Indian J. pure appl. Math., **19** (1) : 66-74, January 1988

AN IMPLICIT NAVIER-STOKES SOLVER FOR TWO-DIMENSIONAL TRANSONIC SHOCK WAVE-BOUNDARY LAYER INTERACTION

N. S. MADHAVAN AND V. SWAMINATHAN

Applied Mathematics Division, Vikram Sarabhai Space Centre
Trivandrum 695022

(*Received* 5 *August* 1986; *after revision* 23 *July* 1987)

A code has been developed using an implicit numerical scheme for solving the two-dimensional unsteady Navier-Stokes equations for the transonic shock wave-boundary layer interaction problem. The method is an adaptation of the implicit Mac Cormack scheme for viscous compressible flows. The technique is illustrated by applying it to study the flow past an 18% symmetric circular arc airfoil. A coordinate transformation is made use of to facilitate direct application of the method. Numerical results obtained through the software, for surface pressure distribution and skin friction exhibit reasonable agreement with those got earlier by Deiwert using an explicit integral scheme.

1. INTRODUCTION

The problem of tronsonic shock wave-boundary layer interaction is of considerable importance in the context of aerodynamic design and has been currently engaging the attention of researchers[1,2]. The present paper is an attempt to numerically simulate the flow field for this problem with the aid of an implicit Navier-Stokes solver. The scheme is analogous to MacCormack's implicit numerical method for solving unsteady two-dimensional Navier Stokes equations[3-13].

The details of the flow field under investigation are depicted in Fig. 1. The freestream flow is subsonic and a supersonic flow region is generated between the

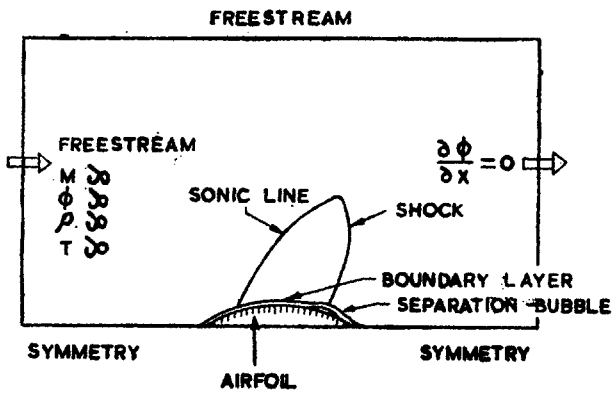

FIG. 1. Flow past circular arc airfoil.

leading and trailing edges of the airfoil. This supersonic flow becomes subsonic by passing through a standing shock. If the shock is sufficiently strong, boundary layer separation can occur.

2. Governing Equations and Transformation

The two-dimensional unsteady Navier-Stokes equations may be written in Cartesian coordinates in conservation form as

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0 \qquad \ldots(1)$$

where

$$U = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{bmatrix}, \quad F = \begin{bmatrix} \rho u \\ \rho u^2 + \sigma_x \\ \rho uv + \tau_{xy} \\ (e + \sigma_x) u + \tau_{yx} v - k \dfrac{\partial T}{\partial x} \end{bmatrix}$$

$$G = \begin{bmatrix} \rho v \\ \rho uv + \tau_{yx} \\ \rho v^2 + \sigma_y \\ (e + \sigma_y) v + \tau_{xy} u - k \dfrac{\partial T}{\partial y} \end{bmatrix} \qquad \ldots(2)$$

and where

$$\left.\begin{aligned} \sigma_x &= p - \lambda \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) - 2 \mu \frac{\partial u}{\partial x} \\ \sigma_y &= p - \lambda \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) - 2 \mu \frac{\partial v}{\partial x} \\ \tau_{xy} &= \tau_{yx} = - \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \end{aligned}\right\} \qquad \ldots(3)$$

in terms of density ρ, x- and y-velocity components u and v, viscosity coefficients λ and μ, total energy per unit volume e, coefficient of heat conductivity k and temperature T. Finally, the pressure p is related to the specific internal energy e_I and ρ by an equation of state $p\,(e_I, \rho)$, where

$$e_I = \frac{e}{\rho} - \frac{u^2 + v^2}{2}.$$

The transformation $\xi = x$, $\eta = y - R\,(x)$, where $y = R\,(x)$ represents the equation of the body surface, is made use of to obtain rectangular computational plane. Equation. (1) then becomes

