AN UPWIND FORMULATION FOR THE SOLUTION OF THIN-LAYER NAVIER-STOKES EQUATIONS

S.K. Saxena*

Abstract

Thin-Layer Navier-Stokes equations in generalized body-fittedcoordinate system and non-dimensional form are solved to numerically simulate high speed viscous flows. A second order accurate explicit upwind MUSCL type algorithm incorporating flm-vector splitting due to Van Leer is formulated within the frame work to finite volume approach. TVD limiters are used to avoid undershoots and overshoots in the shock region. The results of laminar flow calculations are presented for hem-sphere cylinder configuration at supersonic Mach number.

Introduction

Thin-layer Navier-Stokes equations (TLNS) are being widely used now-a-days over transonic to hypersonic Mach number range in numerical simulation of viscous flows of practial interest. The present work considers the compulation of 2D/axisymmetric high speed viscous flows past blunt bodies based on the solution of the TLNS. A time accurate upwind MUSCL type algorithm incorporating flux-vector splitting due to Van Leer is formulated within the frame work of finite volume approach to discretize the Euler terms, while the viscous terms are central differenced. The Van Leer flux-splitting, although incorporated by other authors [1,2] in viscous codes because of its ability to provide crisp shocks and smooth distribution of the fluxes over the whole Mach number range, is known to cause large numerical damping at the contact discontinuity. The modifications to this splitting originally proposed by Hänel [3] have been considered in the present work to reduce the numerical damping and improve the quality of results in the viscous region. The time accuracy is obtained by employing five-step Runge-Kutta scheme for time discretization.

The present work is based on explicit formulation because of its ease in parallelization at a later stage. The equations are cast in generalized body fitted coordinate system to provide flexibilty in handling different body shapes. Another important feature of the present work is me use of a TVD timiter to avoid undershoots and overshoots in the shock region. The numerical scheme is second order accurate except near the shock where *its* accuracy drops to firs* order.

The results of laminar flow calculations are presented for hemisphere-cylinder configuration at supersonic Mach number and compared with computation by other authors employing a different numerical scheme. The present effort emphasizes the ability of the flow solver to handle strong shocks, large flow gradients and viscous-inviscid interactions which are characteristic features of high speed flows.

Mathematical Formulation

The thin-layer Navier-Stokes equations for 2D/ axisymmetrie flows in conservation, ram-diaieiisloial form and generalized body fitted coordinate system CM be written as,

* Computational and Theoretical Fluid Dynamics Division, National Aeronautical Laboratory, Bangalore - 560 017

$$\frac{\partial(y^{j_o}Q)}{\partial t}+\frac{\partial(y^{j_o}F)}{\partial\xi}+\frac{\partial(y^{j_o}G)}{\partial\eta}-\frac{1}{Re}\frac{\partial(y^{j_o}S)}{\partial\eta}+\frac{j_o}{J}\left(H-\frac{1}{Re}S1\right)=0 \quad (1)$$

