N O T I C E

THIS DOCUMENT HAS BEEN REPRODUCED FROM MICROFICHE. ALTHOUGH IT IS RECOGNIZED THAT CERTAIN PORTIONS ARE ILLEGIBLE, IT IS BEING RELEASED IN THE INTEREST OF MAKING AVAILABLE AS MUCH INFORMATION AS POSSIBLE

NI

NASA Technical Memorandum 81279

(NASA-TM-81279) NUMERICAL SOLUTION OF COMPRESSIBLE VISCOUS FLOWS AT HIGH REYNOLDS NUMBERS (NASA) 26 p HC A03/MF A01 CSCL 01A

N81-21025

Unclas

G3/02 41976

Numerical Solution of Compressible Viscous Flows at High Reynolds Numbers

Robert W. MacCormack

March 1981

APR 1981
RECEIVED
NASA STI FACILITY
ACCESS DEPT.

National Aeronautics and
Space Administration

NASA Technical Memorandum 81279

Numerical Solution of Compressible Viscous Flows at High Reynolds Numbers

Robert W. MacCormack, Ames Research Center, Moffett Field California

NASA
National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

Numerical Solution of Compressible Viscous Flows at High Reynolds Numbers

Robert W. MacCormack

Ames Research Center, NASA

Moffett Field, California 94035, U.S.A.

I. Introduction

The equations governing compressible viscous flow have been known for more than a century. Although many special-case solutions have been determined analytically over the years, many others of interest have continued to defy mathematical analysis. Wind tunnels and other experimental facilities have served as invaluable tools in the integration, by physical simulation, of these equations of fluid motion. During the last decade the computer has come to share — through its use of numerical simulations — the work of the earlier analytical and experimental tools in determining new flow solutions.

Like the limits on the range of problems that can be solved analytically, there are limits on the range of flow cases that can be accurately simulated in experimental facilities. The experimental limits are imposed by such factors as tunnel size, wall interference, and stream uniformity [1]. Similarly, the range of computer flow simulations is also limited, principally by computer speed and memory storage. Fortunately, the limits of the theoretical, experimental, and computational techniques are different; as a result, the range of applicability afforded by the three is greater than that attainable with any one of them. Moreover,

in regions where they overlap, one approach can be used to verify the results of another.

Nevertheless, we still cannot solve all fluid flow problems of interest, nor can we anticipate that capability in the near future. However, because of the present rapid and potential large growth of computer capabilities, much emphasis is being placed on the development of computational fluid dynamics [2,3]. Before we can calculate the flow field about a complete aircraft configuration at flight Reynolds numbers, there will have to be great progress in developing powerful and reliable computer hardware, in understanding and modeling the physics of turbulent flow, and in devising accurate and efficient numerical methods. That progress will depend, to a significant degree, on theoretical, experimental, and numerical research. One element, the devising of an efficient numerical method, is discussed in this paper.

During the last 10 years many significant contributions have been made in the development of computational methods for solving the equations of compressible viscous flow. Chief among these has been the development of noniterative, block-tridiagonal implicit methods. These methods, which are not subject to restrictive stability conditions, are much more efficient than the earlier explicit methods. The newer methods are, however, much more complicated than the earlier ones and frequently still require long computation times. The goal of the present research is to develop a method for solving the compressible form of the Navier-Stokes equations at high Reynolds number that is unconditionally stable, computationally more efficient than existing methods, and simple and

straightforward to program. The method to be described is the implicit analogue of the explicit finite-different method the author presented in 1969 (see [4]). The new method uses the 1969 method as its first stage. The second stage removes the restrictive stability condition of the 1969 method by recasting the difference equations in an implicit form. The resulting matrix equations to be solved are either upper or lower block-bidiagonal equations and are solved more easily than the block-tridiagonal matrix equations of existing methods. The method is second-order accurate in space and time and is presented in conservation form in two dimensions. Its extension to three dimensions is straightforward.

II. The Navier-Stokes Equations

In two dimensions and by neglecting body force and heat sources, the unsteady compressible form of the Navier-Stokes equations may be written in conservation form as

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0$$

where

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ e \end{pmatrix}$$

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

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

and where

$$\sigma_x = p - \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) - 2\mu\frac{\partial u}{\partial x}$$

$$\tau_{xy} = \tau_{yx} = -\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)$$

$$\sigma_y = p - \lambda\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right) - 2\mu\frac{\partial v}{\partial y}$$

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 ε and ρ by an equation of state, $p(\varepsilon, \rho)$, where $\varepsilon = e/\rho - (u^2 + v^2)/2$.

The Navier-Stokes equations adequately describe aerodynamic flow at standard temperatures and pressures. If we could efficiently solve these equations there would be no need for experimental tests when designing flight vehicles or other aerodynamic devices. As John Von Neumann said in 1946 [5], "Indeed to a great extent, experimentation in fluid dynamics is carried out under conditions where the underlying physical principles are not in doubt, where the quantities to be observed are completely determined by known equations. The purpose of experiment is not to verify a theory but to replace a computation from an unquestioned theory by direct measurements. Thus wind tunnels, for example, are used at present, at least in part, as computing devices to integrate the partial differential equations of fluid dynamics."