$$\frac{\partial U}{\partial t} + \frac{\partial F'}{\partial \xi} + \frac{\partial G'}{\partial \eta} = 0 \qquad \ldots(4)$$

where

$$F' = F,\ G' = G - \frac{dR}{dx} F. \qquad \ldots(5)$$

For an 18% symmetric circular arc airfoil of chord length unity, for instance, the equation of the body surface, referred to rectangular coordinates with the x-axis along the chord and origin at one end of the chord, is given by

$$x^2 + y^2 - x + 2.68778y = 0 \qquad \ldots(6)$$

and, consequently, the transformed coordinate $\eta = \eta(x, y)$is obtained as

$$\eta = y - \tfrac{1}{2}\{-2.68778 + [7.22416 - 4x - 4x^2]^{1/2}\}. \qquad \ldots(7)$$

3. Implicit Finite-Difference Scheme

Following[3,7], an implicit predictor-corrector scheme may be defined for the numerical integration of the transformed equations as follows :

Predictor

$$\Delta U_{ij}^n = -\Delta t\left(\frac{D_+}{\Delta\xi} F_{ij}^n + \frac{D_+}{\Delta\eta} G_{ij}^n\right).$$

$$\left(I - \Delta t \frac{D_+}{\Delta\xi} |A|.\right)\left(I - \Delta t \frac{D_+}{\Delta\eta} |B|.\right) \delta \overline{U_{ij}^{n+1}} = \Delta U_{ij}^n$$

$$\overline{U_{ij}^{n+1}} = U_{ij}^n + \delta \overline{U_{ij}^{n+1}}.$$

Corrector

$$\Delta \overline{U_{ij}^{n+1}} = -\Delta t\left(\frac{D_-}{\Delta\xi} \overline{F_{ij}^{n+1}} + \frac{D_-}{\Delta\eta} \overline{G_{ij}^{n+1}}\right)$$

$$\left(I + \Delta t \frac{D_-}{\Delta\xi} |A|.\right)\left(I + \Delta t \frac{D_-}{\Delta\eta} |B|.\right) \delta U_{ij}^{n+1} = \Delta \overline{U_{ij}^{n+1}}$$

$$U_{ij}^{n+1} = \tfrac{1}{2}\left[U_{ij}^n + \overline{U_{ij}^{n+1}} + \delta \overline{U_{ij}^{n+1}}\right]. \qquad \ldots(8)$$

The matrices $|A|$ and $|B|$ have positive eigenvalues and are related to the Jacobians $A\left(=\frac{\partial F'}{\partial U}\right)$ and $B\left(=\frac{\partial G'}{\partial U}\right)$. Let S_ξ and S_η and their inverses denote the matrices that diagonalise A and B with $\mu = \lambda = k = 0$, i.e. with the viscous terms neglected. Then, with perfect gas relations, A and B may be arranged as

$$A = S_\xi^{-1} \Lambda_A S_\xi$$

and

$$B = S_\eta^{-1} \Lambda_B S_\eta \qquad \ldots(9)$$

where

$$S_\xi = \begin{bmatrix} 1 - \frac{\alpha\beta}{C^2} & \frac{u\beta}{C^2} & \frac{v\beta}{C^2} & -\frac{\beta}{C^2} \\ -uc + \alpha\beta & c - u\beta & -v\beta & \beta \\ -\frac{v}{\rho} & 0 & \frac{1}{\rho} & 0 \\ uc + \alpha\beta & -c - u\beta & -v\beta & \beta \end{bmatrix}$$

$$S_\eta = \begin{bmatrix} 1 - \frac{\alpha\beta}{c^2} & \frac{u\beta}{c^2} & \frac{v\beta}{c^2} & -\frac{\beta}{c^2} \\ \frac{\eta_x v - \eta_y u}{\rho\alpha_2} & \frac{\eta_y}{\rho\alpha_2} & -\frac{\eta_x}{\rho\alpha_2} & 0 \\ \beta_1\left(\alpha\beta - c\frac{\alpha_1}{\alpha_2}\right) & \beta_1\left(\frac{\eta_x c}{\alpha_2} - u\beta\right) & \beta_1\left(\frac{\eta_y c}{\alpha_2} - v\beta\right) & \beta_1\beta \\ \beta_1\left(\alpha\beta + c\frac{\alpha_1}{\alpha_2}\right) & -\beta_1\left(\frac{\eta_x c}{\alpha_2} + u\beta\right) & -\beta_1\left(\frac{\eta_y c}{\alpha_2} + v\beta\right) & \beta_1\beta \end{bmatrix} \quad \ldots(10)$$

$$\alpha_1 = \eta_x u + \eta_y v, \quad \alpha_2 = (\eta_x^2 + \eta_y^2)^{1/2}$$

$$\alpha = \tfrac{1}{2}(u^2 + v^2), \ \beta = \gamma - 1 \text{ and } \beta_1 = \frac{1}{\sqrt{2}\rho c}.$$

