RESEARCH NOTE

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-diemensional 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 extends 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

چکیده دراین مقاله معادلات جریان حول یک جسم متقارن محوری با فرمی شبیه به دماغه پرنده های مافوق صوت، در رژیم صوت گذر مورد حل قرار گرفت. هدف از انجام چنین مطالعه ای، تعیین تنشهای وارد برجسم براثر مواجه شدن با جریان فوق الذکر بود تا بتوان با کمک این تنشها، پارامترهای موردنیاز جهت آنالیز تنش روی آلیاژ دماغه را ارائه نمود و اطلاعات اولیه لازم جهت جایگزینی آلیاژی سهل الوصولتر از آلیاژ موجود در صنایع هوایی را حاصل آورد. با توجه به هدف فوق علاوه برتنشهای نرمال که از توزیع فشار حول جسم حاصل می آیند، تنشهای مماسی (لزج) نیز موردنیاز بود.بهمین دلیل معادلات لایه مرزی نازک که فرمی رایج بمنظور تعیین میدان لزج است انتخاب گردید. اما بدلیل نیاز به داشتن میدان فشار دراین معادلات، ابتدا معادلات اویلر در رژیم صوت گذر حول جسم متقارن محوری حل شد. روش عددی حل معادلات اویلر از نوع صریح (explicit) بوده و از متد upwind مرتبه دوم نیز دراین روش استفاده گردیده است. بمنظور تائید روش حل معادلات اویلر بطریق فوق، این معادلات برروی ایرفویل NACA0012 در ماخ 0.85 اعمال گردید و با مرجعی مورد مقایسه قرار گرفت. پس از تائید صحت این کد اویلری، در اعداد ماخ بحرانی، میدان فشار حول جسم متقارن محوری تعیین گردید. سپس با استفاده از کدی که جهت حل معادلات لایه مرزی نازک در دسترس بود، با استفاده از توزیع فشار حاصل از میدان غیر لزج، معادلات لایه مرزی حل شد و تنشهای لزج نیز بدست آمد. با استفاده از روش فوق که می توان آن را یک روش حل (لزج-غیرلزج) نامید، تمامی تنشهای موردنظر در هدف پروژه و اطلاعات اولیه جهت انتخاب و تحلیل آلیاژ دماغه حاصل گردید.

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-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 simultaneously 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] has shown an efficient prediction of the incompressible 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 equation. To retain a second order accurate solution, one must independently prescribe the distribution of pressure 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-dimensional, compressible coupled flow over a two-dimensional body is governed by the following equations:

$$\frac{\partial}{\partial x}(\rho u)+\frac{\partial}{\partial y}(\overline{\rho v})=0 \qquad (1)$$

$$\rho u\frac{\partial u}{\partial x}+\overline{\rho v}\frac{\partial u}{\partial y}=\rho_e u_e\frac{du_e}{dx}+\frac{\partial}{\partial y}\left((\mu+\rho\varepsilon_m)\frac{\partial u}{\partial y}\right) \qquad (2)$$

$$\rho u\frac{\partial H}{\partial x}+\overline{\rho v}\frac{\partial H}{\partial y}=\left[\left(k+c_p\rho\frac{\varepsilon_m}{Pr_t}\right)\frac{dT}{dy}+u\left((\mu+\rho\varepsilon_m)\frac{\partial u}{\partial y}\right)\right] \qquad (3)$$

subject to boundary conditions:

$$y=0,\ u=v=0,\ \left(\frac{\partial H}{\partial y}\right)_w=0 \qquad (4)$$

$$y=\delta,\ u=u_e(x),\ H=H_e$$

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 formulation 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 transforming the equations 1-3 into a system of first-order differential equations, the derivatives are approximated 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 accuarcy of the solution is obtained. The nonlinear 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;

$$\frac{\partial Q}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}=0 \tag{5}$$