Unfortunately the solution of these equations for flows at high Reynolds numbers with strong viscous-inviscid interactions has defied mathematical analysis. Of the two key features of such flow — separation and turbulence — we have been able to make substantial progress during the last decade in the calculation of laminar separation using numerical methods. The calculation of turbulence largely remains an unsolved problem. Although the Navier-Stokes equations adequately describe such flows, computer speed and memory limitations make it impossible for the computational mesh to be fine enough in all spatial directions to resolve all significant eddy length scales of a high-Reynolds-number turbulent flow. As Bradshaw said in 1972 [6], "In turbulence studies we are fortunate in having a complete set of equations, the Navier-Stokes equations, whose ability to describe the motion of air at temperatures and pressures near atmospheric is not seriously in doubt (it is easy to show that the smallest significant eddies are many times larger than a molecular mean free path). We are unfortunate because numerical solution of the full time-dependent equations for turbulent flow is not practical with present computers."

In an approach that circumvents the turbulence problem, the Reynolds or "time-averaged" Navier-Stokes equations are solved. Thus, instead of seeking a time and spacially resolved solution of a rapidly fluctuating turbulent flow, only the time-averaged or mean flow solution is sought. This solution is sufficient to determine the principal quantities of interest, such as lift, drag, and heat transfer. The time-averaged equations look very similar to the original Navier-Stokes equations

except that some new terms, called Reynolds stress and turbulent heat-transfer terms, appear. These new terms represent the additional mixing caused by turbulence and are determined by models. The models vary from simple algebraic expressions to sets of additional differential equations that need to be solved. Although much progress has already been made in the understanding and modeling of the physics of turbulence, much more is needed before we will have the capability to numerically predict turbulent flow separation with confidence.

III. Numerical Method Applied to a Model Equation

Before discussing the numerical solution of the Navier-Stokes equations it is worthwhile to consider the solution of the following simpler model equation

$$\frac{\partial U}{\partial t} = -c\,\frac{\partial U}{\partial x} + \nu\,\frac{\partial^2 U}{\partial x^2}$$

The flow variable U governed by this equation convects with speed c and diffuses with kinematic viscosity ν. The implicit analogue of the author's 1969 method applied to solve this equation yields the following set of finite-difference equations [7]:

$$p: \begin{cases} \Delta U_i^n = -\dfrac{\Delta t c}{\Delta x}\left(U_{i+1}^n - U_i^n\right) + \dfrac{\Delta t \nu}{\Delta x^2}\left(U_{i+1}^n - 2U_i^n + U_{i-1}^n\right) \\ \left(1 + \dfrac{\lambda \Delta t}{\Delta x}\right)\delta U_i^{\overline{n+1}} = \Delta U_i^n + \dfrac{\lambda \Delta t}{\Delta x}\,\delta U_{i+1}^{\overline{n+1}} \\ U_i^{\overline{n+1}} = U_i^n + \delta U_i^{\overline{n+1}} \end{cases}$$

$$
c: \begin{cases}
\Delta U_i^{\overline{n+1}} = -\dfrac{\Delta t c}{\Delta x}\left(U_i^{\overline{n+1}} - U_{i-1}^{\overline{n+1}}\right) + \dfrac{\Delta t \nu}{\Delta x^2}\left(U_{i+1}^{\overline{n+1}} - 2U_i^{\overline{n+1}} + U_{i-1}^{\overline{n+1}}\right) \\
\left(1 + \dfrac{\lambda \Delta t}{\Delta x}\right)\delta U_i^{n+1} = \Delta U_i^{\overline{n+1}} + \dfrac{\lambda \Delta t}{\Delta x}\,\delta U_{i-1}^{n+1} \\
U_i^{n+1} = \dfrac{1}{2}\left(U_i^n + U_i^{\overline{n+1}} + \delta U_i^{n+1}\right)
\end{cases}
$$

where λ is chosen so that $\lambda \geq \max\{|c| + (2\nu/\Delta x) - (\Delta x/\Delta t), 0.0\}$.

The above procedure contains two steps. The first step predicts a new solution at time $t = (n+1)\Delta t$ at each mesh point i from the known solution at time $t = n\Delta t$, using a one-sided difference to approximate the first derivative term and a centered difference for the second derivative. The second stage of the predictor step enables the locally calculated solution changes ΔU_i^n to travel and diffuse throughout the flow field and then calculates implicitly the solution change $\delta U_i^{\overline{n+1}}$ to be used in the third and final stage to determine the predicted solution $U_i^{\overline{n+1}}$. The second step, or the corrector step, of the procedure is similar except that it uses opposite one-sided differences to approximate first derivatives.

The second stage of each step represents an implicit approximation to the following equation

$$
\frac{\partial \Delta t \dfrac{\partial U}{\partial t}}{\partial t} = \pm\lambda \frac{\partial \Delta t \dfrac{\partial U}{\partial t}}{\partial x}
$$

with

$$
\Delta U_i^n = \Delta t \frac{\partial U^n}{\partial t_i}
$$

and

$$
\delta U_i^{\overline{n+1}} = \Delta t \frac{\partial U^{\overline{n+1}}}{\partial t_i}, \text{ etc.}
$$

This equation describes the spreading of the solution change $\Delta t(\partial U/\partial t)$, a term of order Δt, with speed $-\lambda$ in the predictor and $+\lambda$ in the corrector. The net effect, if λ is of the order of unity, is the addition to the equations of motion of a term of second order (the difference of two first-order terms). The spreading equation is also related to that obtained by differentiating the model equation by t.