Λ_A and Λ_B are diagonal matrices with diagonal elements $(u, u + c, u, u - c)$ and $(\alpha_1, \alpha_1, \alpha_1 + c\alpha_2, \alpha_1 - c\alpha_2)$, respectively

The matrices $|A|$ and $|B|$ are now defined by

$$|A| = S_\xi^{-1} D_A S_\xi$$

and

$$|B| = S_\eta^{-1} D_B S_\eta \quad \ldots(11)$$

where D_A and D_B are diagonal matrices with diagonal elements λ_{Ai}, $i = 1, \ldots, 4$ and λ_{Bi}, $i = 1, \ldots, 4$, λ_{A1} and λ_{B1} being formed as

$$\lambda_{A1} = \text{Max}\left\{|u| + \frac{2\nu}{\rho\Delta\xi} - \frac{1}{2}\frac{\Delta\xi}{\Delta t}, 0.0\right\} \quad \ldots(12)$$

$$\lambda_{B1} = \text{Max}\left\{|\alpha_1| + \frac{2\nu}{\rho\Delta\eta} - \frac{1}{2}\frac{\Delta\eta}{\Delta t}, 0.0\right\}$$

and so on and

$$\nu = \text{Max}\left\{\mu, \lambda + 2\mu, \frac{\gamma\mu}{pr}\right\}. \quad \ldots(13)$$

For regions of flow in which Δt satisfies the explicit stability criterion, all λ_{Ai} and λ_{Bi} vanish and the set of difference equation (8) just reduce to those of the explicit method.

4. Computer Program

A computer program was developed in FORTRAN for carrying out flow field calculations by the above method for transonic flow past 18% thick circular arc biconvex airfoil and was made operational on the CDC Computer of Vikram Sarabhai Space Centre, Trivandrum.

5. Initial and Boundary Conditions

The airfoil which is initially at rest is impulsively started at time zero at the desired freestream Mach number and pressure. At a sufficient distance upstream of the leading edge (in this case 8 chord lengths), the flow is assumed uniform at freestream conditions ($u = u_\infty$, $v = 0$). A similar assumption is also made in respect of the flow at the upper boundary, 6 chord lengths away. The downstream boundary is positioned far enough downstream of the trailing edge (9 chord lengths away) so that the gradients in the flow direction can be regarded as negligible $\left(\frac{\partial \phi}{\partial x} = 0\right)$. The surface of the airfoil is taken to be impermeable and no-slip boundary conditions are assumed ($u = v = 0$). This is effected by assuming a fictitious layer of mesh inside the airfoil where the values are assigned after each iteration as $u_0 = -u_1$, $v_0 = -v_1$, $p_0 = p_1$ $T_0 = T_1$ and $\rho_0 = \rho_1$. No special treatment is required for separation which will naturally develop as the marching in time progresses. The airfoil is taken to be adiabatic, and ahead and behind it, the flow is taken to be symmetric.

6. Mesh Scheme

Figure 2 desribes the mesh scheme used in the physical domain, while Figure 3 gives scheme in the computational or transformed domain.

A 40 × 30 mesh system was used in the program. Along the ξ – direction, a two-mesh system of a stretched mesh of 10 points fore and aft of the airfoil and a

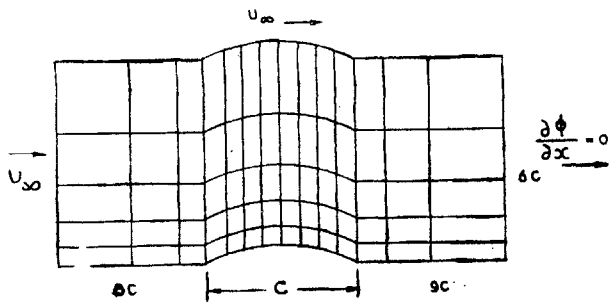

FIG. 2. Mesh about typical circular arc airfoil.

