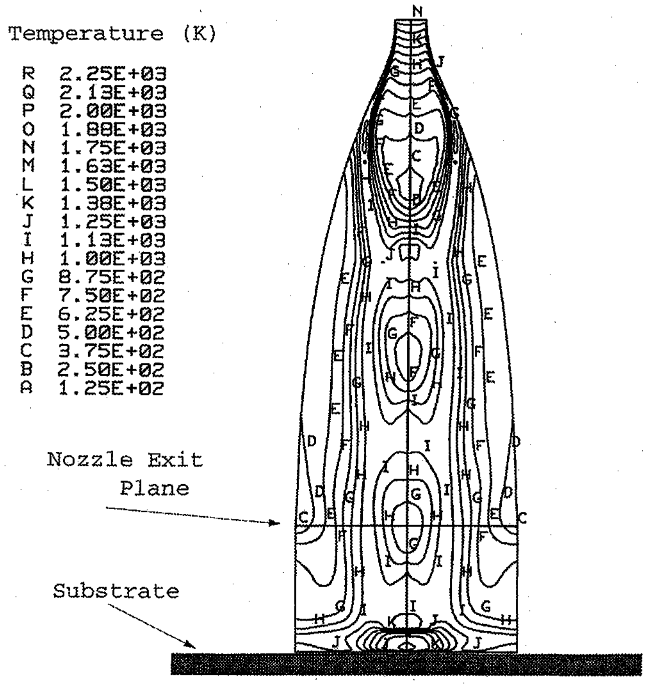


Fig. 1 Temperature contours in nozzle section.

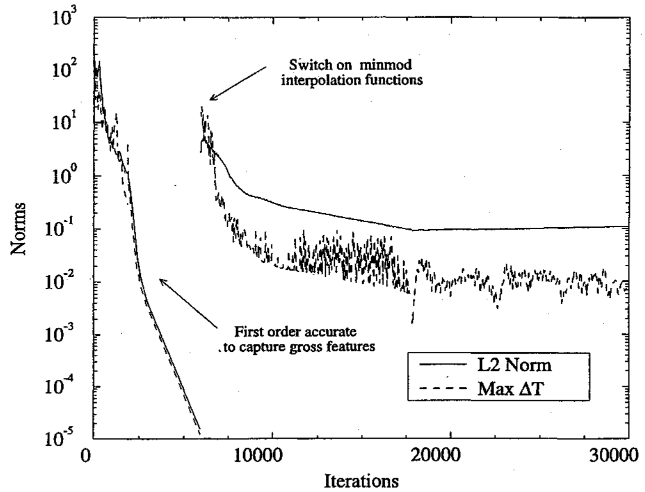


Fig. 2 Convergence characteristics for line relaxation applied to the arc jet flowfield.

Summary

A solution strategy for solving the compressible Navier–Stokes equations applied to supersonic internal flows has been presented. The implicit time-integration scheme consists of line relaxation written in terms of primitive variables. The linearized terms are a simple approximation of the flux splitting used to discretize the spatial conservation laws. The linearized implicit terms are constructed from central differencing with artificial dissipation. A matrix scaling term is introduced for the implementation of the artificial dissipation in the convective terms. The implicit scheme is effective in propagating shocks and handling interactions with the internal viscous flow features found in an arc-heated plasma jet reactor. Future efforts are directed toward investigating time preconditioning to enhance convergence in the large regions of low Mach number flow.

Acknowledgment

Work reported on herein was supported by the U.S. Department of Energy through Sandia National Laboratories under Contract DE-AC04-94AL85000.

References

[1]Moen, C. D., and Dwyer, H. A., "Numerical Simulation of Supersonic Internal Flow for an Arc-Heated Plasma Chemical Vapor Deposition Reactor," AIAA Paper 95-2208, June 1995.

[2]Moen, C. D., and Dwyer, H. A., "Numerical Simulation of Chemical Kinetics in a Supersonic Chemical Vapor Deposition Reactor," AIAA Paper 95-1676, June 1995.

[3]Thomas, J. L., and Walters, R. W., "Upwind Relaxation Algorithms for the Navier–Stokes Equations," AIAA Paper 85-1501, June 1985.

[4]MacCormack, R. W., and Candler, G. V., "The Solution of the Navier–Stokes Equations Using Gauss–Seidel Line Relaxation," *Computers and Fluids*, Vol. 17, No. 1, 1989, pp. 135–150.