with

$Q = J^{-1}\ [\rho\ \rho u,\ \rho v,\ e]^T;$

$F = J^{-1}\ [\rho U,\ \rho uU + \xi_x p,\ \rho vU + \xi_y p,\ (e+p)U]^T$

$G = J^{-1}\ [\rho V,\ \rho uV + \eta_x p,\ \rho vV + \eta_y p,\ (e+p)V]^T,$

$H = [0,0\ -p,\ 0]^T$

$$S=\frac{\mu}{J}\begin{bmatrix}(\eta_x^2+\eta_y^2)\frac{\partial u}{\partial\eta}+\frac{1}{3}\eta_x(\eta_x\frac{\partial u}{\partial\eta}+\eta_y\frac{\partial v}{\partial\eta})-\frac{2}{3}j_o\eta_x\frac{v}{y}\\(\eta_x^2+\eta_y^2)\frac{\partial v}{\partial\eta}+\frac{1}{3}\eta_y(\eta_x\frac{\partial u}{\partial\eta}+\eta_y\frac{\partial v}{\partial\eta})-\frac{2}{3}j_o\eta_y\frac{v}{y}\\(\eta_x^2+\eta_y^2)[\frac{1}{2}\frac{\partial(u^2+v^2)}{\partial\eta}+k\frac{\partial T}{\partial\eta}]+\\1/6[\eta_x^2\frac{\partial(u^2)}{\partial\eta}+\eta_y^2\frac{\partial(v^2)}{\partial\eta}+2\eta_x\eta_y\frac{\partial(uv)}{\partial\eta}]-\frac{2}{3}j_o\frac{v}{y}(\eta_x u+\eta_y v)\end{bmatrix}$$

$$S1=\mu\left[0,0,\frac{2}{3}\eta_x\frac{\partial u}{\partial\eta}+\frac{2}{3}\eta_y\frac{\partial v}{\partial\eta}-\frac{4}{3}\frac{v}{y},0\right]^T$$

where u, v, p, ρ, T and e are velocity components, pressure, density, temperature and total energy per unit volume respectively, and k the coefficient of thermal conductivity and μ the coefficient of viscosity, U and V are contravarient vectors and Re is the Reynolds number based on characteristic length. j_o=0 for plane flow and j_o=1 for axisymmetric flow.

The discretization of metric derivaties is done in a manner consistent with the discretization of the governing equations. Following Hindman [4], a simple test of reproducing uniform flow is employed, starting with the entire computational mesh initialized with a uniform flow and advancing in time.

The equation (1) is discretized using finite volume approach with an explicit upwind scheme as described below.

$$\frac{\partial Q}{\partial t}=-1/y^{j_o}\left[(y^{j_o}F)_{i+\frac{1}{2},j}-(y^{j_o}F)_{i-\frac{1}{2},j}+(y^{j_o}G)_{i,j+\frac{1}{2}}-(y^{j_o}G)_{i,j-\frac{1}{2}}-\frac{1}{Re}\{(y^{j_o}S)_{i,j+\frac{1}{2}}-(y^{j_o}S)_{i,j-\frac{1}{2}}\}+\frac{j_o}{J}(H-\frac{S1}{Re})_{i,j}\right] \quad (2)$$

The inviscid fluxes at the interface are determined using the MUSCL approach. For example,

$$F_{i+\frac{1}{2},j}=F^+(Q^-_{i+\frac{1}{2},j})+F^-(Q^+_{i+\frac{1}{2},j})$$

where, using the limiter [5],

$$(Q^-_{i+\frac{1}{2},j})=Q_{i,j}-\{S/4[(1-KS)\Delta_-+(1+KS)\Delta_+]\}_{i,j}$$

$$(Q^+_{i+\frac{1}{2},j})=Q_{i+1,j}+\{S/4[(1-KS)\Delta_++(1+KS)\Delta_-]\}_{i+1,j}$$

with,

$$S=\frac{2\Delta_+\Delta_-+\epsilon}{(\Delta_+)^2+(\Delta_-)^2+\epsilon}$$

where e is a small number ($\epsilon=10^{-6}$) preventing division by zero in regions of null gradients and

$$(\Delta_+)_{i,j}=Q_{i+1,j}-Q_{i,j}$$

$$(\Delta_-)_{i,j}=Q_{i,j}-Q_{i-1,j}$$

K = -1 for fully upwind scheme, which has been used in the present work.