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:

$$Q=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho e_t\end{bmatrix} \tag{6}$$

$$E=\begin{bmatrix}\rho u\\ \rho u^2+P\\ \rho uv\\ (\rho e_t+P)\,u\end{bmatrix} \tag{7}$$

$$F=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+P\\ (\rho e_t+P)\,v\end{bmatrix} \tag{8}$$

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

$$\frac{\partial \overline{Q}}{\partial t}+\frac{\partial \overline{E}}{\partial \zeta}+\frac{\partial \overline{F}}{\partial \eta}=0 \tag{9}$$

Where:

$$\zeta=\zeta(x,y)$$
$$\eta=\eta(x,y) \tag{10}$$

and

$$\overline{Q}=\frac{Q}{J} \tag{11}$$

$$\overline{E}=\frac{1}{J}[\zeta_x E+\zeta_y F] \tag{12}$$

$$\overline{F}=\frac{1}{J}[\eta_x E+\eta_y F] \tag{13}$$

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 domian is:

$$\zeta=x/L \tag{14}$$

$$\eta=\frac{y-y_b(x)}{T-y_b(x)} \tag{15}$$

Where $y_b(x)$ 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:

$$\frac{\partial \overline{Q}}{\partial \tau}+A\frac{\partial \overline{Q}}{\partial \zeta}+B\frac{\partial \overline{Q}}{\partial \eta}=0 \tag{16}$$

Where:

$$A=\frac{\partial \overline{E}}{\partial \overline{Q}} \qquad B=\frac{\partial \overline{F}}{\partial \overline{Q}} \tag{17}$$

In this finite difference form, the above equation becomes:

$$\frac{\overline{Q}_{i,j}^{n+1} - \overline{Q}_{i,j}^{n}}{\Delta t} + \frac{(A^{+}Flux_{\zeta}^{+} + A^{-}Flux_{\zeta}^{-})}{\Delta \zeta} + \frac{(B^{+}Flux_{\eta}^{+} + B^{-}Flux_{\eta}^{-})}{\Delta \eta} = 0 \tag{18}$$

where A^{+}, A^{-}, B^{+}, B^{-}, $Flux_{\zeta}^{+}$, $Flux_{\zeta}^{-}$, $Flux_{\eta}^{+}$ and $Flux_{\eta}^{-}$ are defined as follows;

$$A^{+} = \frac{X_A D_A X_A^{-1} + X_A |D_A| X_A^{-1}}{2} \tag{19}$$

$$A^{-} = \frac{X_A D_A X_A^{-1} - X_A |D_A| X_A^{-1}}{2} \tag{20}$$

$$B^{+} = \frac{X_B D_B X_B^{-1} + X_B |D_B| X_B^{-1}}{2} \tag{21}$$

$$B^{-} = \frac{X_B D_B X_B^{-1} - X_B |D_B| X_B^{-1}}{2} \tag{22}$$

$$Flux_{\zeta}^{+} = A^{+}\left(\overline{Q}_{i+\frac{1}{2},j} - \overline{Q}_{i-\frac{1}{2},j}\right) \tag{23}$$

$$Flux_{\zeta}^{-} = A^{-}\left(\overline{Q}_{i+\frac{1}{2},j} - \overline{Q}_{i-\frac{1}{2},j}\right) \tag{24}$$

$$Flux_{\eta}^{+} = B^{+}\left(\overline{Q}_{i,j+\frac{1}{2}} - \overline{Q}_{i,j-\frac{1}{2}}\right) \tag{25}$$

$$Flux_{\eta}^{-} = B^{-}\left(\overline{Q}_{i,j+\frac{1}{2}} - \overline{Q}_{i,j-\frac{1}{2}}\right) \tag{26}$$

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

$$\Delta t \leq Min\left[\frac{(\Delta \zeta)}{(\lambda_{maxA})}, \frac{(\Delta \eta)}{(\lambda_{maxB})}\right] \tag{27}$$

where λ_{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_{∞} = 0.85 and at a zero angle of attack. The Figure indicates there exists a shock away from the leading edge of the airfoil.

