where subscript e indicates the edge of the shear layer. It is immediately apparent from Eqs. (3-5) that the solution is independent of C_μ so that the Launder et al. correction of C_μ is directly translated to a growth rate modification via $y_{1/2}/x = C_\mu^{1/2}\eta_{1/2}$.

The form of the vortex stretching term of Eq. (5) is derived from Pope's equations for axisymmetric flow.[4] That is,

$$C_{\epsilon 3}\chi = \frac{C_{\epsilon 3}}{4}\left(\frac{k}{\epsilon}\right)^3\left(\frac{\partial q_z}{\partial r} - \frac{\partial q_r}{\partial z}\right)^2\frac{q_r}{r} \tag{8a}$$

where χ is the invariant, r the radial direction, z the direction of the axis of symmetry and q_r and q_z are the respective velocities. Pope requires $C_{\epsilon 3} = 0.79$ to reproduce the round jet growth rate. Substituting, from Eq. (1), and using the thin shear layer approximation,

$$C_{\epsilon 3}\chi = \left(\frac{C_{\epsilon 3}}{4C_\mu}\right)\left(\frac{G}{H}\right)^3 U'^2\left(C_\mu^{-1/2}\frac{V}{\eta}\right) \quad \text{round jet} \tag{8b}$$

$$C_{\epsilon 3}\chi = \left(\frac{C_{\epsilon 3}}{4C_\mu}\right)\left(\frac{G}{H}\right)^3 U'^2 U \quad \text{radial jet} \tag{8c}$$

so, from Eq. (2), a common formula is

$$C_{\epsilon 3}\chi = \bar{C}_{\epsilon 3}(G/H)^3 U'^2 F' \tag{8d}$$

It is evident from Eqs. (8b) and (8c) that the vortex stretching source of dissipation implied for the round jet has a radial jet counterpart. Moreover, the strain rate v/y becomes negative in the outer region of the round jet whereas the strain rate u/x is positive across the entire radial jet. For this reason, it may be expected that the hypothesized increase in dissipation by vortex stretching will be greater in the radial jet than in the round jet.

Results and Conclusions

Using the technique of Paullay et al.,[8] the far-field similarity equations were solved for a variety of free jet cases (see Table 1). The results support the contention that the Pope modification to the k-ϵ model affects the radial jet more than the round jet. The growth rate is reduced by 58% for the former and 28% for the latter. Unfortunately, round jet observations[3] agree with the modified model calculation while radial jet experiments[10,11] support the original model calculation. The plane jet calculations are known to match experiments[2,8] and are shown for comparison. It would appear that the round jet/plane jet anomaly has been exchanged for a round jet/radial jet anomaly.

These results indicate that radial far-field behavior can impose an additional constraint on turbulence models. The similarity form of the thin shear layer equations can aid in determining the consequences of model modification. Such a vortex stretching modification to the k-ϵ model fails when applied to the radial jet.

References

[1] Launder, B. E., Morse, A. P., Rodi, W., and Spalding, D. B., "The Prediction of Free Shear Flows—A Comparison of Six Turbulence Models," NASA SP-322, 1972.

[2] Launder, B. E. and Spalding, D. B., "The Numerical Computation of Turbulent Flows," *Computational Methods in Applied Mechanics and Engineering*, Vol. 3, 1974, pp. 269-289.

[3] Rodi, W., "The Prediction of Free Turbulent Boundary Layers by Use of a Two-Equation Model of Turbulence," Ph.D. Thesis, University of London, England, 1972.

[4] Pope, S. B., "An Explanation of the Turbulent Round-Jet/Plane-Jet Anomaly," *AIAA Journal*, Vol. 16, 1978, pp. 279-281.

[5] Raiszadeh, F. and Dwyer, H. A., "A Study with Sensitivity Analysis of the k-ϵ Turbulence Model Applied to Jet Flows," AIAA Paper 83-0285, Jan. 1983.

[6] McGuirk, J. J. and Rodi, W., "The Calculation of Three Dimensional Free Jets," *Turbulent Shear Flows I*, Springer-Verlag, Berlin, 1979, pp. 71-83.

[7] Bradshaw, P., "Complex Strain Fields," *The 1980-81 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows: Comparison of Computation and Experiment*, Vol. II, Stanford University, Stanford, Calif., 1982, pp. 700-712.

