi-second order upwind finite-volume method is used to numerically solve the inviscid flow field. This method has the advantages of providing stability for the explicit scheme used, which in addition dampens the dispersion of the numerically generated errors, notably those at the shock zone.

Utilizing this method of solution, changes the transformed equation to:

\[
\frac{\partial \tilde{Q}}{\partial \tilde{t}} + A \frac{\partial \tilde{Q}}{\partial \tilde{\zeta}} + B \frac{\partial \tilde{Q}}{\partial \tilde{\eta}} = 0
\]

(16)

Where:

\[
A = \frac{\partial \tilde{E}}{\partial \tilde{Q}} \quad B = \frac{\partial \tilde{F}}{\partial \tilde{Q}}
\]

(17)
In this finite difference form, the above equation becomes:

\[
\bar{Q}_{i,j}^{n+1} + \sum_{\iota} \left( A^* \text{Flux}_{i,j}^{+} \right) - \sum_{\iota} \left( B^* \text{Flux}_{i,j}^{-} \right) = 0
\]

where \( A^*, A, B^, B, \text{Flux}_{i,j}^{+}, \text{Flux}_{i,j}^{-}, \text{Flux}_{i,j}^{\xi}, \text{Flux}_{i,j}^{\eta} \) are defined as follows:

\[
A^* = \frac{X_a D_a X_a^{-1} + X_a |D_a| X_a^{-1}}{2}
\]

\[
A = \frac{X_a D_a X_a^{-1} - X_a |D_a| X_a^{-1}}{2}
\]

\[
B^* = \frac{X_b D_b X_b^{-1} + X_b |D_b| X_b^{-1}}{2}
\]

\[
B = \frac{X_b D_b X_b^{-1} - X_b |D_b| X_b^{-1}}{2}
\]

\[
\text{Flux}_{i,j}^{\xi} = A^* \left( \bar{Q}_{i+1/2,j}^{+} - \bar{Q}_{i-1/2,j}^{-} \right)
\]

\[
\text{Flux}_{i,j}^{\xi} = A \left( \bar{Q}_{i+1/2,j}^{+} - \bar{Q}_{i-1/2,j}^{-} \right)
\]

\[
\text{Flux}_{i,j}^{\eta} = B^* \left( \bar{Q}_{i,j+1/2}^{+} - \bar{Q}_{i,j-1/2}^{-} \right)
\]

\[
\text{Flux}_{i,j}^{\eta} = B \left( \bar{Q}_{i,j+1/2}^{+} - \bar{Q}_{i,j-1/2}^{-} \right)
\]

\[D_A \text{ and } D_B \text{ are the diagonal matrices and } X_A \text{ and } X_B \text{ are the characteristic vectors of matrices } A \text{ and } B, \text{ respectively. Also } \bar{Q}_{i+1/2,j} \text{ and } \bar{Q}_{i-1/2,j} \text{ represent the vector of unknown variables on two sides of cell } (i,j). \text{ Similarly, } \bar{Q}_{i,j+1/2} \text{ and } \bar{Q}_{i,j-1/2} \text{ are the values on the top and bottom surfaces of the calculation cell } (i,j). \text{ To assure a stable solution, the time step is chosen according to:}

\[
\Delta t \leq \min \left( \frac{\Delta \zeta}{\lambda_{\text{max}}} \right) \quad \left( \frac{\Delta \eta}{\lambda_{\text{max}}} \right)
\]

where \( \lambda_{\text{max}} \) is the maximum characteristic of matrices A and B. Also, superscripts '+' and '-' denote the direction of the characteristic vectors.

The finite volume approach assures a conservative form of solution for the transonic inviscid flow field equations. This provides us with the ability to capture shock as it happens and where it happens. The integrity of the isentropic flow solution is checked by examining the value of the total enthalpy of the inviscid flow field.

The parameter of interest is the velocity distribution over the outer edge of the boundary layer.

**RESULTS AND DISCUSSION**

Solution of the compressible Navier-Stokes equations using the thin-layer approximation near the surface of the airfoil requires the specification of the velocity profile on the exterior of the viscous layer. This is obtained by solving the Euler equation (5). After constructing the mesh through an algebraic formulation around the airfoil NACA 0012, the inviscid flow field is numerically solved by obtaining a finite-volume approximation of the governing equations. Figure 1 shows the calculated Mach contours at \( M_a = 0.85 \) and at a zero angle of attack. The figure indicates there exists a shock away from the leading edge of the airfoil.

![Figure 1. Computed Mach contours on a NACA-0012 wing with zero angle of attack and \( M_a = 0.85 \)](image-url)
A comparison between the calculated $C_p$ values and the results of Reference 4 is shown in Figure 2. It should be mentioned that the results of Reference 4 is obtained for a grid system of $190 \times 30$, while the present results is for a grid system of $60 \times 20$ mesh.