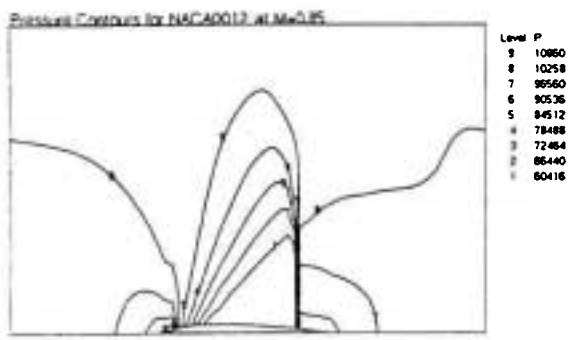

Figure 1. Computed Mach contours on a NACA-0012 wing with zero angle of attack and M_{∞} = 0. 85

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 × 30, while the present results is for a grid system of 60×20 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 mentioneal 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,

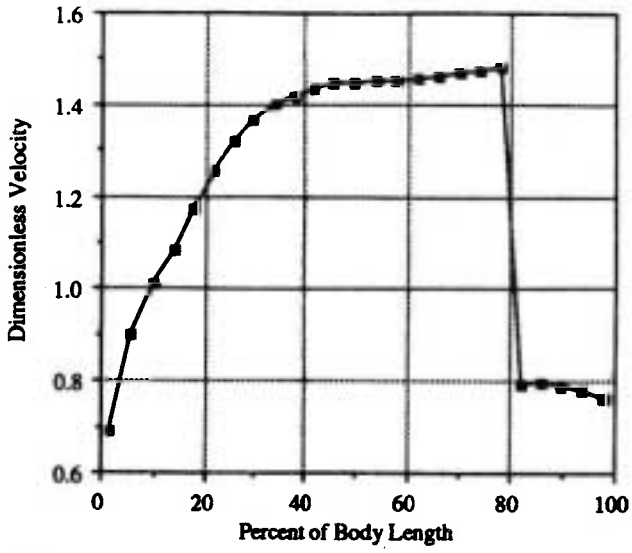

Figure 3. Computed velocity profile on the outer edge of the boundary layer over a NACA-0012 wing of Figure 1.

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-

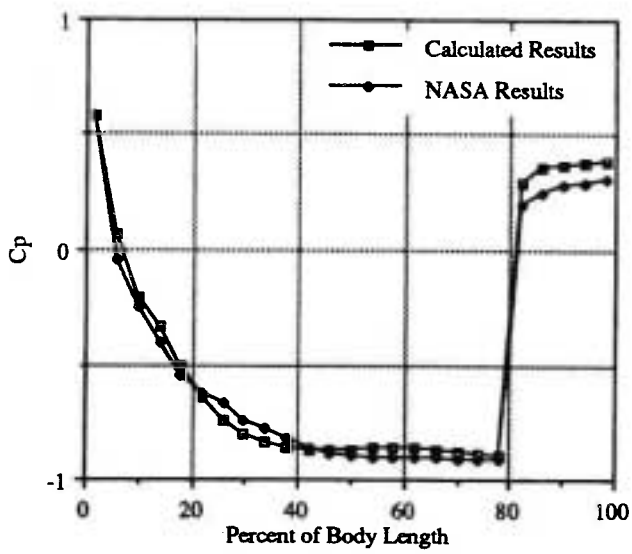

Figure 2. Comparison of the calculated values of C_p with those of Reference 4 for flow over a NACA-0012 wing of Figure 1.

