JOURNAL OF AIRCRAFT
Vol. 33, No. 2, March–April 1996

Evaluation of Jameson–Schmidt–Turkel Dissipation Scheme for Hypersonic Flow Computations

J. P. Singh*
National Aerospace Laboratories, Bangalore 560 017, India

In the present work, the use of the Jameson–Schmidt–Turkel (JST) dissipation scheme is successfully extended to compute hypersonic flows over an airfoil. It is shown that this scheme is not only capable of computing flows at speeds all the way up to Mach 50, but also the rate of convergence and the final results are as good as those obtained using the Swanson–Turkel modification of the JST scheme and better than the Yoon–Kwak flux limited dissipation specially designed to compute high-speed flows. The scheme is seen to be so robust that it does not require a well-prepared starting solution to compute flows with either strong shocks or with high incidence, or both.

Introduction

THE last decade has seen very significant progress in the development of solvers for Reynolds-averaged Navier–Stokes equations. Several algorithms and methods have been devised to handle specific flow situations. A good number of these flow solvers use the finite volume discretization and Runge–Kutta time-stepping scheme with the addition of an explicit artificial viscosity to provide 1) a high-order background dissipation to avoid the odd–even decoupling and 2) a second component active in the vicinity of the shock to prevent oscillations in that region. This form of artificial viscosity, which has been in popular use since 1981 with the work of Jameson et al.,[1] has been largely confined to the computation of transonic flows. There have also been some attempts to compute supersonic flows using the same scheme. For example, Radespiel[2] has reported successful multigrid computations up to Mach 2 on a NACA 0012 airfoil and Singh et al.[3] reported similar success up to about Mach 4 on the same airfoil using the Jameson–Schmidt–Turkel (JST) dissipation scheme. Earlier in 1990, Swanson and Turkel[4] presented a total variation diminishing (TVD) variant of the JST scheme from which Turkel et al.[5] obtained good results for hypersonic computations up to Mach number 20 (without real-gas effects). They reported no multigrid convergence for high-speed flows with the JST scheme. The present work evaluates the versatility of the JST dissipation by computing hypersonic flow over a NACA 0012 airfoil for speeds as high as Mach 50 (without real-gas effects) and angles of attack as high as 30 deg. There are no reports to the knowledge of the present author that present computed results for strongly shocked flows at such high Mach numbers and angles of attack using the JST scheme. All of the results presented in this phase of the work are on a single grid level. The results obtained using the JST scheme and its TVD variant due to Swanson and Turkel are almost indistinguishable from each other.

A result of the computation using the flux-limited dissipation model of Yoon and Kwak[6] is also presented. Deese and Agarwal[7] have used this scheme to compute hypersonic flows. The present experience is that the basic JST scheme is as good as its TVD variant of Ref. 4 and even better than the flux-limited Yoon–Kwak scheme, as is clearly seen from the results, particularly the residual history, in the present report and in Ref. 7. The basic reason to look for an alternative to the JST scheme for hypersonic flow appears to be that the oscillations in the vicinity of the shock become uncontrollable as the Mach number rises. Based on the present work, it appears that these oscillations, which are encountered during the evolution phase, are checked simply by restricting the pressure and temperature to freestream values at points where the oscillation leads to negative temperature and/or pressure during the first 100 time steps or so. This points to the fact that it is the violent oscillation during the initial transient that is responsible for the problem. In fact, it was encouraging to note that every computation all the way up to Mach 50 and angles of attack up to 30 deg (present computations were tried only up to these limits) was started from the scratch initial solution, i.e., no use was made of any previously converged solution at lower Mach numbers as inputs to the next computation to reach the final high Mach number and/or the angle of attack.