The results confirm the correctness of the numerical scheme used for the solution of the inviscid flow field. In order to obtain the thin-layer results one needs to specify the inviscid velocity profile over the outer edge of the viscous layer. Figure 3 shows the calculated inviscid velocity profile using the above mentional transonic finite-volume approach. This result serves as a means for defining the pressure distribution inside the boundary layer.

The interaction between the results of the inviscid code and the boundary layer calculation is through the displacement thickness and the induced blowing velocity on the upper edge of the boundary layer. The following describes the iterative procedure for obtaining the coupled inviscid-viscous results;

The calculation first starts by specifying the lower boundary condition over the surface of the rigid airfoil. The inviscid Euler equations are solved for the velocity distribution over the surface of the airfoil. Then the boundary layer equations 1-3 are solved, which in turn calculate the velocity and boundary layer profile over the airfoil. The calculated boundary layer thickness is then used for specifying the "new" airfoil geometry and along with the injected velocity profile is used as the lower boundary condition for the next iteration of the inviscid transonic code. The above procedure is repeated until a convergence between the two flow-field results is obtained. Figure 4 shows the calculated boundary layer thick-
ness. It indicates that after the first few cells, due to the amount of turbulence, the thickness of the boundary layer enlarges, until the flow separation happens. The calculated values of $C_{f}$ are shown in Figure 5. Here we see an expected trend for the calculated values of $C_{f}$.

**CONCLUSIONS**

The computational results of subsonic-transonic flow over an airfoil are presented in this paper. An explicit upwind finite-volume scheme solves the conservative Euler equation for determining the velocity (or pressure) distribution over the outer edge of the viscous boundary layer. This result is coupled with the solution of the boundary-layer equations using an incompressible, two-dimensional, implicit finite-difference code. An iterative procedure matches the results of the two procedures, until convergence of the calculated values of two schemes is obtained on the edge of the boundary-layer. The calculated values of the pressure distribution exhibits an excellent comparison with the results available in the literature. The computational results show reasonable values and an expected trend for the calculated values inside the boundary-layer.

**NOMENCLATURE**

- $\rho$: Density
- $u$: $x$-component of velocity
- $v$: $y$-component of velocity
- $\overline{\rho v}$: averaged value of $\rho v$
- $C_{p}$: specific heat
- $Pr_{t}$: turbulent prandtl number
- $\varepsilon_{m}$: eddy viscosity
- $K$: thermal conductivity
- $\rho_{i}$: density of inviscid flow
- $u_{i}$: $x$-component of inviscid velocity
- $H$: total enthalpy
- $\delta$: boundary layer thickness
- $e_{i}$: total energy
- $P$: static pressure
- $\zeta_{i}$: computational domain coordinates
- $J$: jacobian of coordinate transformation
- $\overline{Q}$: transformed form of $Q$ matrix
- $\overline{E}$: transformed form of $E$ matrix
- $\overline{F}$: transformed form of $F$ matrix
- $L$: length of body
- $Y_{s}(x)$: curve of body as a function of $x$
- $t$: transformed form of time
- Flux$_{\zeta}$: net flux at right boundary of a finite volume cell
- Flux$_{\zeta}$: net flux at left boundary of a finite volume cell
- Flux$_{\zeta}$: net flux at top boundary of a finite volume cell
- Flux$_{\zeta}$: net flux at bottom boundary of a finite volume cell
- $\Delta t$: time step
- $\Delta \zeta$: $p v$ step
- $\Delta \eta$: $\eta$ step
- $A$: jacobian matrix of $\overline{E}$ with respect to $\overline{Q}$
- $B$: jacobian matrix of $\overline{F}$ with respect to $\overline{Q}$
- $A^{*}$: computed $A$ at the right boundary of a finite volume cell
- $A^{*}$: computed $B$ at the top boundary of a finite volume cell
- $B^{*}$: computed $B$ at the top boundary of a finite volume cell
- $B^{*}$: computed $B$ at the bottom boundary of a finite volume cell

42 - Vol. 10, No. 1, February 1997

International Journal of Engineering
cell

$D_A$: a diagonal matrix whose elements are eigenvalues of $A$

$D_B$: a diagonal matrix whose elements are eigenvalues of $B$

$X_A$: matrix of eigenvectors of $A$

$X_B$: matrix of eigenvectors of $B$

$X_A^T$: inverse matrix of $X_A$

$X_B^T$: inverse matrix of $X_B$

$\lambda_{\text{max}}$: maximum of eigenvalues of $A$

$\lambda_{\text{max}}$: maximum of eigenvalues of $B$

$M_a$: mach number of free stream

REFERENCES


