NASA Technical Paper 1934

November 1981

Some Observations on a New Numerical Method for Solving the Navier-Stokes Equations

Ajay Kumar

NASA

NASA
Technical
Paper
1934

1981

Some Observations on a New Numerical Method for Solving the Navier-Stokes Equations

Ajay Kumar
Langley Research Center
Hampton, Virginia

NASA
National Aeronautics
and Space Administration

**Scientific and Technical
Information Branch**

SUMMARY

A new explicit-implicit method for solving Navier-Stokes equations has been studied. The method has second-order accuracy in space and time, preserves the conservation form, and is much less complex than other implicit methods since it requires no block or scalar tridiagonal inversions. It is used to solve a complex, two-dimensional, steady-state, supersonic-flow problem. A discussion is given of the computational efficiency of the new method and of the quality of the solution obtained from it at high Courant-Friedrich-Lewy (CFL) numbers. Modifications are discussed and certain observations are made about the method which may be helpful in using it successfully.

INTRODUCTION

Many numerical methods have been developed in the past several years to solve compressible Navier-Stokes equations, but the emphasis recently has been to develop implicit methods which are not subject to the conventional explicit stability conditions. However, the implicit methods are still restricted in time-step size by accuracy and stability of the solution. Also, their programming complexity and computing time per time-step are much greater than those of the explicit methods. Recently, MacCormack (ref. 1) has developed a new method which is an implicit analog of his unsplit explicit method (ref. 2). This new method removes the explicit stability conditions, thus allowing larger time-step size to advance the solution. It can be added easily to existing codes which use the explicit method of reference 2. Further, the new method has second-order accuracy in space and time, preserves the conservation form, and requires no block or scalar tridiagonal inversions. Reference 1 describes the method in detail and discusses the computational efficiencies obtained from it for a test problem.

The gain in computational efficiency from the new method and its simplicity to be programmed make it very attractive over the other complex implicit methods. This method has been incorporated into an existing code developed in reference 3 which uses the unsplit explicit method of reference 2. The code is operational on the Control Data CYBER 203 vector processing computer. The purpose of the present report is to indicate modifications that were made and to make certain observations which may be helpful in using the new method successfully. The method has been applied to a two-dimensional supersonic-flow problem involving shock and expansion waves and their interactions with each other and with the boundary layer. A discussion of the computational efficiency and the quality of the solution with increasing time-step size is also presented.

SYMBOLS

CFL	Courant-Friedrich-Lewy limit
c	velocity of sound
c_v	specific heat at constant volume

e	total energy per unit volume
h	static enthalpy
M	Mach number
N_{Pr}	Prandtl number
n	number of time-steps
p	pressure
R	gas constant
T	temperature
t	time
Δt	time-step size
u	velocity in x-direction
v	velocity in y-direction
x,y	Cartesian coordinates
Δx	grid spacing in x-direction
Δy	grid spacing in y-direction
γ	ratio of specific heats
μ	effective viscosity, $\mu_\ell + \mu_t$
ρ	density

Subscripts:

ℓ	laminar
t	turbulent
∞	free stream

GOVERNING EQUATIONS

The two-dimensional Navier-Stokes equations in conservation form are used to describe the flow field. These can be written as

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

where

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

$$F = \begin{bmatrix} \rho u \\ \rho u^2 + \sigma_x \\ \rho uv + \tau_{xy} \\ (e + \sigma_x)u + \tau_{yx} v + q_x \end{bmatrix}$$

$$G = \begin{bmatrix} \rho v \\ \rho uv + \tau_{yx} \\ \rho v^2 + \sigma_y \\ (e + \sigma_y)v + \tau_{xy} u + q_y \end{bmatrix}$$

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

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

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

$$\lambda = -2\mu/3$$

$$e = \rho\left(c_v T + \frac{u^2 + v^2}{2}\right)$$

$$q_x = -\left(\frac{\mu_\ell}{N_{Pr,\ell}} + \frac{\mu_t}{N_{Pr,t}}\right)\frac{\partial h}{\partial x}$$

$$q_y = -\left(\frac{\mu_\ell}{N_{Pr,\ell}} + \frac{\mu_t}{N_{Pr,t}}\right)\frac{\partial h}{\partial y}$$

In order to complete the set of governing equations, the equation of state $p = \rho RT$ is used. In the aforementioned equations, μ is the sum of laminar viscosity and turbulent viscosity. The laminar viscosity for air is calculated from Sutherland's formula. The turbulent viscosity is calculated from an algebraic, two-layer, eddy-viscosity model due to Baldwin and Lomax (ref. 4).