Discretized Navier–Stokes Equations and Artificial Dissipation

The time-dependent Navier–Stokes equations are discretized using the method of lines in which the spatial derivatives are discretized using the cell-centered finite volume scheme. These equations can be written as

$$\frac{d\bar{W}_{i,k}}{dt} + (\bar{Q}_c + \bar{Q}_v - \bar{D})_{i,k} = 0 \tag{1}$$

where $\bar{W} = (\rho, \rho u, \rho w, \rho E)^T$, u and w are Cartesian velocities, E is energy, and Q_c and Q_v are the convective and viscous fluxes, respectively. This discretization is based on central differencing and requires the explicit addition of numerical damping as mentioned previously. This is indicated by D in Eq. (1). In the present work, the JST scheme for the numerical dissipation[1] is used, which contains a blend of second and fourth differences. The second difference term damps oscillations in the vicinity of the shock sensed by a pressure-based sensor. The fourth difference term helps to avoid the odd–even decoupling and provides the background dissipation in the smooth regions of the flow.

The JST numerical dissipation scheme of Ref. 9 is given as (for the case of density)

$$\rho_{i+1/2,k} - \rho_{i-1/2,k} + \rho_{i,k+1/2} - \rho_{i,k-1/2} \tag{2}$$

Received April 6, 1995; presented as Paper 95-1768 at the AIAA 13th Applied Aerodynamics Conference, San Diego, CA, June 19–22, 1995; revision received Sept. 5, 1995; accepted for publication Sept. 5, 1995.

*Scientist, Flosolver Unit, Computational and Theoretical Fluid Dynamics Division. Senior Member AIAA.

where

$$\rho_{i+1/2,k} = \lambda_{i+1/2,k}[\varepsilon^{(2)}_{i+1/2,k}\delta_x p_{i,k} - \varepsilon^{(4)}_{i+1/2,k}\delta^3_x p_{i-1,k}] \quad (3)$$

δ_x is the forward-difference operator and λ are the scaled spectral radii.[9] The coefficients $\varepsilon^{(2)}$ and $\varepsilon^{(4)}$ are written as

$$\varepsilon^{(2)}_{i+1/2,k} = \kappa^{(2)} \max(\nu_{i+1}, \nu_i) \quad (4)$$

$$\varepsilon^{(4)}_{i+1/2,k} = \max(0, \kappa^{(4)} - \varepsilon^{(2)}_{i+1/2,k}) \quad (5)$$

In the present computations, $\kappa^{(2)} = 0.5$ and $\kappa^{(4)} = 1/32$ or 1/64. The code uses the same values of $\kappa^{(2)}$ and $\kappa^{(4)}$ for all speeds from near-incompressible Mach numbers to high hypersonic Mach numbers. The pressure sensor ν is given as[4]

$$\nu = |p_{i+1,k} - 2p_{i,k} + p_{i-1,k}|/[(1-\omega)(P_{\text{TVD}})_{i,k} + \omega P_{i,k}] \quad (6)$$

where $\omega = 1$ gives the standard JST scheme and $\omega = 0$ gives its TVD variant due to Swanson and Turkel.[4] Further,

$$(P_{\text{TVD}})_{i,k} = |p_{i+1,k} - p_{i,k}| + |p_{i,k} - p_{i-1,k}| \quad (7)$$

$$P_{i,k} = p_{i+1,k} + 2p_{i,k} + p_{i-1,k} \quad (8)$$

Methodology

The semidiscrete governing Eqs. (1) are solved by time marching using explicit five-stage hybrid Runge–Kutta local time stepping with the viscous fluxes computed only at the first stage and the numerical dissipation computed at alternate stages. The residues are smoothed using Martinelli's[9] variable coefficient scheme. The eddy viscosity is computed using the Baldwin–Lomax model.[8] All of the computations have been made using Courant–Friedrichs–Lewy number = 1.6 for an adiabatic wall condition.