The philosophy behind the procedure is as follows. First the rate of solution change is calculated locally at each mesh point, using an explicit approximation to the governing physical equations. This local rate of change is only valid for a short time, equal approximately to the time required for a flow disturbance to travel from one mesh point to its neighbor. Explicit procedures are restricted, usually for stability reasons, to time steps Δt less than or equal to this characteristic disturbance transit time. Second, this time-step restriction is removed in the second stage by allowing the locally determined rates of solution change to convect and diffuse globally throughout the flow field, governed by an equation related to the physics of the flow. This latter equation is solved implicitly to determine the rate of solution change at each point that approximates the actual rate during the entire interval Δt.

The method is, according to linear theory, unconditionally stable (unbounded Δt), requires the solution of bidiagonal equations only, and is second-order accurate under the constraint that $\nu\Delta t/\Delta x^2$ remains bounded as Δt and Δx approach zero (i.e., λ remains of the order of unity).

Note that if the quantity λ is zero, the second stages reduce to

$$\delta U_i^{\overline{n+1}} = \Delta U_i^n$$

and

$$\delta U_i^{n+1} = \Delta U_i^{n+1}$$

or no implicit procedure at all, and the method is identical to the 1969 method. Such a choice for λ results if the chosen time step Δt already satisfies the stability condition of the 1969 method,

$$\Delta t \leq \frac{\Delta x}{|c| + \frac{2\nu}{\Delta x}}$$

Because of this feature, the method has an advantage in numerical efficiency over existing implicit methods. Not only are the numerical procedures simpler — bidiagonal versus tridiagonal — but in flow regions for which Δt satisfies the above stability condition, the method reduces from an implicit to a simpler explicit one.

IV. Numerical Method Applied to the Navier-Stokes Equations

Applying the method to solve the Navier-Stokes equations we obtain the following implicit predictor-corrector set of finite-difference equations.

$$p: \begin{cases} \Delta U_{i,j}^n = -\Delta t\left(\dfrac{\Delta_+ F_{i,j}^n}{\Delta x} + \dfrac{\Delta_+ G_{i,j}^n}{\Delta y}\right) \\ \left(I - \Delta t\,\dfrac{\Delta_+|A|\cdot}{\Delta x}\right)\left(I - \Delta t\,\dfrac{\Delta_+|B|\cdot}{\Delta y}\right)\delta U_{i,j}^{\overline{n+1}} = \Delta U_{i,j}^n \\ U_{i,j}^{\overline{n+1}} = U_{i,j}^n + \delta U_{i,j}^{\overline{n+1}} \end{cases}$$

$$
c: \begin{cases}
\Delta U_{i,j}^{\overline{n+1}} = -\Delta t \left(\dfrac{\Delta_- F_{i,j}^{\overline{n+1}}}{\Delta x} + \dfrac{\Delta_- G_{i,j}^{\overline{n+1}}}{\Delta y} \right) \\
\left(I + \Delta t \dfrac{\Delta_- |A| \cdot}{\Delta x} \right) \left(I + \Delta t \dfrac{\Delta_- |B| \cdot}{\Delta y} \right) \delta U_{i,j}^{n+1} = \Delta U_{i,j}^{\overline{n+1}} \\
U_{i,j}^{n+1} = \dfrac{1}{2} \left(U_{i,j}^{n} + U_{i,j}^{\overline{n+1}} + \delta U_{i,j}^{\overline{n+1}} \right)
\end{cases}
$$

where $\Delta_+/\Delta x$, $\Delta_-/\Delta x$, $\Delta_+/\Delta y$ and $\Delta_-/\Delta y$ are difference operators defined by

$$\frac{\Delta_+ Z_{i,j}}{\Delta x} = \frac{Z_{i+1,j} - Z_{i,j}}{\Delta x}$$

$$\frac{\Delta_- Z_{i,j}}{\Delta x} = \frac{Z_{i,j} - Z_{i-1,j}}{\Delta x}$$

$$\frac{\Delta_+ Z_{i,j}}{\Delta y} = \frac{Z_{i,j+1} - Z_{i,j}}{\Delta y}$$

and

$$\frac{\Delta_- Z_{i,j}}{\Delta y} = \frac{Z_{i,j} - Z_{i,j-1}}{\Delta y}$$

As for the model equation, the first derivative terms are one-sided differenced (as shown above) and the second derivative terms are centrally differenced. The matrices $|A|$ and $|B|$ are matrices with positive eigenvalues and are related to the Jacobians of F and G. Let S_x, S_y, and their inverses denote the matrices that diagonalize A and B with $\mu = \lambda = k = 0$ (viscous terms neglected). If the gas equation of state is perfect, $p = (\gamma - 1)\rho\varepsilon$, $A = S_x^{-1}\Lambda_A S_x$, and $B = S_y^{-1}\Lambda_B S_y$, where

$$S_x = \begin{pmatrix} 1 & 0 & 0 & -1/c^2 \\ 0 & \rho c & 0 & 1 \\ 0 & 0 & 1 & 0 \\ 0 & -\rho c & 0 & 1 \end{pmatrix} \begin{pmatrix} 1 & 0 & 0 & 0 \\ -u/\rho & 1/\rho & 0 & 0 \\ -v/\rho & 0 & 1/\rho & 0 \\ \alpha\beta & -u\beta & -v\beta & \beta \end{pmatrix}$$