[5]Shuen, J.-S., Chen, K.-H., and Choi, Y.-H., "A Coupled Implicit Method for Chemical Non-Equilibrium Flows at All Speeds," *Journal of Computational Physics*, Vol. 106, No. 2, 1993, pp. 305–318.

[6]Shuen, J.-S., and Yoon, S., "Numerical Study of Chemically Reacting Flows Using an LU Scheme," AIAA Paper 88-0436, Jan. 1988.

[7]Liou, M.-S., and Steffens, C. J., Jr., "A New Flux Splitting Scheme," NASA Lewis Research Center, TR TM 104404, 1991.

[8]Hirsch, C., *Numerical Computation of Internal and External Flows*, Vol. 2, Wiley, New York, 1990.

Rotating Dissipation for Accurate Shock Capture

C. de Nicola,*
University of Naples "Federico II," Naples 80125, Italy
G. Iaccarino,†
Centro Italiano Ricerche Aerospaziali, Capua (CE) 80143, Italy
and
R. Tognaccini‡
University of Naples "Federico II," Naples 80125, Italy

I. Introduction

AN open problem in the simulation of supersonic flows is the capture of shocks that are, in general, oblique with respect to grid lines. Especially for upwind-biased algorithms, the use of a collection of one-dimensional operators in two or three dimensions produces poor results because of the wrong choice in the direction of propagation of the characteristic variables. The fault in the domain of dependence is easy to understand by examining a shock oblique to the mesh. With the grid-dependent schemes, the velocity component in one of the grid directions may be subsonic upstream of the shock, and the numerical information can propagate nonphysically in the upstream region, reducing the shock resolution. The spreading of the discontinuity would be reduced if a new mesh, orthogonal to the shock, was employed. This forms the basis of the so-called rotated upwind formulation, which attempts to reproduce an orthogonal shock situation without actually changing the mesh. The development of this grid-independent approach has led to a family of very promising schemes.[1] The concept of rotating the numerical scheme to follow the physical properties of the flow was introduced by Jameson[2] to solve the full potential equation. Roe[1] addressed the problem of solving a multidimensional hyperbolic system, and encouraging applications to the Euler equations have been obtained by Van Leer et al.[3] and Dadone and Grossman.[4]

Received July 31, 1995; revision received March 14, 1996; accepted for publication March 18, 1996.

*Associate Professor, Dipartimento di Progettazione Aeronautica, Piazzale Tecchio 80.

†Research Scientist, via Maiorise.

‡Staff Research Scientist, Dipartimento di Progettazione Aeronautica, Piazzale Tecchio 80.

Recently, Paillere et al.[5] introduced a new grid-independent scheme based on a conservative linearization of the Euler equations.

Our aim is to introduce some concepts related to the upwind multidimensional schemes into a standard central method.[6] We present a new approach to rotated differencing that is simple and efficient in terms of computational efforts. The basic idea is, first, to compute the standard grid-aligned flux (advection + artificial dissipation) and, finally, to use a correction that is rotated with respect to the grid and aligned with the direction normal to the shock wave. In this way, we can use any grid-aligned scheme and obtain a better description of oblique discontinuities.

The application of this new approach to a standard supersonic inviscid test case (compression ramp) is presented here, showing a significant improvement in the accuracy with respect to the grid-aligned method. The computational cost is only slightly increased because of the calculation of the shock inclination and the new rotated term, without affecting the convergence behavior.

II. New Rotated Dissipation Operator

We are interested in the solution of the Euler equations governing the motion of a steady, compressible, nonviscous gas. Given a structured grid, the spatial discretization is obtained by using a finite volume approach. We present a simple introduction of the standard central discretization; a detailed description of the algorithm can be found in Ref. 6. The physical convective flux through a cell interface is approximated by a summation of a numerical convective contribution (based on the average of the flow states at both sides of the interface) and an artificial dissipative term. The latter is made up of two contributions. The first one is a third-order term providing a low background level of dissipation sufficient to prevent odd–even decoupling; furthermore, to capture shock waves without overshoots, an additional first-order operator is added locally by a sensor designed to detect pressure jumps. This dissipative term makes the scheme equivalent to a first-order upwind discretization at a shock.[7]