Results and Discussion

A C-grid generator appropriate for generating grids for transonic flows has been adapted to compute grids for the NACA 0012 airfoil to resolve the bow shock in the present computations as best as possible. Figure 1 shows a 321 × 81 C-mesh used in the present work to compute the flow over a NACA 0012 airfoil. To resolve the boundary layer well, the first grid line enveloping the contour is placed about $10^{-5}c$ away. The grid used by Turkel et al. in Ref. 5 is well clustered ahead of the leading edge and is well aligned with the shock offering a good shock resolution. In contrast to this, the mesh used in the present computations does not have such mesh clustering. This fact should be kept in mind while comparing the sharpness of the shock. However, unlike the experience of the authors of Ref. 5, no convergence problem was faced with the use of the present grid to compute high-speed flows. In the following, a few computed results are presented with comparisons where available.

It is emphasized that all of the computations were started from scratch, i.e., a starting solution with the Cartesian components of velocity based on freestream Mach number, density, and temperature taken as unity and pressure computed from gasdynamic equations. No attempt was made deliberately to use partially or fully converged solution at lower Mach numbers or incidence, progressively being increased to higher values, as a starting solution for flows at higher Mach numbers or incidence with a view to demonstrating the robustness of the JST scheme to compute high-speed flows. As can be seen in the results presented, convergence to good results is obtained without any problem whatsoever, even under flow conditions of extreme severity from the point of view of either shock strength or incidence or both.

Figure 2 shows the convergence history for the computation of flow with $M_\infty = 10$ and $\alpha = 0$ deg using JST as well as the Yoon–Kwak schemes. The Reynolds number is 10^7 and transition is forced at 20% chord. It is seen that the JST scheme produces a fairly clean convergence resulting in a residual drop of three orders in about 1000 time steps and continues to fall at a slower rate subsequently, which is typical of single-grid

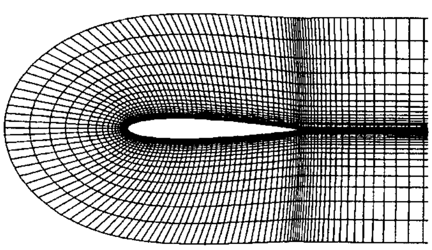

Fig. 1 Present C-mesh (321 × 81).

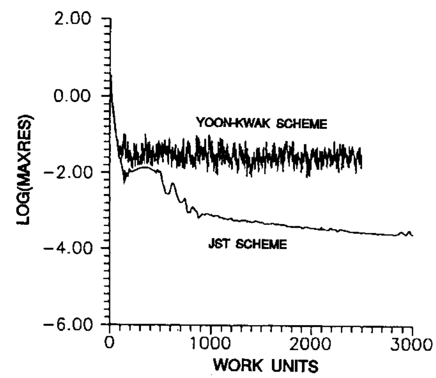

Fig. 2 Comparison of the convergence histories, $M = 10$, $\alpha = 0$ deg.

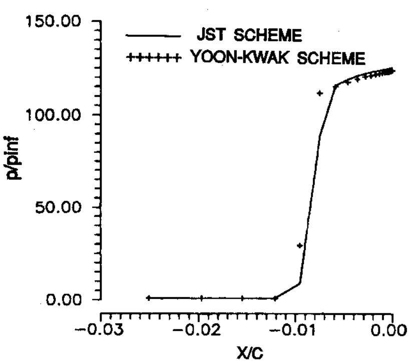

Fig. 3 Comparison of the jump in pressure across the shock $M = 10$, $\alpha = 0$ deg.

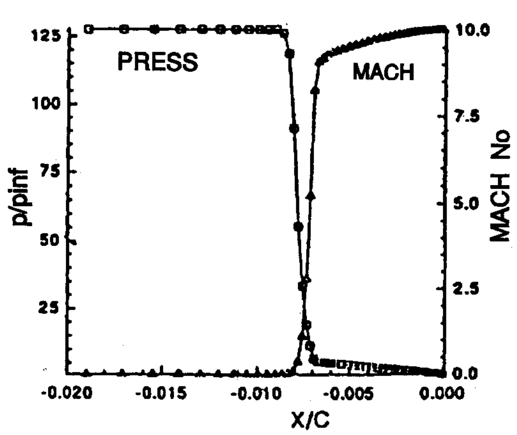

Fig. 4 Pressure and Mach number jump from Ref. 5, $M = 10$, $\alpha = 0$ deg.