$$S_y = \begin{pmatrix} 1 & 0 & 0 & -1/c^2 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & \rho c & 1 \\ 0 & 0 & -\rho c & 1 \end{pmatrix} \begin{pmatrix} 1 & 0 & 0 & 0 \\ -u/\rho & 1/\rho & 0 & 0 \\ -v/\rho & 0 & 1/\rho & 0 \\ \alpha\beta & -u\beta & -v\beta & \beta \end{pmatrix}$$

$$\Lambda_A = \begin{pmatrix} u & 0 & 0 & 0 \\ 0 & u+c & 0 & 0 \\ 0 & 0 & u & 0 \\ 0 & 0 & 0 & u-c \end{pmatrix} \qquad \Lambda_B = \begin{pmatrix} v & 0 & 0 & 0 \\ 0 & v & 0 & 0 \\ 0 & 0 & v+c & 0 \\ 0 & 0 & 0 & v-c \end{pmatrix}$$

and where $c = \sqrt{\gamma p/\rho}$ is the speed of sound, $\alpha = (1/2)(u^2 + v^2)$ and $\beta = \gamma - 1$.

The matrices S_x and S_y are each given above as the product of two matrices. For each, the right matrix represents a transformation from conservative to nonconservative variables, for example, from $(\delta\rho, \delta\rho u, \delta\rho v, \delta e)$ to $(\delta\rho, \delta u, \delta v, \delta p)$. The left matrix transforms from nonconservative to characteristic form $(\delta\rho - \delta p/c^2, \rho c\delta u + \delta p, \delta v, -\rho\delta u + \delta p)$ and $(\delta\rho - \delta p/c^2, \delta u, \rho c\delta v + \delta p, -\rho c\delta v + \delta p)$ for the S_x and S_y matrices, respectively. The inverses S_x^{-1} and S_y^{-1} are simple to derive. The matrices $|A|$ and $|B|$ are defined by

$$|A| = S_x^{-1} D_A S_x \quad \text{and} \quad |B| = S_y^{-1} D_B S_y$$

where

$$D_A = \begin{pmatrix} \lambda_{A1} & 0 & 0 & 0 \\ 0 & \lambda_{A2} & 0 & 0 \\ 0 & 0 & \lambda_{A3} & 0 \\ 0 & 0 & 0 & \lambda_{A4} \end{pmatrix} , \qquad D_B = \begin{pmatrix} \lambda_{B1} & 0 & 0 & 0 \\ 0 & \lambda_{B2} & 0 & 0 \\ 0 & 0 & \lambda_{B3} & 0 \\ 0 & 0 & 0 & \lambda_{B4} \end{pmatrix}$$

$$\lambda_{A1} = \max \left\{ |u| + \frac{2\nu}{\rho\Delta x} - \frac{1}{2}\frac{\Delta x}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{A2} = \max \left\{ |u + c| + \frac{2\nu}{\rho\Delta x} - \frac{1}{2}\frac{\Delta x}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{A3} = \max \left\{ |u| + \frac{2\nu}{\rho\Delta x} - \frac{1}{2}\frac{\Delta x}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{A4} = \max \left\{ |u - c| + \frac{2\nu}{\rho\Delta x} - \frac{1}{2}\frac{\Delta x}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{B1} = \max \left\{ |v| + \frac{2\nu}{\rho\Delta y} - \frac{1}{2}\frac{\Delta y}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{B2} = \max \left\{ |v| + \frac{2\nu}{\rho\Delta y} - \frac{1}{2}\frac{\Delta y}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{B3} = \max \left\{ |v + c| + \frac{2\nu}{\rho\Delta y} - \frac{1}{2}\frac{\Delta y}{\Delta t} \quad , \quad 0.0 \right\}$$

$$\lambda_{B4} = \max \left\{ |v - c| + \frac{2\nu}{\rho\Delta y} - \frac{1}{2}\frac{\Delta y}{\Delta t} \quad , \quad 0.0 \right\}$$

and

$$\nu = \max \{\mu, \lambda + 2\mu, k\}$$

Viscous effects are included through the use of the viscous coefficient ν. For some test problems this coefficient had to be increased during the initial part of the calculation when large transients in the solution occurred.

For regions of the flow in which Δt satisfies the following explicit stability conditions,

$$\Delta t \leq \frac{1}{2} \frac{1}{(|u| + c)/\Delta x + (2\nu/\rho\Delta x^2)}$$

and

$$\Delta t \leq \frac{1}{2} \frac{1}{(|v| + c)/\Delta y + (2\nu/\rho\Delta y^2)}$$

all λ_A and all λ_B vanish and the set of difference equations reduces to the 1969 explicit equations. For other regions in which neither relation is satisfied, the resulting difference equations are either upper or lower block bidiagonal equations with fairly straightforward solutions.

V. Numerical Results

The method was applied to solve for the interaction of a shock wave incident upon a boundary layer. The flow field is sketched in Fig. 1. As shown in Fig. 2, the initial condition was that of uniform flow, and the condition at the top mesh boundary was such that a shock wave of given strength would be generated and impinge upon a flat plate at a given point. The conditions at the upstream boundary were held fixed at their initial supersonic free-stream values; the downstream boundary conditions were obtained by zero-order interpolation; the lower boundary conditions were obtained by reflection.