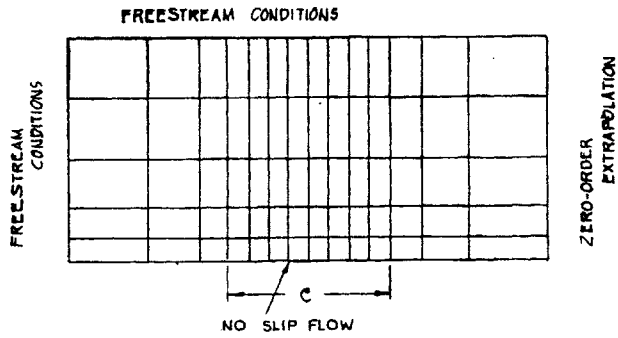

FIG. 3. Mesh system and boundary conditions in transformed computational plane.

uniform mesh of 20 points over the surface of the airfoil was applied, while in the η-direction a stretched mesh was used.

As the airfoil is thick and the flow is transonic, boundary layer separation is likely. To simulate the phenomenon reliably for turbulent flow, it is necessary to resolve the boundary layer to the sublayer scale. This sublayer scale can be taken to be nearly proportional to $1/\sqrt{Re}$ MacCarmack and Baldwin[13] so that for high Reynolds number flows of interest, the mesh resolution near the airfoil surface will be extremely fine. In the present case, we have chosen $\Delta y_{\min} = \frac{1}{3} \cdot \frac{c}{\sqrt{Re}}$, where c the chord length.

7. TURBULENCE MODEL

The well-known Cebeci-Smith's zero-equation equilibrium model of turbulence[14] was applied. Thus the eddy viscosity ϵ is evaluated using

$$\epsilon = \min(\epsilon_i, \epsilon_0) \qquad \ldots(14)$$

where ϵ_i is the eddy viscosity in the inner region given by

$$\epsilon_i = 0.16y^2 [1 - \exp(-y\sqrt{\tau_w \rho}/26\mu)] \left|\frac{\partial u}{\partial y}\right| \rho \qquad \ldots(15)$$

while ϵ_0, the outer eddy viscosity, is got from

$$\epsilon_0 = 0.168\, u_{\max}\, \rho \int_0^{y_\delta} (1 - u/u_{\max})\, dy \qquad \ldots(16)$$

where

$$\tau_w = \mu_w \left(\frac{\partial u}{\partial y}\right)_{y=0} \qquad \ldots(17)$$

and u_{max} is the maximum velocity in the profile extending from $y = 0$ to $y = y_\delta$.

8. Computational Details and Results

The data processing rate was about 7.6 sec per iteration, i.e. 6.3×10^{-3} sec per grid point including prediction and correction. This is, on an average, about 60% more than that for the corresponding explicit scheme. Thus when a Courant number of 5 was used, the implicit numerical scheme took about one-third of the corresponding time for the explicit method. Courant numbers upto 10 could be used without any difficulty. A maximum number of 1180 iterations were required to obtain convergence when the difference in the values of the flow parameters became less than about 0.1% (Shang and Hankey[15]) and this took about 2.5 hours on the computer. This compares very favourably with the computation time to reach steady state reported by Deiwert for similar problems on CDC 7600 computer[1]. It is found that, in the process, the soulutions have been carried out for a time corresponding to the mean flow travelling approximately 2.6 chord lengths. It is also observed that for the implicit method, the memory requirement is marginally more owing to the fact that the Δu's of all elements of the state vector need to be stored. Thus, for the 40×30 mesh system used, while the explicit method requires about 3×10^4 60-bit words of core memory, an additional 6×10^3 words will have to be available for the implicit case. In other words, the increase in memory requirement for the implicit scheme is of the order of 20 per cent.

Figure 4 presents the computed pressure co-efficient variation over the surface of an 18% thick symmetric circular arc airfoil corresponding to Mach number 0.775 flow with a Reynolds number of 10^6. As can be noted, the results are in good agreement those of Deiwert[1] through an explicit integral scheme. The inviscid pressure distribution, obtained through an Euler code of the present authors[12] for the same problem, is also shown for comparison. The viscous solution lies to the left of the inviscid