Table 1 Comparison of jump in flow quantities across the shock just ahead of the leading edge at $M_\infty = 10$

Scheme	p/p_∞	ρ/ρ_∞	T/T_∞
Theory	116.5	5.714	20.39
JST	117.9	5.76	20.6
Yoon-Kwak	115.08	5.0	22.8

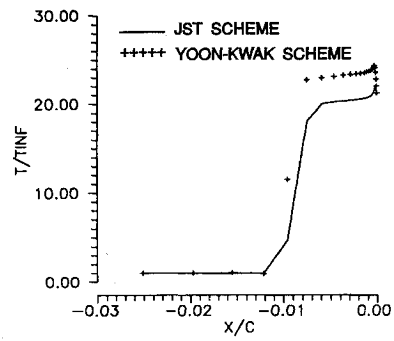

Fig. 5 Comparison of the jump in temperature across the shock, $M = 10$, $\alpha = 0$ deg.

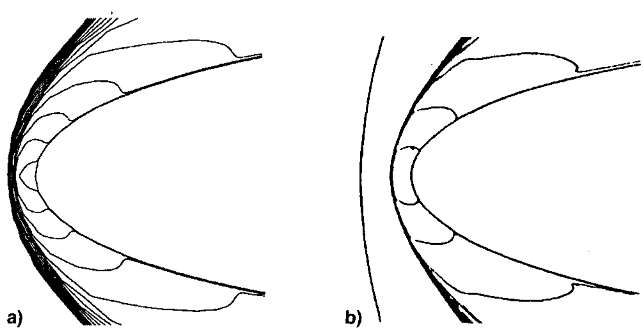

Fig. 6 Mach contours $M = 10$, $\alpha = 0$ deg a) present and b) Ref. 5.

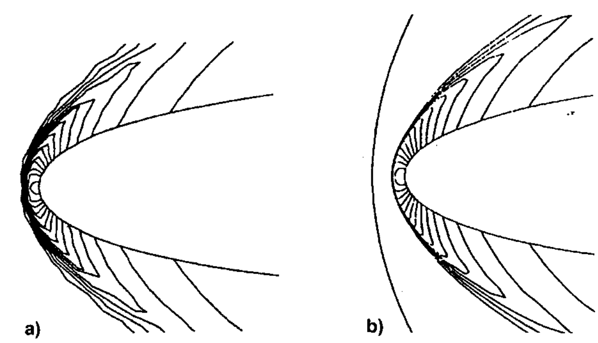

Fig. 7 Pressure contours $M = 10$, $\alpha = 0$ deg a) present and b) Ref. 5.

computations. In contrast, the Yoon–Kwak scheme produces a very kinkish residue history flattening off at relatively higher level. This nature of the Yoon–Kwak scheme is also seen in Ref. 7. Figure 3 shows the jumps in pressure across the shock computed using the JST scheme, its TVD variant, and the Yoon–Kwak schemes. All three schemes predict the pressure jump within about 1.2% accuracy as compared with the theoretical value. The jump in the quantities across the shock is computed just ahead of the leading edge, where the shock is locally normal. The theoretical values are obtained using one-dimensional normal shock jump conditions. For a comparison of jumps in pressure across the shock, the corresponding result of Ref. 5 is reproduced in Fig. 4, in which the sharpness of

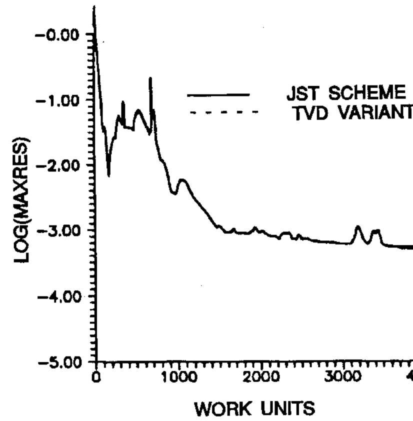

Fig. 8 Comparison of the convergence history, $M = 50$, $\alpha = 0$ deg.