The mesh contained 32 × 32 points, with 16 spanning the boundary layer. The time step was chosen so that the free stream moved approximately 1% during each time step. With this choice the time step satisfied the

explicit stability conditions everywhere except in the fine mesh spanning the boundary layer.

In Fig. 3, the computed results are compared with experiment [8] and with boundary-layer theory in the absence of a shock wave [9]. The results are for Mach 2 laminar flow at a Reynolds number of 2.95×10^5. The calculation (1) used Sutherland's formula to calculate molecular viscosity; (2) was run for 256 time steps, at which time the mesh was rezoned to cover just the interaction region; and (3) was run for an additional 256 time steps. It required about 1.5 min of computer time on a CDC 7600 computer. The results for skin friction and surface pressure compare favorably with those of theory and experiment.

In Fig. 4 the calculated velocity profiles ahead, within, and aft of the separation region are compared with the computed results, using the 1969 method alone. The two sets of results agree closely; however, the computer time required by the newer method was more than an order of magnitude less than that required by the 1969 method.

The computation times for a series of laminar and turbulent boundary-layer interactions with shock waves are given in Table 1. For each problem the flow was computed to the same physical time, which for the new method required 256 time steps. For the turbulent flow cases, a simple algebraic eddy viscosity model [10] was used to account for the effects of turbulence. For each case, the table shows the Reynolds number, the ratio of the time step used to the maximum allowed by explicit stability conditions (Courant-Friedrich-Lewy number, or CFL), the computer time required per time step per grid point on a CDC 7600 computer, and the total computer

time required. The tabulated results show that the new method is one to three orders of magnitude more efficient than the 1969 method.

For the test cases considered, the new method is more than twice as fast per time step per grid point as the block-tridiagonal implicit methods in use today. Part of the reason for this is the mesh-point spacing and time step chosen; of the total number of mesh points more than half required only the use of the 1969 explicit method. At these mesh points the chosen time step already satisfied the local explicit stability condition; therefore, the implicit procedure, the second stage, was omitted. The implicit procedures were required only in the fine mesh spanning the boundary layer, where explicit stability conditions would have imposed a severe time-step restriction. It is estimated that if the implicit procedures were used at each grid point, the time step per grid point for a two-dimensional calculation would be 2.45×10^{-4} sec for laminar flow and 2.75×10^{-4} sec for turbulent flow. The difference between these last two values represents the additional computation needed to evaluate the turbulence model relations.

VI. Conclusion

A new numerical method has been devised for solving the equations of compressible viscous flow. The method represents the implicit analogue of the explicit method presented by the author in 1969. It is unconditionally stable, second-order accurate, and, for many applications, is more efficient than other methods in use today. Because the new method uses the 1969 method as its first stage, many existing computer programs in which the 1969 method is used can be updated by adding the described implicit second stage.

VII. References

[1] Chapman, D. R., Mark, H., and Pirtle, M. W.: Computer vs wind tunnel. Astronautics and Aeronautics (Apr. 1975), pp. 22-35.

[2] Chapman, D. R.: Computational aerodynamics development and outlook. AIAA Paper 79-0129 (Jan. 1979).

[3] Chapman, D. R.: Trends and pacing items in computational aerodynamics. Proceedings of the Seventh International Conference on Numerical Methods in Fluid Dynamics (to be published as a volume in Lecture Notes in Physics, Springer-Verlag, 1981).

[4] MacCormack, R. W.: The effect of viscosity in hypervelocity impact cratering. AIAA Paper 69-354, Cincinnati, Ohio (Apr. 1969).

[5] Von Neumann, John: On the Principles of Large Scale Computing Machines. Collected Works, Vol. V, Macmillan Co. (1963).

[6] Bradshaw, P.: The understanding and prediction of turbulent flow. (Sixth Reynolds-Prandtl Lecture) Aeronaut. J., Vol. 76 (1972), pp. 403-418.

[7] MacCormack, R. W.: A numerical method for solving the equations of compressible viscous flow. AIAA Paper 81-0110, St. Louis, Missouri (Jan. 1981).

[8] Hakkinen, R. J., Greber, I., Trilling, L., and Abarbanel, S. S.: The interaction of an oblique shock wave with a laminar boundary layer. NASA Memorandom 2-18-59w (1959).

[9] Van Driest, F. R.: Investigation of laminar boundary layers in compressible fluids using the Crocco method. NACA TN-2597 (Jan. 1952).

[10] Cebeci, T. and Smith, A. M. O.: Analysis of Turbulent Boundary Layers. Academic Press, New York (1974).

Table 1 Computation time

Case	Method	CFL	CDC 7600 time / step grid point	Total time
Laminar	1969	0.9	1.25×10^{-4} sec	12 min
$R = 3 \times 10^5$	New	20	1.55×10^{-4} sec	41 sec
Turbulent	1969	.9	1.55×10^{-4} sec	2 hr*
$R = 3 \times 10^6$	New	160	1.85×10^{-4} sec	48 sec
Turbulent	1969	.9	1.55×10^{-4} sec	15 hr*
$R = 3 \times 10^7$	New	1200	1.85×10^{-4} sec	48 sec

*Estimated.