A generalized 2D Van Leer split flux-vector is given by [6]

$$F^{\pm},G^{\pm}=Ke_1^{\pm}\begin{bmatrix}1\\\frac{k_1}{K\beta}(\pm2c-\bar{u})+u\\\frac{k_2}{K\beta}(\pm2c-\bar{u})+v\\H\end{bmatrix} \quad (3)$$

where $K=\sqrt{k_1^2+k_2^2}$

for $F^{\pm}$

$$k_1=\xi_x;\ k_2=\xi_y,\ e_1^{\pm}=\pm\frac{\rho}{4c}(\bar{u}\pm c)^2,\ \bar{u}=U/K,\ M_\xi=\bar{u}/c$$

for $G^{\pm}$

$$k_1=\eta_x;\ k_2=\eta_y,\ e_1^{\pm}=\pm\frac{\rho}{4c}(\bar{u}\pm c)^2,\ \bar{u}=V/K,\ M_\eta=\bar{u}/c$$

The density in the flux vector is scaled by the Jacobian and c is defined as $c=\sqrt{\beta p/p}$, where $p=\gamma$ for ideal gas. H is the total enthalpy.

For subsonic flow, $|M_\xi|$ or $|M_\eta| < 1$, the split fluxes are given by equation (3) described above. When the flow is supersonic the split fluxes are simply given, for example as

$$F = F,\ F^- = 0 \quad \text{for } M_\xi > 1$$

$$F^+ = 0,\ F^- = F \quad \text{for } M_\xi \le -1$$

Modification to Van Leer flux vector splitting: A number of modifications to Van Leer splitting were suggested by Hänel [3] to improve its performance in hypersonic viscous flow computations. In the original scheme, the split fluxes are expressed in terms of density p, the velocities u and v and static speed of sound $c=\sqrt{\gamma p/\rho}$. All these quantities change strongly in high Mach number flows and, therefore they can cause large numerical errors and reduce convergence. Replacing these variables by less or by even invariant quantities can improve the accuracy of computations. Based on these ideas, the energy flux is modified to contain total enthalpy, static speed of sopund is replaced by critical speed of sound, $c^*=\sqrt{2(\gamma-1)/(\gamma+1).H}$ and velocity u and v at the cell interface are obtained by using upwinding according to the direction of normal velocity.

Time Discretization

Here a 5-step Runge-Kutta scheme with a maximum Courant number of 3.5 is employed as described below.

$$\left.\begin{aligned} Q^{(o)} &= Q^n \\ &\vdots \\ \Delta Q^{(I)} &= -\alpha_I \Delta t\, Res\,(Q^{I-1})/J \\ Q^I &- Q^{(o)} + \Delta Q^{(I)} \\ &\vdots \\ Q^{n+1} &= Q^N \end{aligned}\right| \quad I = 1, N, \quad N = 5$$

The coefficients in the Runge-Kutta steps are chosen as α_I = (0.059, 0,14, 0,273, 0.5, 1).

Boundary Condiations

At the axis, symmetry boundary condition is used, At this inflow boundary, the flow Is at free stream conditions.

The outflow boundary condition employs extrapolation from inside. At the body surface, u = O. v = O, $3p/\partial\eta$ = O, $\partial T/\partial\eta$ = O for wall.

Results and Discussion

An efficient computer code has been developed based on the formulation described above with restart facility. An algebraic grid generation procedure has been used to generate fee grid and the same is shown in Fig. 1. A good approximation to initial flow field is obtained by employing Newton impact theory on the body, obilique shock relations at the shock approximately computed and located through an empirical relation and interpolating tetween them. A separate program generates the grid and the initial flow field required and this goes as an input to the main program.

The code is validated by computing the flow past a hemisphere-cylinder configuration at a Mach number of 2.94 and Reynolds number of 2.2 x 10^5 per meter with adiabatic wall condition. Figs. 2 and 3 show temperature and pressure distribution respectively over the body and it cart be seen that they compare well with the results of computation from Ref. [7]. The difference between the two computations is *less* than 5%. Fig. 4 represents the constant pressure contour indicating that the shock is captured crisply. Fig, 5 shows the pressure distribution along the stagnation line which again indicates a sharp description of fte blunt body shock. Fig. 6 shows the temperature profile in the boundary layer near the hemisphere-cylinder junction. The convergence history is shown in Fig. 7.