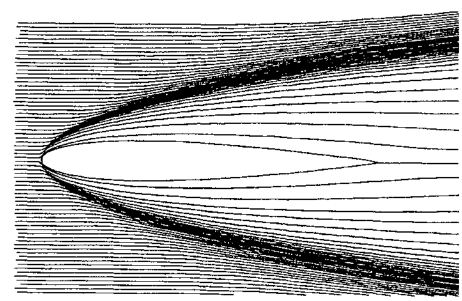

Fig. 9 Streamline pattern, $M = 50$, $\alpha = 0$ deg.

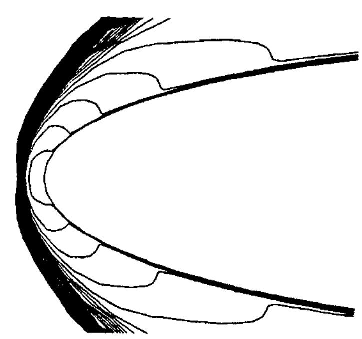

Fig. 10 Mach contours $M = 50$, $\alpha = 0$ deg (present).

the shock is better due to the reasons of the grid quality mentioned earlier. Figure 5 shows the jump in temperature computed using these three schemes. The JST scheme and its TVD variant predict jump in temperature within about 1% of the theoretical value and the Yoon–Kwak scheme overpredicts by nearly 12%. Table 1 gives a comparison of the accuracy in shock jump prediction at $M_\infty = 10$.

Figure 6a shows the Mach number contours at $M_\infty = 10$ obtained using the standard JST scheme as well as its TVD variant. The two results are indistinguishable on the plotting scale. The corresponding computed results of Ref. 5 are reproduced in Fig. 6b for qualitative comparison where the shock thickness ahead of the leading edge is sharper due to the grid topology. However, the shock thickness is comparable away from the leading edge where the two grid topologies are not much different. A similar comparison of pressure contours is presented in Figs. 7a and 7b. The contour levels in the figures of Ref. 5 are not marked, but since Ref. 5 and the present work use the same basic components of the scheme (i.e., cell-centered finite volume formulation, five-stage Runge–Kutta scheme with weighted evaluation of the dissipation computed at the first, third, and fifth stages with variable coefficient implicit residual smoothing at each stage), the results of the two

Table 2 Summary of shock jumps with JST scheme

Variable	$M_\infty = 10$ Theory	$M_\infty = 10$ Computation	$M_\infty = 20$ Theory	$M_\infty = 20$ Computation	$M_\infty = 50$ Theory	$M_\infty = 50$ Computation
ρ/ρ_∞	5.714	5.76	5.926	6.07	5.988	6.08
P/P_∞	116.5	117.9	466.5	473	2918.5	2930
T/T_∞	20.39	20.6	78.7	79.2	487	481

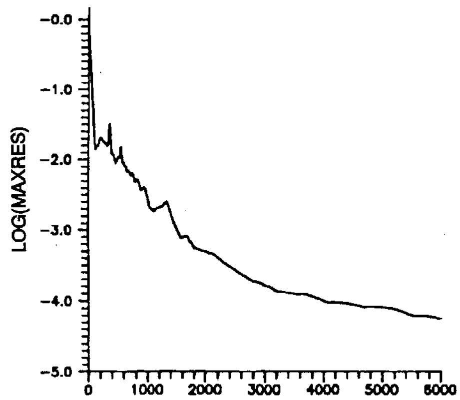

Fig. 11 Convergence history $M = 25$, $\alpha = 30$ deg (present).