Figure Captions

Fig. 1 Sketch of shock, boundary-layer interaction

Fig. 2 Initial flow field for shock, boundary-layer interaction

Fig. 3 Comparison of results. a) Surface pressure. b) Skin friction

Fig. 4 Comparison of velocity profiles

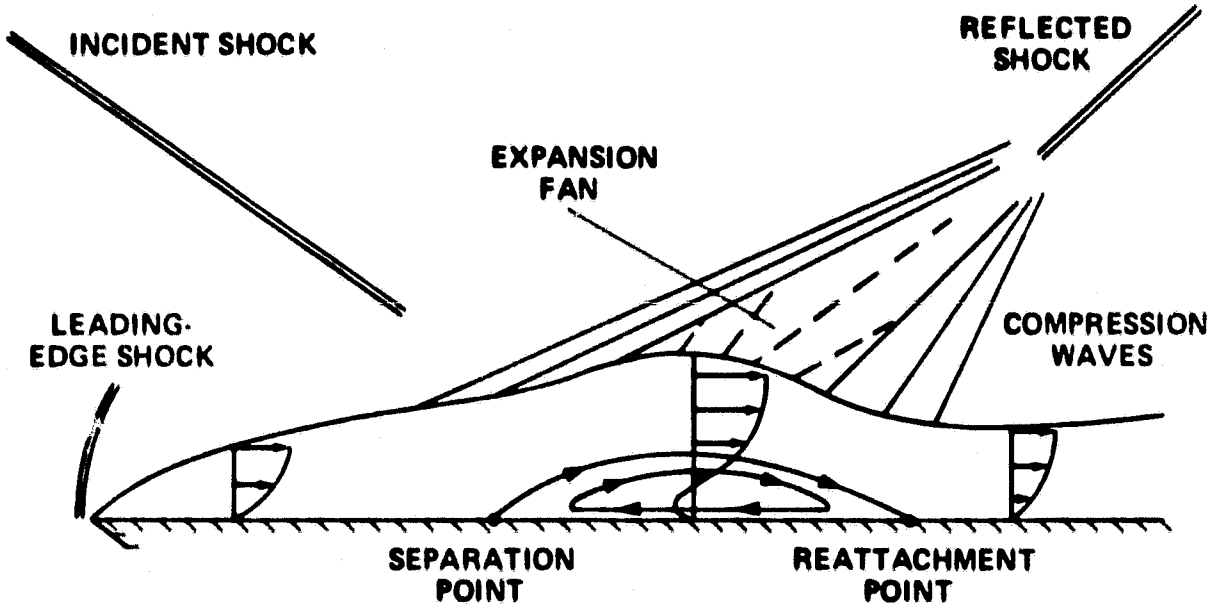
INCIDENT SHOCK
REFLECTED SHOCK
EXPANSION FAN
LEADING-EDGE SHOCK
COMPRESSION WAVES
SEPARATION POINT
REATTACHMENT POINT