METHOD OF SOLUTION

The governing equations are solved by a new method, developed by MacCormack (ref. 1), which is an implicit analog of his unsplit explicit method (ref. 2). The method consists of a predictor step and a corrector step which can be written as follows (nomenclature used is same as that in ref. 1):

Predictor step:

$$\Delta U^n_{i,j} = -\Delta t\left[\left(F^n_{i+1,j} - F^n_{i,j}\right)\Big/\Delta x + \left(G^n_{i,j+1} - G^n_{i,j}\right)\Big/\Delta y\right]$$

$$\left(I + \frac{\Delta t}{\Delta x}|A|^n_{i,j}\right)\delta U^*_{i,j} = \Delta U^n_{i,j} + \frac{\Delta t}{\Delta x}|A|^n_{i+1,j}\,\delta U^*_{i+1,j}$$

$$\left(I + \frac{\Delta t}{\Delta y}|B|^n_{i,j}\right)\delta U^{\overline{n+1}}_{i,j} = \delta U^*_{i,j} + \frac{\Delta t}{\Delta y}|B|^n_{i,j+1}\,\delta U^{\overline{n+1}}_{i,j+1}$$

$$U^{\overline{n+1}}_{i,j} = U^n_{i,j} + \delta U^{\overline{n+1}}_{i,j}$$

Corrector step:

$$\Delta U^{\overline{n+1}}_{i,j} = -\Delta t\left[\left(F^{\overline{n+1}}_{i,j} - F^{\overline{n+1}}_{i-1,j}\right)\Big/\Delta x + \left(G^{\overline{n+1}}_{i,j} - G^{\overline{n+1}}_{i,j-1}\right)\Big/\Delta y\right]$$

$$\left(I + \frac{\Delta t}{\Delta x}|A|^{\overline{n+1}}_{i,j}\right)\delta U^{**}_{i,j} = \Delta U^{\overline{n+1}}_{i,j} + \frac{\Delta t}{\Delta x}|A|^{\overline{n+1}}_{i-1,j}\,\delta U^{**}_{i-1,j}$$

$$\left(I + \frac{\Delta t}{\Delta y}|B|^{\overline{n+1}}_{i,j}\right)\delta U^{n+1}_{i,j} = \delta U^{**}_{i,j} + \frac{\Delta t}{\Delta y}|B|^{\overline{n+1}}_{i,j-1}\,\delta U^{n+1}_{i,j-1}$$

$$U^{n+1}_{i,j} = \frac{1}{2}\left(U^n_{i,j} + U^{\overline{n+1}}_{i,j} + \delta U^{n+1}_{i,j}\right)$$

Each step contains two stages. The first stage uses the explicit method which is subject to restrictive explicit stability conditions. The second stage removes these stability conditions by transforming numerically the equations of the first stage into an implicit form. It is seen from the equations that the method requires the solution of upper or lower block bidiagonal equations.

Here, I is the unit matrix and δU, δU^*, and so forth, represent the change in U with time-step; $A = \partial F/\partial U$ and $B = \partial G/\partial U$ are the Jacobians of F and G. The matrices $|A|$ and $|B|$ are the matrices with positive eigenvalues and are related to the Jacobians A and B. Their definitions are given in reference 1 and are not repeated here. This method is more efficient than existing methods for solving the equations of compressible viscous flow because, for regions of the flow in which Δt satisfies the explicit stability conditions, the method reduces to the simple explicit method, and, for other regions, block bidiagonal equations need only be solved rather than the block tridiagonal equations of the existing implicit methods.

BOUNDARY CONDITIONS

The flow variables at the inflow boundary are held fixed at given free-stream values, whereas first-order extrapolation is used to obtain the flow variables at the outflow boundary. Reference 1 uses reflection-type boundary conditions which are not, in general, convenient to use. In the present calculations, no-slip boundary conditions are used along the solid surfaces.