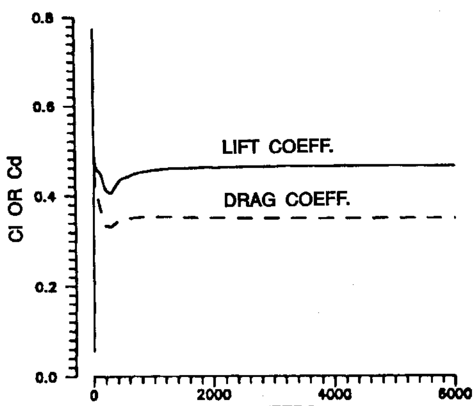

Fig. 12 Evolution of lift and drag $M = 25$, $\alpha = 30$ deg (present).

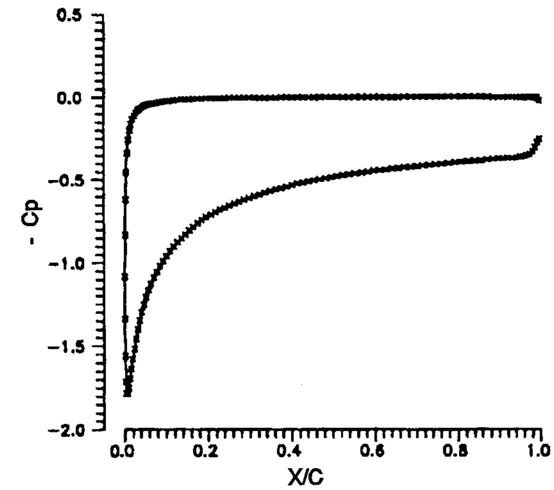

Fig. 13 Pressure distribution $M = 25$, $\alpha = 30$ deg (present).

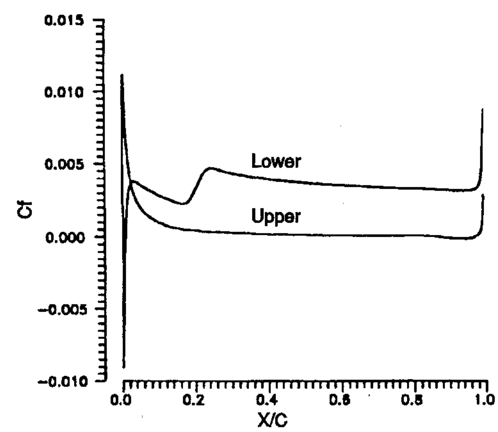

Fig. 14 Skin friction distribution $M = 25$, $\alpha = 30$ deg (present).

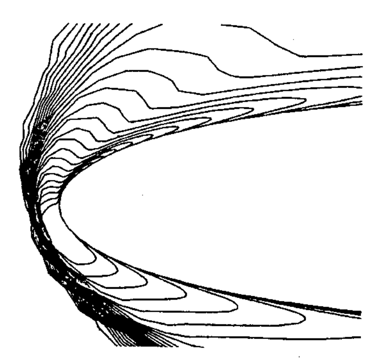

Fig. 15 Mach contours $M = 25$, $\alpha = 30$ deg (present).

computations are expected to be close to each other. The results of a similar exercise at $M_\infty = 20$ were identical in nature.

To test the ultimate efficacy and robustness of the JST scheme, a few more studies with increased flow severity were conducted. In the first case, flow computation was made under a severely shocked flow condition at $M_\infty = 50$. Although the flows without real-gas effects at high hypersonic Mach numbers, especially as high as Mach number 50, are only of theoretical interest, this case is chosen to demonstrate the capability of the basic JST scheme and other modifications in the code to handle the involved severity of the solution process of the basic gasdynamic equations. Figure 8 shows the residual history for this case. A good convergence to more than three orders in about 1500 time steps is seen. This residual drop, once again, is typical of computations using a single grid level. Figures 9 and 10 show the streamline pattern and Mach number contour for Mach 50 flow, which are seen to be smooth and qualitatively good. Table 2 gives a summary of the computed jump, ahead of the leading edge, in various flow quantities across the shock. It can be seen that the present method with the JST scheme is capable of computing the jump in these quantities within an accuracy of 1–2% as compared with the theoretical values.