The artificial dissipative flux across the interface $(i, i+1)$ is

$$\boldsymbol{D}_{i+\frac{1}{2}} = \boldsymbol{D}^{GA}_{i+\frac{1}{2}} = R\left[\epsilon^{(2)} - \epsilon^{(4)}\delta_2\right](\boldsymbol{u}_{i+1} - \boldsymbol{u}_i) \qquad (1)$$

where $\boldsymbol{u}_i$ is the vector of conservative variables in the cell i, δ_2 is the second-order central-difference operator, $\epsilon^{(2)}$ and $\epsilon^{(4)}$ are adaptive dissipative coefficients (switched by the shock sensor), and R is a scaling factor made proportional to the spectral radius of the local Jacobian matrix. The time integration is based on a multistage Runge–Kutta scheme with local time-stepping preconditioning and an elliptic implicit residual smoothing formulation.[6]

Smooth flows are well described by using the third-order dissipative contribution. On the other hand, the accuracy near shock waves only depends on the first-order term [the difference scaled by the coefficient $\epsilon^{(2)}$ in Eq. (1)]. This contribution was recently changed by Swanson and Turkel[7] to increase shock sharpness. Here, we present a different modification addressed to an improvement of oblique shock resolution.

The new dissipative operator has been developed according to the following design criteria: normal and oblique shocks have to be captured with the same accuracy; the new scheme must reduce to the basic one for grid-aligned shocks (consistency); the new approach must be effective both in terms of computational cost and convergence performance; the flux conservation must be ensured.

The first step is the practical detection of the direction of upwinding; we used the local pressure gradient.

Once the new direction has been determined, the successive step in the classic rotated approach is to evaluate the two fluxes in the dominant (upwind) and minor directions ($\boldsymbol{D}_D$ and $\boldsymbol{D}_M$), and then to compute the total dissipative flux through the considered interface:

$$\boldsymbol{D}^{\text{rot}}_{i+\frac{1}{2}} = \boldsymbol{D}_D \cos(\beta) + \boldsymbol{D}_M \sin(\beta) \qquad (2)$$

where β is the angle between the dominant direction and the direction normal to the cell face (see Fig. 1). In the present approach the minor dissipation is neglected since along a shock wave the flow is continuous and the new operator can be expressed as

$$\mathbf{D}_{i+\frac{1}{2}} = R^{\text{rot}}\epsilon^{(2)}(\boldsymbol{u}_{k+1} - \boldsymbol{u}_k)\cos(\beta) - R\epsilon^{(4)}\delta_2(\boldsymbol{u}_{i+1} - \boldsymbol{u}_i) \quad (3)$$

where R^{rot} is the spectral radius of the Jacobian matrix based on the velocity normal to the shock wave.

To avoid interpolation procedures, we follow the approach outlined previously[4] and introduced to compute the rotated fluxes via a Riemann solver. It consists of choosing, for example, $k+1$ coincident with $(i+1, j)$ or $(i+1, j-1)$ with reference to Fig. 1 on the basis of the value of angle β.

To use this formulation in a standard approach, we have designed a correction term that, in conjuction with the basic dissipation, approximates the new rotated operator. Hence, the original scheme can be upgraded by using

$$\boldsymbol{D}_{i+\frac{1}{2}} = \boldsymbol{D}^{GA}_{i+\frac{1}{2}} + \Delta R\epsilon^{(2)}(\boldsymbol{u}_{k+1} - \boldsymbol{u}_k)\cos(\beta) \qquad (4)$$

where ΔR is $(R^{\text{rot}} - R)$.

Note that this modification preserves the original nature of the switch between first- and third-order dissipation, and the rotated differencing is just enabled near the shock waves. The main result of such a procedure is to preserve the original convergence properties of the scheme mainly influenced by the third-order dissipative term.

III. Results

The selected test has been proposed by Van Leer.[1] It consists of a supersonic channel flow ($\text{Mach}_{\text{inlet}} = 2$) dominated by a shock wave reflected twice by the walls. The algebraic grid is made up of 96×32 cells.

We performed the flow calculations by using the standard central scheme and the new rotated central scheme.