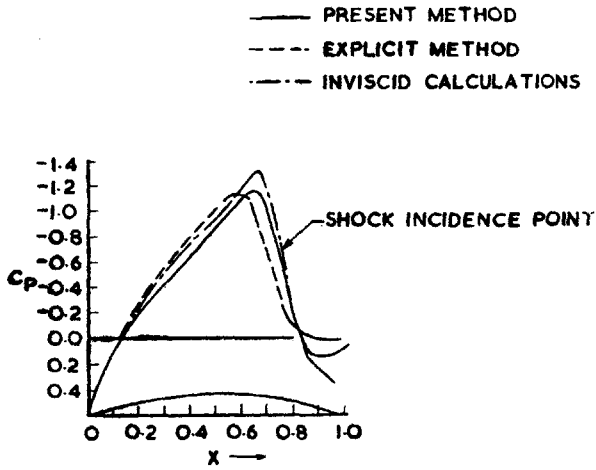

FIG. 4. Pressure coefficient over 18 percent circular arc air foil ($M_\infty = 0.775$, $R_e = 10^6$).

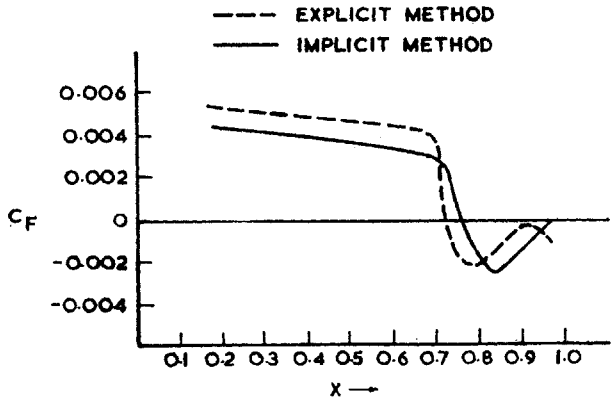

FIG. 5. Skin friction coefficient.

solution because of the boundary layer displacement effects. It is observed from the pressure distribution that the shock is centered between 73 and 78% of the chord.

Figure 5 describes the variation of the skin friction coefficient over the airfoil surface which is again in reasonable conformity with Deiwert's computed values[1], exhibited in the same figure.

9. CONCLUSION

An implicit numerical method, an adaptation of the well-known MacCormack scheme, for solving two-dimensional unsteady Navier-Stokes equations transformed into a rectangular computational domain is presented in this paper. The scheme is applied to simulate high Reynolds number transonic flow past 18% symmetric circular arc airfoil and a computer program developed for the purpose. The numerical results for surface pressure distribution and skin friction coefficient, obtained through the software, exhibt good agreement with the earlier computed values of Deiwert. The new method entails a substantial reduction in the computation time while retaining second-order accuracy.

ACKNOWLEDGEMENT

The authors would like to express their deep appreciation to the referees for their suggestions to improve this paper.

REFERENCES

1. G. S. Deiwert, *AIAA JI* 13 (1975), 1355-59.
2. G. R, Inger, *AIAA*, 82 (1982), 621-36.
3. R. W. MacCormack, *AIAA JI.* 20 (1982), 1275-81.
4. J. S. Shang, and R. W. MacCormack, *AIAA Paper* 83-1668, 1983.
5. R. N. Gupta, P. A. Gnoffo, and R. W. MacCormack, *AIAA JI* 23 (1985), 723-32.
6. V. Swaminathan, and N. S. Madhavan, *Proc. 3rd Int. Conf. Num. Methods for Laminar and Turbulent Flow*, Pineridge Press, Paper No. 21, 1983, 221-31.
7. N. S. Madhavan, and V. Swaminathan, *Int. JI. Num. Methods Fluids* 6 (1986), 387-93.

8. S. L. Lawrence, J. C. Tannehill, and D. S. Chaussee, *AIAA JI* **22** (1984), 1755–63.
9. R. W. MacCormack, *AIAA Paper* 69–354, 1984.
10. R. W. MacCormack, In : *Lecture Notes in Physics, Num. Meth. Fluid Dynamics*, Springer Verlag **8** (1971), 151–63.
11. V. Swaminathan, and N. S. Madhavan, In : *Aspects of Mechanics, Lecture Notes in Applied Mathematics*, (ed. : D. K. Sinha) South Asian Pub. Co. Delhi; 1985, pp. 53–59.
12. N. S. Madhavan, and V. Swaminathan, *Indian J. Pure appl. Math.* **17** (1986), 1164–73.
13. R. W. MacCormack, and B. S. Baldwin, *AIAA Paper* 75–1, 1975.
14. T. Cebeci, A. M. O. Smith, and G. Mosinkis, *AIAA JI* **8** (1970), 1974–82.
15. J. S. Shang, and W. L. Hankey, *AIAA Paper* 75–3, 1975.