In the next case, a flow condition with additional severity of high angle of attack is considered that corresponds to $M_\infty = 25$ and $\alpha = 30$ deg. Figure 11 shows the residue history for this case. The residue drops by three orders of magnitude in about 1500 time steps and continues to fall by more than four orders in 6000 time steps. Figure 12 shows the evolution of

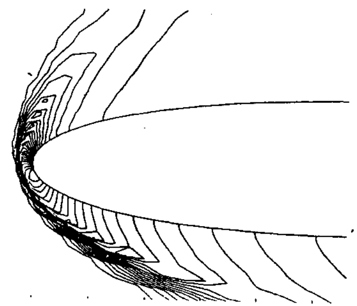

Fig. 16 Pressure contours $M = 25$, $\alpha = 30$ deg (present).

lift and drag. The computation shows that the lift and drag values start with a large oscillation, but quickly settle down to a steady value, even under such difficult conditions. Figures 13 and 14 show the computed pressure and skin friction (normalized with respect to freestream conditions) and Figs. 15 and 16 show the Mach number and pressure contours for this case, which appear to be qualitatively quite satisfactory.

Conclusions

The JST numerical dissipation has been used to compute hypersonic flows including those at high angles of attack using a cell-centered finite volume explicit Runge–Kutta local time stepping scheme.

In contrast with the experience seen in the literature, the present experience indicates that the JST scheme does not seem to pose any convergence problem at high Mach numbers. In fact, the results obtained are in near total agreement with those obtained using its TVD variant due to Swanson and Turkel,[4] designed to enhance the capability of the basic JST scheme to compute high Mach number flows. The scheme is also seen to perform better than the flux-limited scheme of Yoon and Kwak.[6] The JST scheme is also found to be so robust that with a simple fix during the early evolution of the solution it is possible to compute flows with severe conditions of either a very strong shock or a high angle of attack or both without even taking recourse to the successive use of a previously stabilized solution to be used as input to start the next computation with higher Mach number and incidence. The scheme predicts jumps in flow quantities across the shock just ahead of the leading edge, where the shock is locally normal, within about 1–2% accuracy as compared with the theoretical values obtained using one-dimensional normal shock jump conditions.

Acknowledgments

The author thanks K. S. Yajnik, Head, Centre for Mathematical Modelling and Computer Simulation (CMMACS), NAL, Bangalore, India, for permitting him to make extensive use of the Convex 3820 computer and K. S. Ravichandran for several useful discussions during the course of this work.

References

[1]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Simulation of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time Stepping-Schemes," AIAA Paper 81-1259, June 1981.

[2]Radespiel, R., "A Cell-Vertex Multigrid Method for the Navier-Stokes Equations," NASA TM 101557, Jan. 1989.

[3]Singh, J. P., Schwamborn, D., and Kloppmann, C., "Development of a General Purpose Multigrid Accelerated Navier-Stokes Solver," National Aerospace Lab., NAL SP-9315, Bangalore, India, July 1993.

[4]Swanson, R. C., and Turkel, E., "On Central Difference and Upwind Schemes," NASA Langley Research Center, Inst. for Computer Applications in Science and Engineering Rept. 90-44, Hampton, VA, June 1990.

[5]Turkel, E., Swanson, R. C., Vatsa, V. N., and White, J. A., "Multigrid for Hypersonic Viscous Two- and Three-Dimensional Flows," AIAA Paper 91-1572, June 1991.

[6]Yoon, S., and Kwak, D., "Artificial Dissipation Models for Hypersonic External Flows," AIAA Paper 88-3708, July 1988.

[7]Deese, J. E., and Agarwal, R. K., "Three Dimensional Hypersonic Navier-Stokes Calculations Using Central Difference Methods with Adaptive Dissipation," AIAA Paper 90-3069, Aug. 1990.

[8]Baldwin, B. S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[9]Martinelli, L., "Calculation of Viscous Flows with a Multigrid Method," Ph.D. Dissertation, MAE Dept., Princeton Univ., Princeton, NJ, Oct. 1987.