In Fig. 2, the iso-Mach contours ($\Delta M = 0.05$) are presented. These results also can be compared with computations performed previously.[1] The analysis of this figure shows that there is a significant improvement in the oblique shock resolution by employing the new rotated dissipation. The reflection of the wave on the walls does

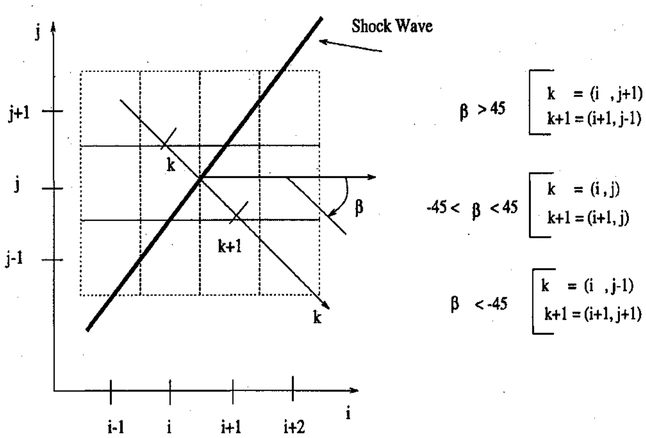
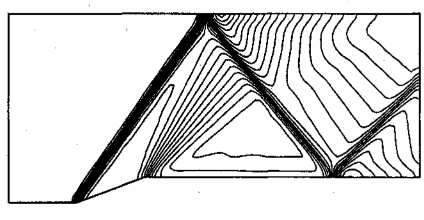
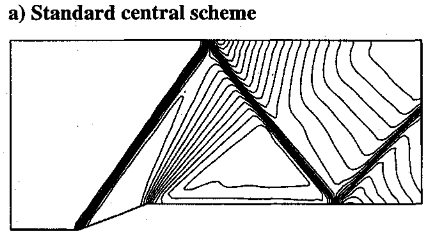


Fig. 1 Shock-aligned frame.



a) Standard central scheme



b) New rotated central scheme

Fig. 2 Supersonic ramp, iso-Mach lines (ΔM = 0.05).

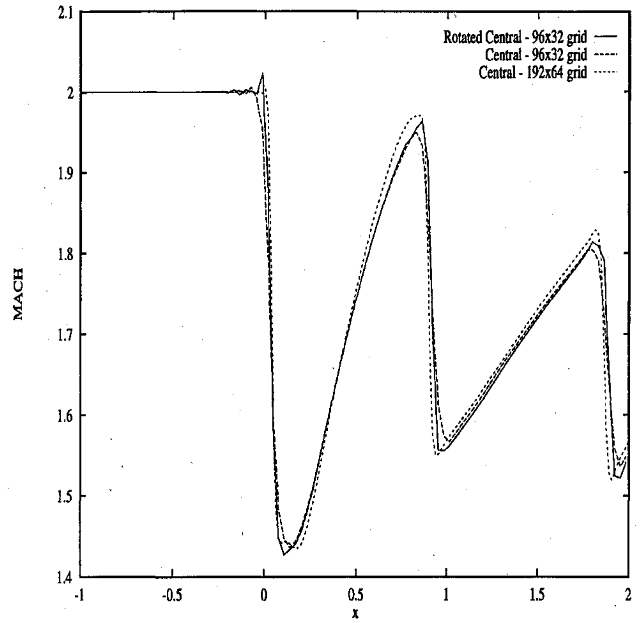


Fig. 3 Supersonic ramp, Mach number distribution at middle height of the channel.

not produce excessive smearing. This interesting property is fully exploited by analyzing Fig. 3 where the Mach-number distributions at middle height of the channel are presented. To assess the accuracy of the new procedure, we compare the results with the calculation performed with the standard scheme on a very fine grid (192 × 64 cells).

The standard central scheme (Fig. 2a) smears the first oblique shock as it approaches the upper wall. There is a large increase of the spreading in the resolution of the second and third reflected shocks. The introduced diffusion destroys the normal shock on the upper surface, and an oblique reflection practically occurs (there is no subsonic zone downstream of the shock). The shock thickness is reduced and shock reflections are better defined by using the new rotated scheme (Fig. 2b).

These improvements are clearly presented in Fig. 3. Sharp shocks are computed by the rotated technique and compared to the highly spread discontinuities of the standard central approach. This result is attributable to the resolution of shocks that are described by two inner cells, i.e., the same accuracy obtainable in case of grid-aligned discontinuities. The shock thickness is now comparable with the solution obtained by using the standard scheme over the refined grid.

As mentioned, the plots of Fig. 2 can be directly compared with previous results.[1] The reported scheme is a first-order rotated upwind that uses two Riemann solutions (in the dominant and in the minor directions) to build up the numerical dissipation. The flow-angle direction used in this case is the velocity magnitude gradient, but because of the numerical noise in regions of uniform flow (convergence halts at some error level), this has been limited.[1] That solution is comparable to that obtained with the present central rotated scheme, which is much simpler and more robust.