Concluding Remarks

A second order accurate explicit upwind MUSCL type formulation has been presented in generalized body fitted coordinate system within the finite volume framework to solve Thin-Layer Navier-Stokes equations. The Van Leer flux-splitting used in fee flow solver to handle inviscid terms through an upwinding procedure

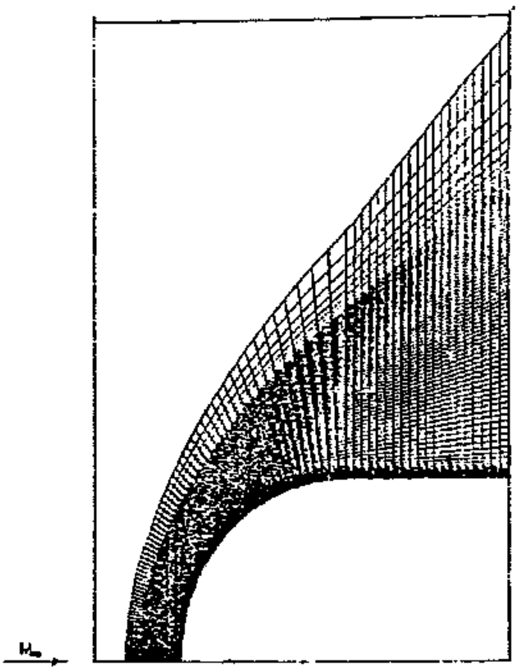

Fig. 1. Hemisphere-Cylinder Grid (120*120)
(M_∞=2.94, Re=2.2*10^5/m)