[8] Paullay, A. J., Melnik, R. E., Rubel, A., Rudman, S., and Siclari, M. J., "Similarity Solutions for Plane and Radial Jets Using a k-ϵ Turbulence Model," *Journal of Fluids Engineering, Transactions of the ASME*, Vol. 107, March 1985, pp. 79-85.

[9] Rubel, A. and Melnik, R. E., "Jet, Wake and Wall Jet Similarity Solutions Using a k-ϵ Turbulence Model," AIAA Paper 84-1523, June 1984.

[10] Tanaka, T. and Tanaka, E., "Experimental Study of a Radial Turbulent Jet," *Bulletin of the Japan Society of Mechanical Engineers*, Vol. 19, 1976, pp. 792-799.

[11] Witze, P. O. and Dwyer, H. A., "The Turbulent Radial Jet," *Journal of Fluid Mechanics*, Vol. 75, 1976, pp. 401-417.

Simplified Implicit Block-Bidiagonal Finite Difference Method for Solving the Navier-Stokes Equations

E. von Lavante* and V. S. V. Lyer†
Texas A&M University, College Station, Texas

Introduction

MUCH effort has been expended in recent times in developing a reliable and efficient method for solving the Navier-Stokes equations in two and three dimensions. MacCormack[1] introduced recently an implicit method based on his earlier explicit predictor-corrector scheme[2] that promised a significant increase in computational efficiency while retaining the simplicity of the explicit algorithm. The method was subsequently further developed and studied by several other investigators. Von Lavante and Thompkins[3] extended the method to curvilinear coordinate systems; Casier et al.[4] studied a class of schemes based on the bidiagonal solution technique and discussed several boundary condition procedures. Kordulla and MacCormack[5] applied a modified version of the aforementioned algorithm to transonic inviscid and viscous flow calculations about several airfoils. Although mostly encouraging results were reported, stability problems were encountered in cases with relatively strong shocks. Besides, it was felt that the computational efficiency of this method should still be improved. The present work, motivated by these results, studied the effects of simplifying the original implicit block-bidiagonal algorithm by introducing its spectral normal form. The resulting scheme is twice as fast as the original method and much more robust. On the other hand, it tends to overestimate the viscous effects. It was therefore decided to combine the two methods in a procedure in which approximate results were first obtained by the spectral normal form of the implicit MacCormack scheme (SNIMC) and, then, the full implicit MacCormack scheme (FIMC) was applied as post-processor. The resulting procedure was tested on several test cases with favorable results.

Received May 21, 1984; revision received July 30, 1984.

*Assistant Professor, Aerospace Engineering Department. Member AIAA.

†Graduate Student, Aerospace Engineering Department. Student Member AIAA.

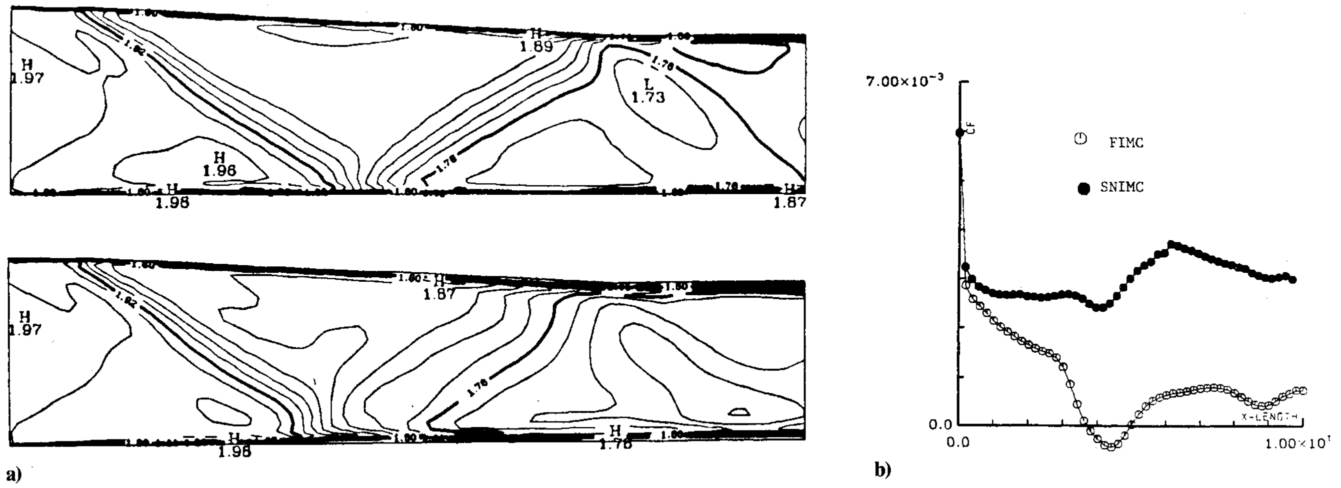

Fig. 1 Solution of flow in a diffuser channel: a) Mach number contours of SNIMC results (top) and of FIMC results (bottom); and b) variation of skin friction coefficient.