To start the implicit sweep in the i-direction (i varying from 1 to I), $|A|\delta U_{I,j}$ is required in the predictor step and $|A|\delta U_{1,j}$ is required in the corrector step. Both of these end fluxes in the i-direction are set equal to zero for the present problem because, at the inflow boundary, the conditions are held fixed and, at the outflow boundary, the mesh spacing is large enough so that matrix $|A|$ vanishes. The implicit sweep in the j-direction (j varying from 1 to J) requires evaluation of either $|B|\delta U_{i,J}$ or $|B|\delta U_{i,1}$. These are obtained by calculating $|B|$ and δU at the boundary points. Matrix $|B|$ is calculated by using the boundary values of the flow variables from the previous step, whereas δU at the boundary mesh points is obtained based on the changes in U at the adjacent mesh points, again from the previous step. As an example, for no-slip and adiabatic-wall boundary conditions with

$$u_{i,1} = v_{i,1} = 0$$

$$p_{i,1} = p_{i,2}$$

$$T_{i,1} = T_{i,2}$$

the $(\delta U)_{i,1}$ can be written as

$$\left(\delta U_1\right)_{i,1} = \left(\delta U_1\right)_{i,2}$$

$$\left(\delta U_2\right)_{i,1} = \left(\delta U_3\right)_{i,1} = 0$$

$$\left(\delta U_4\right)_{i,1} = \left(\delta p\right)_{i,2}\Big/\left(\gamma - 1\right)$$

where U_1, U_2, U_3, and U_4 are the components of U.

ARTIFICIAL DAMPING

To maintain a stable solution, it was found necessary to use artificial damping in both the explicit and implicit stages of the method. The fourth-order damping, already present in the explicit code of reference 3, is retained for the explicit stage of the method. For the implicit stage, the damping term given in reference 1 is used. This term is written as

$$\frac{|\delta p/c^2 - \delta\rho|}{(\Delta t/\Delta y)[(\gamma - 1)/\gamma]\rho}$$

The contribution from this term becomes small as the steady state is approached. The method developed instabilities if either of the two damping terms was eliminated.

DISCUSSION OF RESULTS

The modified code is applied to a two-dimensional test problem shown in figure 1, in which the turbulent supersonic flow is calculated between two walls for air under perfect gas assumption. The upper wall is kept parallel to the free stream, whereas at the lower wall, the flow undergoes a 10° compression at 2 cm and then a 10° expansion at 4 cm. The total length of the flow domain is 10 cm with an initial height of 2 cm. This problem is chosen since it involves most of the features of a complex supersonic flow, such as shock—expansion-wave interaction and shock—boundary-layer interaction which results in a separated region. Solutions are obtained at CFL values of 1, 15, and 22.5 for inflow conditions of $M_\infty = 5$, $p_\infty = 0.1013$ MPa, and $T_\infty = 293$ K. The Reynolds number based on the free-stream conditions and the length of the flow domain is 11×10^6. An assessment is made of the efficiency of the new method on the Control Data CYBER 203 vector processing computer and of the quality of the solution obtained from the method.

The physical domain, shown in figure 1, is numerically transformed to a computational domain. Since, for the present problem, matrix $|A|$ vanished at all the grid points and the grid is not skewed in the x-direction, the numerical transformation did not affect the implicit steps. In general, matrices $|A|$ and $|B|$ need to be defined properly in the transformed coordinate system. A grid size of 51×51 is used in the calculations. To resolve the turbulent boundary layer, a stretching function is used in the transformation that allows concentration of grid points near the walls in the physical domain. The grid in the computational domain still remains equally spaced. The coefficient in the stretching function is chosen so that the first grid point away from the walls is located at approximately 4.5 μm. The computer code is written for the Control Data CYBER 203 vector processing computer. The explicit steps of the code are fully vectorized, whereas only a portion of the implicit steps could be vectorized.

To start the solution, free-stream conditions are assumed initially at all the grid points except at the boundaries where appropriate boundary conditons are used. This starting solution works well for CFL values of 1 and 15, but for CFL = 22.5, negative temperature developed near the expansion shoulder in the first few time-steps. For this case, the solution is advanced with CFL = 15 for the first 100 time-steps to establish a reasonable initial solution, and then the CFL value is increased to 22.5 over the next 200 time-steps. The solution is advanced in time until it reaches the time required by the free-stream to traverse the flow domain three times. As mentioned in reference 1, $\mu\, \Delta t/\rho(\Delta y)^2$ must be less than 0.5 to avoid any possible dependence of the steady-state solution on Δt. In the present calculations, this quantity exceeded 0.5 for CFL values of 15 and 22.5. For these cases, the time-step was successively reduced near the end of the calculations.