Fig. 1

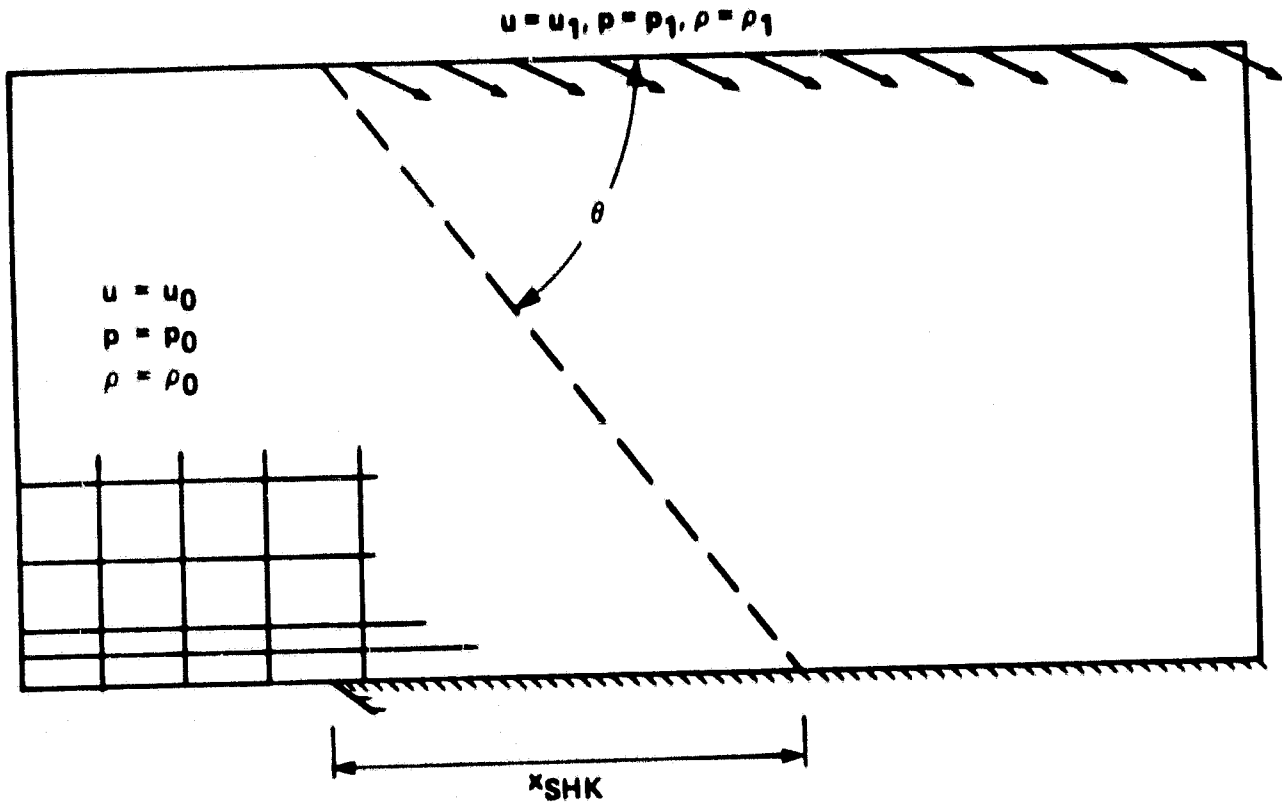

Fig. 2

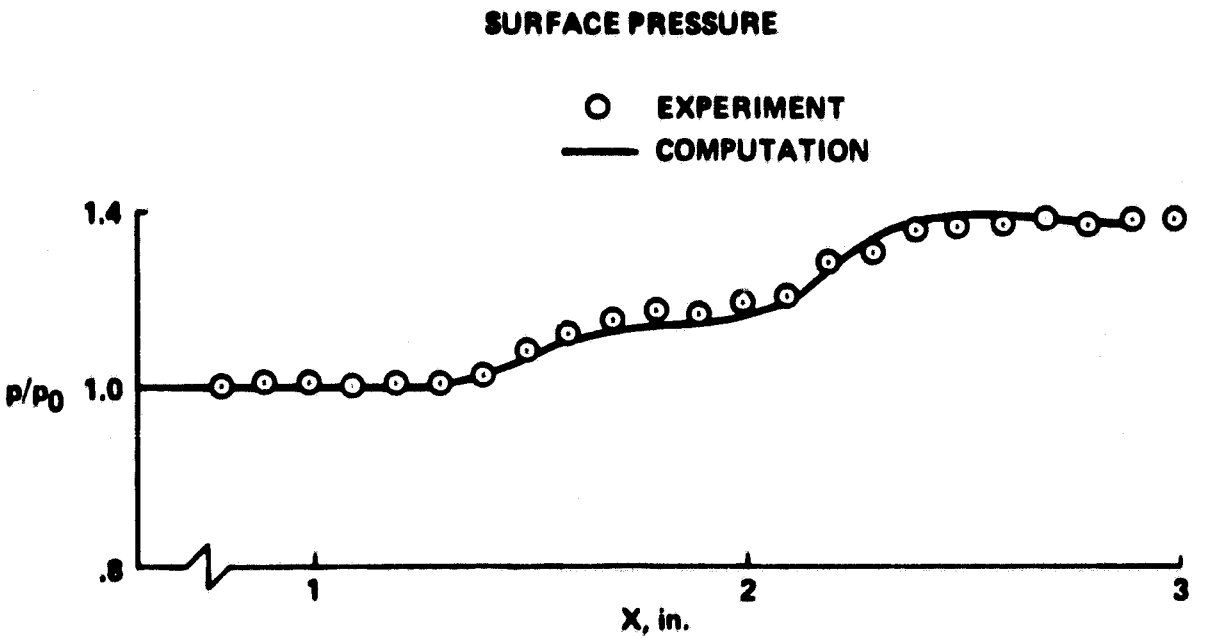
SURFACE PRESSURE
EXPERIMENT
COMPUTATION
1.4
p/p0
1.0
.8
1
2
3
X, in.

Fig. 3a

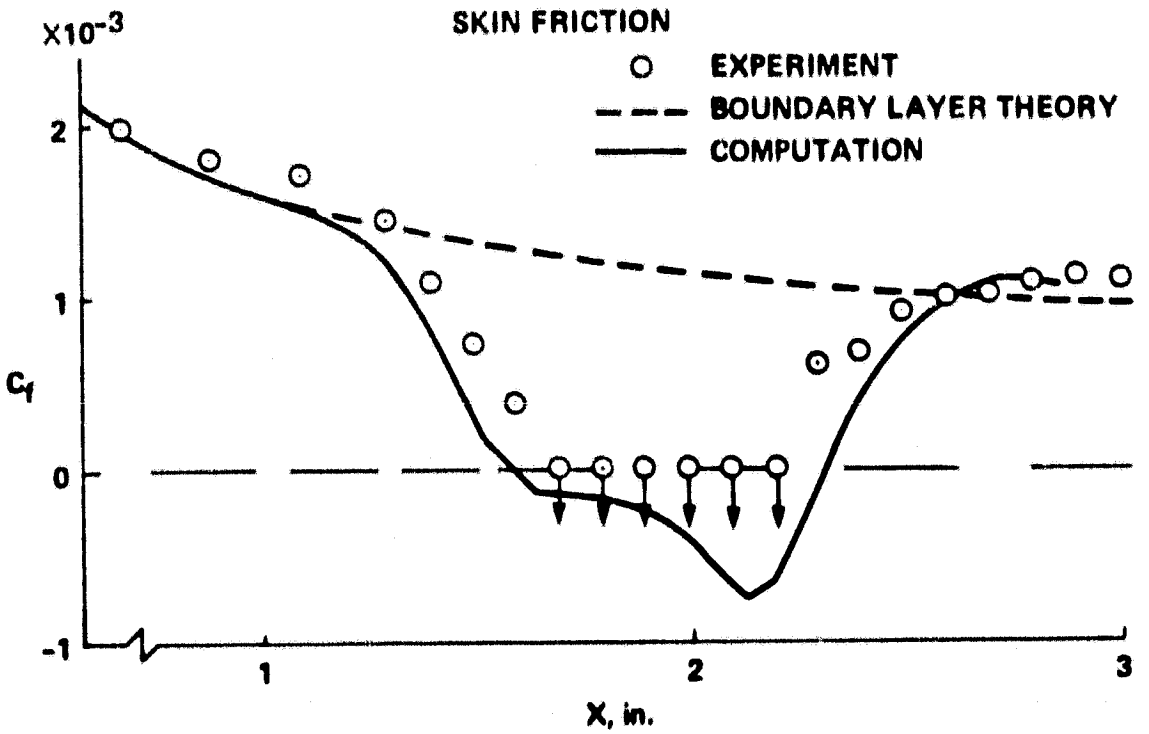
SKIN FRICTION
X10-3
EXPERIMENT
BOUNDARY LAYER THEORY
COMPUTATION
2
1
0
-1
Cf
1
2
3
X, in.