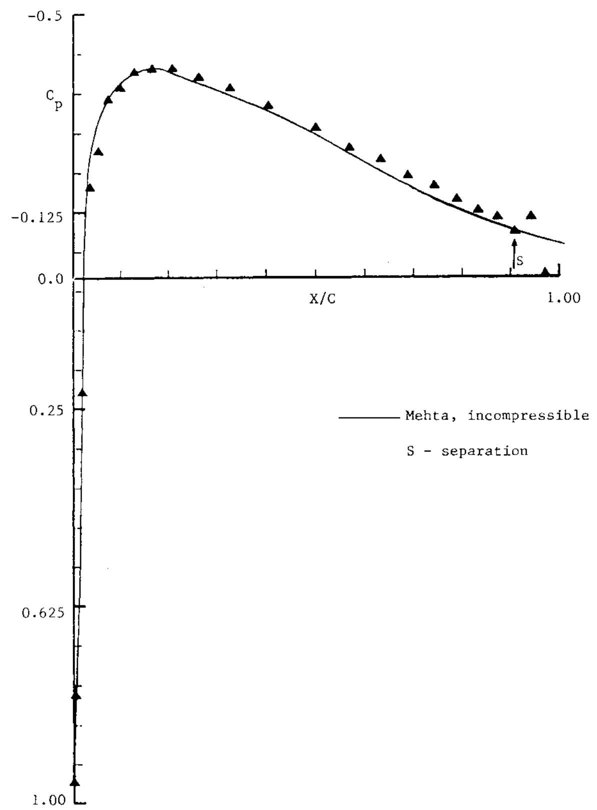

Fig. 2 NACA 0012 results; C_p distribution on airfoil.

Numerical Algorithm

For two-dimensional flows, the compressible Navier-Stokes equations can be written in conservation law form in general, curvilinear coordinates, using vector notation[3,5,6] as

$$\frac{\partial \hat{U}}{\partial t} + \frac{\partial \hat{F}}{\partial \xi} + \frac{\partial \hat{G}}{\partial \eta} = 0 \tag{1}$$

where $\hat{U}$ is the dependent variable vector $J^{-1}(\rho,\rho u,\rho v,e)^T$ and $\hat{F}$ and $\hat{G}$ the usual flux vectors.

The FIMC scheme, as proposed by MacCormack,[1] solves Eq. (1) using an implicit predictor-corrector algorithm similar to its explicit version[2] that results in a block-bidiagonal system of equations. The details of the algorithm development were given in several recent publications[3-5] and will not be repeated here. More recently, Kordulla and MacCormack[5] found out that the accuracy and robustness of the scheme can be improved by increasing the contribution of the explicit parts in proportion to the two-dimensional CFL number. After this modification, the implicit part of the original scheme becomes

Predictor:

$$[I-\Delta t\Delta_+ |\hat{A}|^n]\,[I-\Delta t\Delta_+ |\hat{B}|^n]\,C_2^n\delta\hat{U}^{\overline{n+1}} = C_2^n\Delta\hat{U}^n \tag{2a}$$

$$U^{\overline{n+1}} = \hat{U}^n + C_2^n\delta\hat{U}^{\overline{n+1}} + C_1^n\Delta\hat{U}^n \tag{2b}$$

Corrector:

$$[I+\Delta t\Delta_- |\hat{A}|^{\overline{n+1}}]\,[I+\Delta t\Delta_- |\hat{B}|^{\overline{n+1}}]\,C_2^{\overline{n+1}}\delta\hat{U}^{n+1} = C_2^{\overline{n+1}}\Delta\hat{U}^{\overline{n+1}} \tag{2c}$$

$$\hat{U}^{n+1} = \tfrac{1}{2}[\hat{U}^n + \hat{U}^{\overline{n+1}} + C_2^{\overline{n+1}}\delta U^{n+1} + C_1^{\overline{n+1}}\Delta\hat{U}^{\overline{n+1}}] \tag{2d}$$

where

$$C_1 = \min(1,\ 0.9/\text{CFL});\quad C_2 = (1-C_1);\quad \Delta\xi = \Delta\eta = 1.0$$