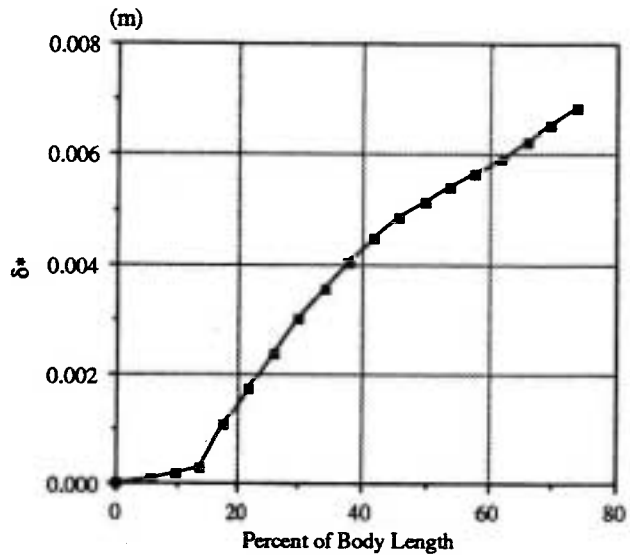

Figure 4. Computed displacement thickness on a NACA-0012 wing with zero angle of attack and $M_\infty = 0.85$

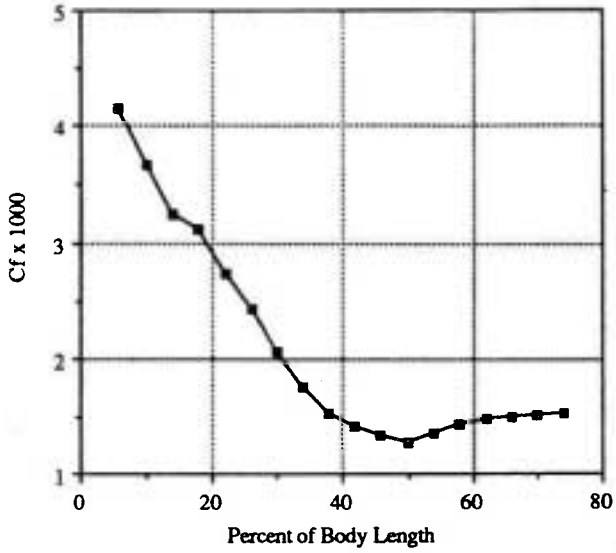

Figure 5. Computed values of C_f for flow over a NACA-0012 wing of Figure 1.

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

ρ : Density

u: x-component of velocity

v: y-component of velocity

$\overline{\rho v}$: averaged value of ρv

C_p: specific heat

Pr_t: turbulent prandtle number

ε_m: eddy viscosity

K: thermal conductivity

ρ_e: density of inviscid flow

u_e: x- component of inviscid velocity

H: total enthalpy

δ: boundary layer thickness

e_t: total energy

P: static pressure

ζ,η: 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

$Y_b(x)$: curve of body as a function of x

t : transformed form of time

Fluxζ^+: net flux at right boundary of a finite volume cell

Fluxζ^-: net flux at left boundary of a finite volume cell

Fluxζ^+: net flux at top boundary of a finite volume cell

Flux$\overline{\zeta}$, η: net flux at bottom boundary of a finite volume cell

Δt: time step

$\Delta\zeta$: $\overline{\rho v}$ step

$\Delta\eta$: η 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 fnite volume cell

B^-: computed B at the bottom boundary of a finite volume

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_B^{-1}: inverse matrix of X_A

X_A^{-1}: inverse matrix of X

X_B: inverse matrix of X_B

λ_{maxA}: maximum of eigenvalues of A

λ_{maxB}: maximum of eigenvalues of B

M_∞: mach number of free stream

REFERENCES

1. T, Cebeci and H. M. Jang, "Interactive Boundary-Layer Method for Unsteady Airfoil Flows: Quasisteady Model," J. Aircraft, Vol. 27, No. 8, (1990) 673-678.
2. T. Cebeci and A. M. O. Smith, "Analysis of Turbulent Boundary Layers", Academic, New York, (1974).
3. T. Cebeci and P. Bradshaw "Physical and Computational Aspects of Convective Heat Transfer", Springer-Verlag, New York, (1989).
4. C. Hirsch, "Computation of Internal and External Flows", Wiley, New York, (1988).