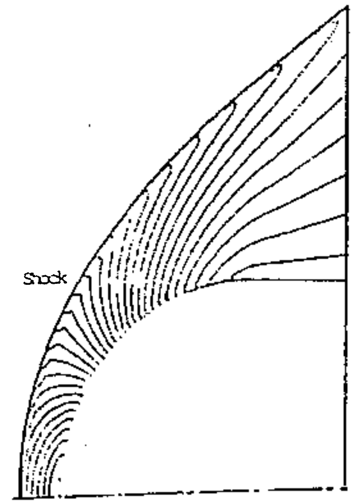

Fig. 4. Constant Pressure Contours

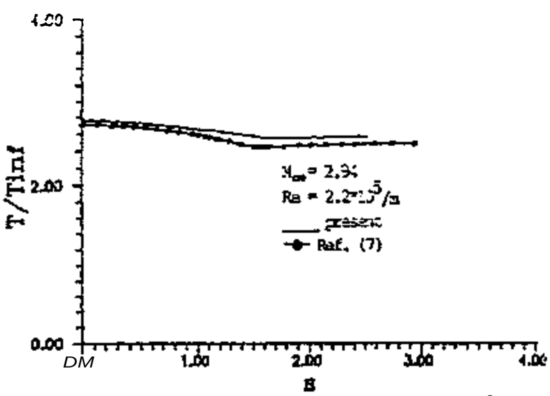

Fig. 2. Temperature Distribution Over The Body

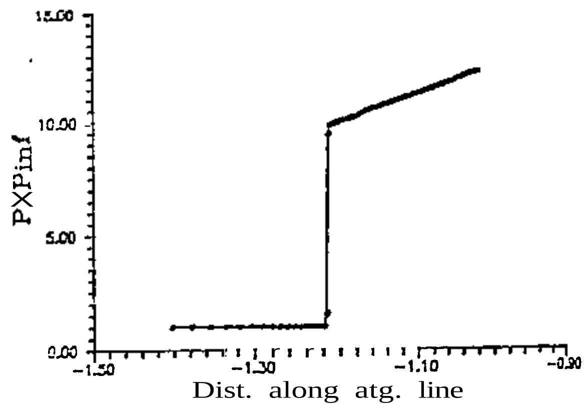

Fig. 5, Pressure Distribution Along the Stagnation Line

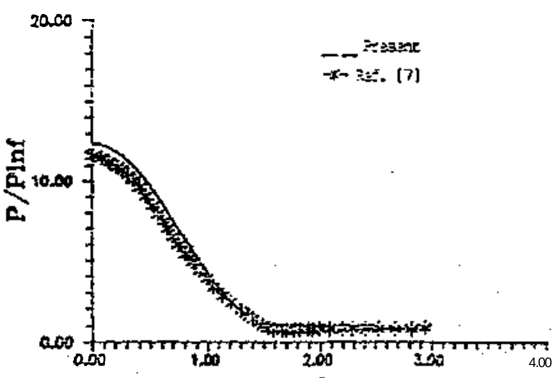

Fig. 3. Pressure Distribution Over The Body

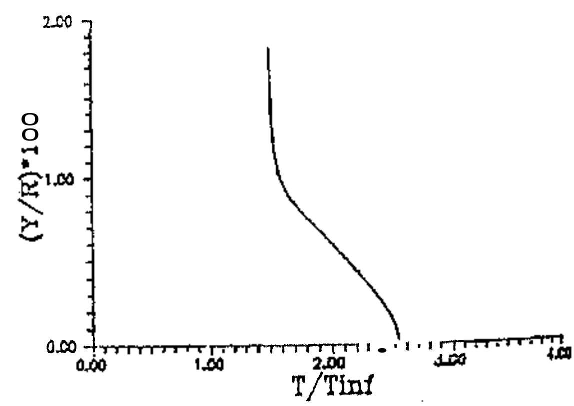

Fig. 6. Temperature Distribution in the Boundary Layer

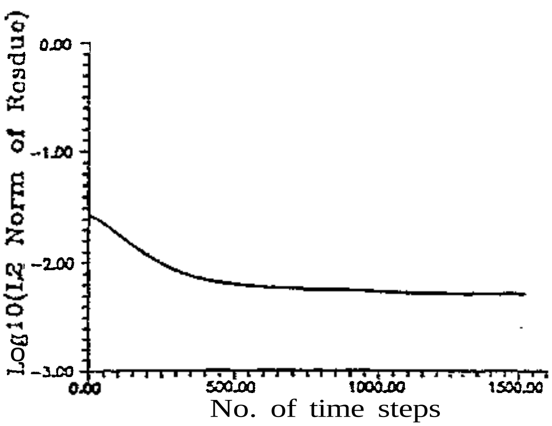

Fig. 7. Convergence History

seem to provide reasonably good simulation of high speed viscous flows. The shock is captured crisply with no artificial dissipation added.

References

1. Yee, H.C., *et al:* "High-resolution shock capturing scheme for inviscid and viscous hypersonic flows", NASA TM 100097 (1988).

5. Anderson, K.W., Thomas, J.L. and Van Leer, B,, "A comparison of finite volume flux-vector splittings for the Euler equations", AIAAJ, 24(9), 1453 (1986).

6. Thomas, J.L., Van Leer, B. and Waiter, R.W., "Implicit flux splitting schemes for the Euler equations", AIAA-85-1680.

7. Viviand, H. and Ghazzi, W., "Numerical solution of the compressible Navier-Stokes equations at high Reynolds number with applications to the blunt body problem", lecture notes in physics, vol. 59 (1976).

2. Lafon, A. and Cousteix, J., "Computations of the Thin-Layer Navier-Stokes equations for a 20 flow", Rech. Aerosp. - n° 1989-3.

3. Hänel, D., and Schwane, R., "An implicit flux-vector splitting scheme for the computation of viscous hypersonic flow", AIAA-89-0274.

4. Hindman, R.G., "Generalized coordinate forms of governing fluid equations and associated geometrically induced errors", AIAAJ 20 (10), 1359 (1982).