Δ_+ and Δ_- are one-sided spatial difference operators and $|\hat{A}|$ and $|\hat{B}|$ the matrices with positive eigenvalues related to the Eulerian flux Jacobians A and B, as explained in Ref. 1. In the present work, this modification was included in the FIMC scheme. The resulting algorithm is stable for high CFL-numbers, is simple and relatively fast because it requires inversions of block-bidiagonal systems of equations and uses implicit part only where needed for stability. It has, however, several disadvantages, such as slow rate of convergency, oscillations at moderate and strong shocks, questionable time accuracy, and strong sensitivity to the correct choice of grid lines distribution. The determination of the eigenvector matrices S_ξ, S_ξ^{-1}, S_η, and S_η^{-1} has to be carried out at each node twice per time step and is, therefore, very costly.

The above algorithm can be significantly simplified and accelerated when the given Jacobian matrices $|\hat{A}|$ and $|\hat{B}|$ are substituted by their spectral radii $\lambda_{A,Sr}\cdot I$ and $\lambda_{B,Sr}\cdot I$. This simplification does not formally change the stability properties of the scheme and results in significant time savings since the matrices S_ξ, S_ξ^{-1}, S_η, S_η^{-1} are not needed. A similar approach has also been suggested by Lerat et al.[6] The spectral radius of, for example, $|\hat{A}|$ can be calculated directly from

$$\lambda_{A,Sr} = \max(|\lambda_i^*|) = |U_\xi| + c\sqrt{\xi_x^2 + \xi_x^2} + \frac{2\nu}{\rho}(\xi_x^2 + \xi_y^2) - \frac{1}{2\Delta t} \tag{3}$$

with a similar expression for $\lambda_{B,Sr}$. The SNIMC scheme is then obtained by replacing the original implicit part, Eqs. (2a) and (2c) by the following:

Predictor:

$$[I - \Delta t \Delta_+ \lambda_{A,Sr}^n \cdot I]\,[I - \Delta t \Delta_+ \lambda_{B,Sr}^n \cdot I]\, C_2^n \delta \hat{U}^{\overline{n+1}} = C_2^n \Delta \hat{U}^n \tag{4a}$$

Corrector:

$$[I + \Delta t \Delta_- \lambda_{A,Sr}^{\overline{n+1}} \cdot I]\,[I + \Delta t \Delta_- \lambda_{B,Sr}^{\overline{n+1}} \cdot I]\, C_2^{\overline{n+1}} \delta \hat{U}^{n+1} = C_2^{\overline{n+1}} \Delta \hat{U}^{\overline{n+1}} \tag{4b}$$

The SNIMC scheme is three times faster than the FIMC scheme, has better rate of convergency, and is more robust. In the inviscid part of the flowfield, where the largest eigenvalue has dominating influence, the flow is correctly modeled. In the boundary layer, however, the smallest eigenvalue is the most important one, making the validity of the preceding simplification questionable. It was observed that in shear layers, the viscous effects were strongly overestimated resulting in incorrect skin friction coefficients.

It was finally decided to combine both SNIMC and FIMC into one numerical procedure in which the SNIMC is first applied in order to obtain an approximate solution. The correct solution is then generated by the FIMC used as postprocessor. The ratio of number of iterations using SNIMC to the number of iterations for FIMC was strongly problem-dependent and is given later.

Test Examples

A. Supersonic Diffusor

The supersonic diffusor from Ref. 3 with inflow Mach number $M_{\text{in}} = 2.0$ was used as a simple test case. The Reynolds number at the shock-lower wall intersection was $Re_x = 3.0 \times 10^5$. The grid had 51×51 points and was numerically generated using the elliptic grid generation method.[7] Explicit boundary conditions were used for simplicity; at the supersonic inflow, all elements of $\hat{U}$ were specified; at the supersonic outflow, $\hat{U}$ was extrapolated; and at the wall, the no-slip boundary conditions $u = 0$, $v = 0$ coupled with the condition for normal derivative of pressure and density, $\rho_n = 0$ and $p_n = 0$ were used. Figure 1 shows the Mach number contour plots after 500 iterations using SNIMC (Fig. 1a) and the final result after another 500 iterations using FIMC at CFL = 125. Although the inviscid core flows are quite similar, the boundary-layer results, especially at the lower wall, are different. This discrepancy is also noticeable in the comparison of the skin friction coefficients in Fig. 1b. The final results agreed very well with computational results obtained by other authors.[3]