Fig. 3b

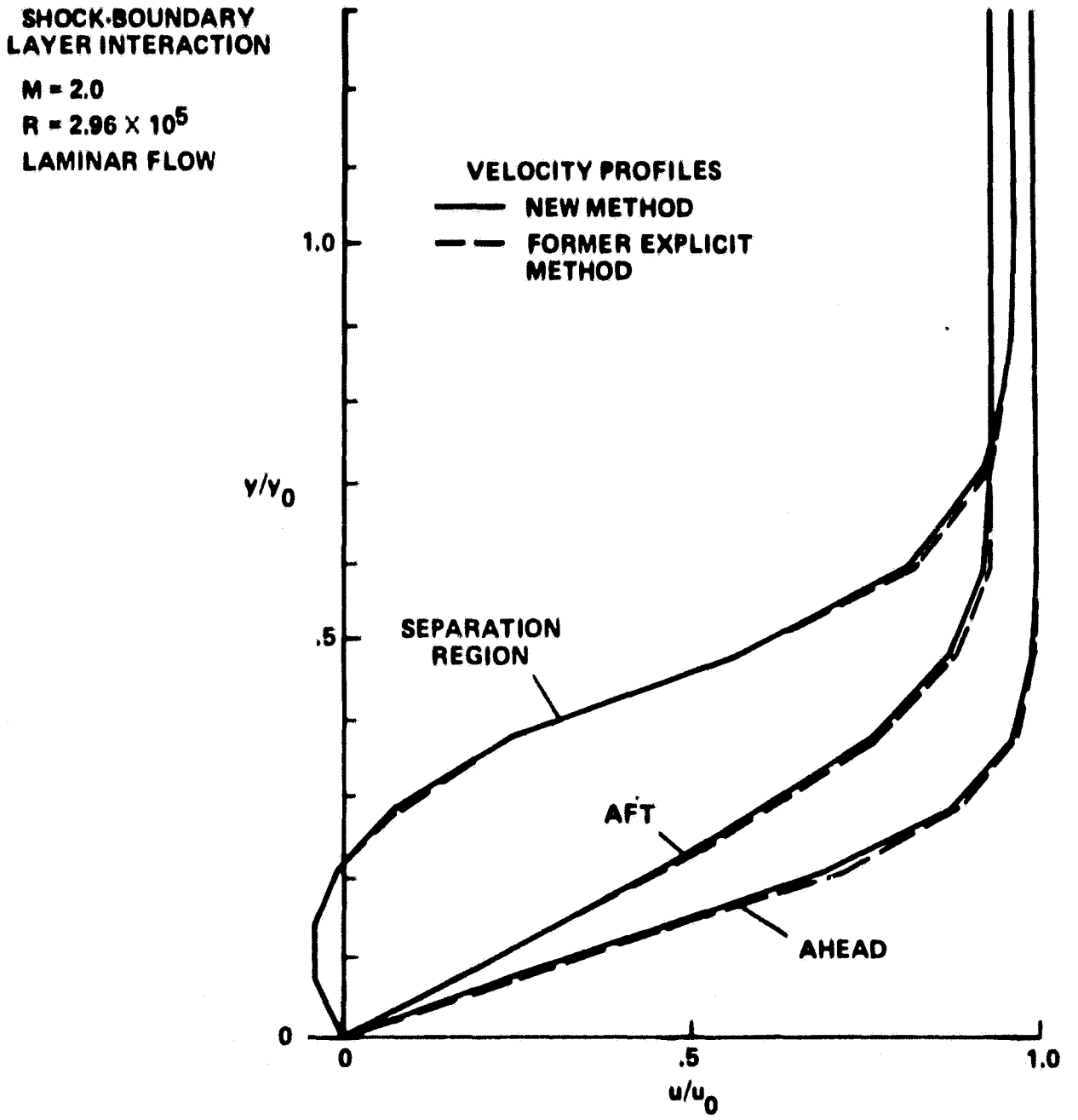
SHOCK-BOUNDARY LAYER INTERACTION
M = 2.0
R = 2.96 X 10^5
LAMINAR FLOW
VELOCITY PROFILES
NEW METHOD
FORMER EXPLICIT METHOD
1.0
.5
0
y/y0
SEPARATION REGION
AFT
AHEAD
0
.5
1.0
u/u0

Fig. 4