Acknowledgment

This work was carried out under cooperation between the Centro Italiano Ricerche Aerospaziali and the University of Naples in the framework of the ENFLOW project (Contract No. 940584).

References

[1]Van Leer, B., "Progress in Multidimensional Upwind Differencing," *Lecture Notes in Physics*, Vol. 414, Springer–Verlag, Berlin, 1992, pp. 1–26.

[2]Jameson, A., "Iterative Solution of Transonic Flow over Airfoils, Including Flows at Mach 1," *Communications on Pure and Applied Mathematics*, Vol. 27, 1974, pp. 283–309.

[3]Van Leer, B., Levy, D. W., and Powell, K. G., "Use of a Rotated Riemann Solver for the Two-Dimensional Euler Equations," *Journal of Computational Physics*, Vol. 106, No. 10, 1993.

[4]Dadone, A., and Grossman, B., "Characteristic-Based, Rotated Upwind Scheme for the Euler Equations," *AIAA Journal*, Vol. 30, No. 9, 1992, pp. 2219–2226.

[5]Paillere, H., Van der Weide, E., and Deconink, H., "Multidimensional Upwind Methods for Inviscid and Viscous Compressible Flows," Lecture Series 1995-02, von Kármán Inst. for Fluid Mechanics, Brussels, Belgium, 1995.

[6]Jameson, A., "Transonic Flow Calculations," Mechanical Engineering Dept., MAE Rept. 1651, Princeton Univ., Princeton, NJ, 1983.

[7]Swanson, R. C., and Turkel, E., "On Central Difference and Upwind Scheme," *Journal of Computational Physics*, Vol. 101, No. 2, 1992.

Modeling Mass Entrainment in a Quasi-One-Dimensional Shock Tube Code

C. J. Doolan* and P. A. Jacobs†
University of Queensland, Brisbane 4072, Australia

Introduction

QUASI-ONE-DIMENSIONAL simulations have been shown to provide quite detailed information about the transient gasdynamics occurring in reflected-mode shock tunnels.[1,2] These models do well in describing the compression process within a free-piston shock tube driver and also in computing the trajectory of the shock wave within a shock tube.[1] However, the speed of the contact surface is usually incorrect because typical boundary-layer models do not redistribute the mass from the core flow to the walls. In quasi-one-dimensional formulations, this can be done by implementing a mass-loss model to the core flow.

Tani et al.[2] apply a mass-loss model to a quasi-one-dimensional formulation by using a simple displacement thickness model. Only qualitative agreement was found as the boundary-layer thickness was underestimated for strong shock waves. More sophisticated approaches such as those of Sharma and Wilson[3] use axisymmetric, two-dimensional simulations of the unsteady Navier–Stokes equations to investigate laminar boundary-layer shock tube flow. Although these methods give satisfactory results, the required computational effort is high, even for the simple shock tube cases they simulate. Conversely, quasi-one-dimensional codes are an attractive tool for engineering design because of their lower computational requirements. Including a mass-loss model for accurate test time and shock speed predictions would provide a useful tool for the designer of shock and expansion tubes where knowledge of the contact surface trajectory is important. Here, it is assumed the test time is the difference between the times of arrival of the contact surface and the shock wave at the measuring station.

Mirels[4] has provided an approximate analytical solution to the shock tube boundary-layer problem. In this case, a steady solution was obtained by assuming that the shock and contact surface had reached their equilibrium positions (i.e., mass flow through the shock front equals mass loss into the boundary layer). This approach gave a solution where the shock speed was independent of the fill pressure. However, for design purposes Mirels' model requires a knowledge of the physical performance of the particular shock tube driver in question. Still, Mirels' results showed that power-law velocity profiles in integral type analyses could be used to describe the boundary layer in a shock tube.

In this Note a quasisteady mass-entrainment model for shock tube boundary layers, specifically for use in quasi-one-dimensional codes, is described. Using this model, we calculate shock tube test times for the turbulent regime.

Received Oct. 24, 1995; revision received Feb. 12, 1996; accepted for publication Feb. 13, 1996.

*Graduate Student, Department of Mechanical Engineering. Member AIAA.

†Lecturer, Department of Mechanical Engineering. Member AIAA.