B. Subsonic NACA 0012 Airfoil

The laminar flow about a NACA 0012 airfoil at zero angle of attack was calculated for $M_\infty = 0.2$ and $Re = 10^4$. Computations were carried out on a numerically generated half C grid consisting of 45×36 points. The explicit boundary conditions were: upstream, ρ, u, v were specified and p was extrapolated using quasi-one-dimensional Euler equations; downstream, ρ, u, and v were extrapolated using the same procedure and p was obtained from the Rudy-Strickwerda nonreflective boundary condition.[8] The usual solid wall and symmetric boundary conditions were applied at the remaining boundaries. Convergence was assumed when the initial maximum relative residual was reduced by a factor of 10^3. Converged solution was obtained after 700 interations using SNIMC and 800 iterations using FIMC at CFL = 100. The resulting c_p profile, shown in Fig. 2, agrees well with incompressible flow calculation.

C. Transonic Compressor Cascade

The two-step procedure was applied to the solution of transonic flow in an arbitrary compressor cascade. The inflow relative Mach number was 1.38. The axial Mach number was less than unity, which required extrapolation of one variable along the local characteristic from inside the computational domain at inflow. The outflow was subsonic following the nearly normal passage shock; nonreflective boundary condition was applied at the outflow with the static pressure being prespecified.

The SNIMC scheme was first applied for 500 iterations. This was followed by 500 FIMC iterations with a maximum CFL number in the η direction of 70. Turbulent viscosity was included using an algebraic model developed by Baldwin and Lomax.[9] The final results showed the development of a passage shock and separation at the trailing-edge suction side. The SNIMC scheme results showed only relatively mild separation due to overestimation of viscosity effects.

Conclusions

A new method has been proposed to overcome the numerical instability problems associated with the solution of two-dimensional Navier-Stokes equations when solved by the implicit MacCormack algorithm. This involves the use of the spectral normal form in the block-bidiagonal system of equations. Application of this method to several flow cases showed that this modification improves computational efficiency threefold and is much more robust; however, viscous effects are seen to be largely overestimated. Hence, this method requires a postprocessor using the full implicit MacCormack scheme to refine the boundary layer part of the solution. Numerical instabilities are now avoided since a nearly converged solution is used as the initial profile.

References

[1] MacCormack, R. W., "A Numerical Method for Solving the Equations of Compressible Viscous Flow," AIAA Paper 81-0110, Jan. 1981.

[2] MacCormack, R. W., "The Effect of Viscosity in Hypervelocity Impact Cratering," AIAA Paper 69-354, Jan. 1969.

[3] von Lavante, E. and Thompkins, W. T., "An Implicit, Bi-Diagonal Numerical Method for Solving the Navier-Stokes Equations," AIAA Paper 82-0063, Jan. 1982.

[4] Casier, F., Deconinck, H., and Hirsch, C., "A Class of Central Bidiagonal Schemes with Implicit Boundary Conditions for the Solution of Euler's Equations," AIAA Paper 83-0126, Jan. 1983.

[5] Kordulla, W. and MacCormack, R. W., "Transonic Flow Computation Using an Explicit-Implicit Method," *Proceedings of the Eighth International Conference on Numerical Methods in Fluid Dynamics*, Springer-Verlag, New York 1982, pp. 286-295.

[6] Lerat, A., Sides, J. and Daru, V., "An Implicit Finite Volume Method for Solving the Euler Equations," *Proceedings of the Eighth International Conference on Numerical Methods in Fluid Dynamics*, Springer-Verlag, New York, 1982, pp. 343-356.

[7] von Lavante, E., "Elliptic Generation of Two-Dimensional Grids for Internal Flow Calculations," Texas A&M University, College Station, Texas, TEES Rept. 12415-82-01, 1982.

[8] Rudy, D. H. and Strikwerda, J. C., "A Nonreflecting Outflow Boundary Condition for Subsonic Navier-Stokes Calculations," *Journal of Computational Physics*, Vol. 36, 1980, pp. 55-70.

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