It is found that the present problem at high CFL required the reversal of the order of differencing for the predictor and corrector steps from one time-step to the next time-step; that is, if forward and backward differences are used in time-step n, then backward and forward differences should be used in time-step n + 1. Without this symmetric operation of the differencing, the solution at a CFL value of 10 failed in the 27th time-step. Even at a low CFL value of 2, the quality of the solution is much better with the symmetric operation.

In order to assess the quality of the solution with increasing CFL, pressure distributions on the upper and lower walls are compared at CFL values of 1, 15, and 22.5. Also compared are the velocity and pressure profiles at three locations on the upper and lower walls. These locations lie ahead of, aft of, and in the separation region. Figure 2 shows the pressure distributions on the upper and lower walls. It is seen that the lower wall pressures are almost identical at the three CFL values, whereas on the upper wall there are small differences near the 0.07-m location. This is the region where the shock from the lower wall separates the boundary layer on the upper wall.

Figures 3 and 4 show the velocity and pressure profiles, respectively, at $x = 0.05$ m, which lies ahead of the separation region. It is seen that there are negligible differences in the profiles at various CFL values. Figures 5 and 6 show the velocity and pressure profiles, respectively, at $x = 0.074$ m, which lies in the separated region. The velocity and the pressure profiles at the lower wall again compare very well, whereas the velocity profile on the upper wall shows significant differences close to the wall at various CFL values. It is seen that the extent of the separation region is reduced at higher CFL values. The differences are also present in the pressure profiles on the upper wall. The separation region may have been affected by the artificial damping in the explicit and the implicit steps which is required to keep the solution stable.

Figures 7 and 8 show the velocity and pressure profiles, respectively, at $x = 0.088$ m, which lies downstream of the separation region. Here again, the profiles compare well at various CFL values.

The increase in computing time due to the addition of implicit steps is approximately 50 percent on the Control Data CYBER 203 vector processing computer. It takes approximately 2×10^{-5} sec per grid point per time-step for the explicit method. This time increases to about 3.0×10^{-5} sec with the addition of the implicit steps which could be vectorized only partially. Even with the increased computing time per time-step, there is a significant savings in overall computing time. As an example, the total computing time required is approximately 10 times less at CFL = 15 than at CFL = 1.

As with the other implicit methods, the gain in the computing efficiency is expected to be higher with a highly stretched grid in one coordinate direction. Although reference 1 describes the method as unconditionally stable, it is not found to be so in the present calculations. For the present problem, solutions could not be obtained for a CFL value of 30 or above.

CONCLUDING REMARKS

A new explicit-implicit method has been applied to a complex, two-dimensional, steady-state, supersonic-flow problem by using Navier-Stokes equations. The method originally used reflection-type boundary conditions, but in the present calculations, no-slip boundary conditions are used. It is found for the present problem that the method requires the reversal of the order of differencing in the predictor and corrector steps from one time-step to the next time-step. It is also necessary to use artificial damping in both the explicit and implicit stages of the method to maintain a stable solution. A detailed comparison of the results at Courant-Friedrich-Lewy (CFL) values of 1, 15, and 22.5 shows that the flow field is predicted very well at high CFL values except in the immediate neighborhood of separation. For the present problem, solutions could not be obtained for a CFL value of 30 or above.

The code is operational on the Control Data CYBER 203 computer. There is approximately a 50 percent increase in computing time per time-step due to the addition of implicit steps in the explicit method, but the overall saving in the computing time is very significant. As an example, for a CFL value of 15, the computing time is reduced by a factor of 10 over a CFL value of 1. It can be concluded from this study that the method has great potential in computing complicated fluid-dynamics problems since it is computationally efficient and is relatively much simpler than other implicit methods. Existing explicit codes using the unsplit MacCormack method can easily be modified for the new method.

Langley Research Center
National Aeronautics and Space Administration
Hampton, VA 23665
September 23, 1981

REFERENCES

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

2. MacCormack, Robert W.: The Effect of Viscosity in Hypervelocity Impact Cratering. AIAA Paper No. 69-354, Apr.-May 1969.

3. Kumar, Ajay: Numerical Analysis of the Scramjet Inlet Flow Field Using Two-Dimensional Navier-Stokes Equations. AIAA-81-0185, Jan. 1981.

4. Baldwin, Barrett; and Lomax, Harvard: Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows. AIAA Paper 78-257, Jan. 1978.

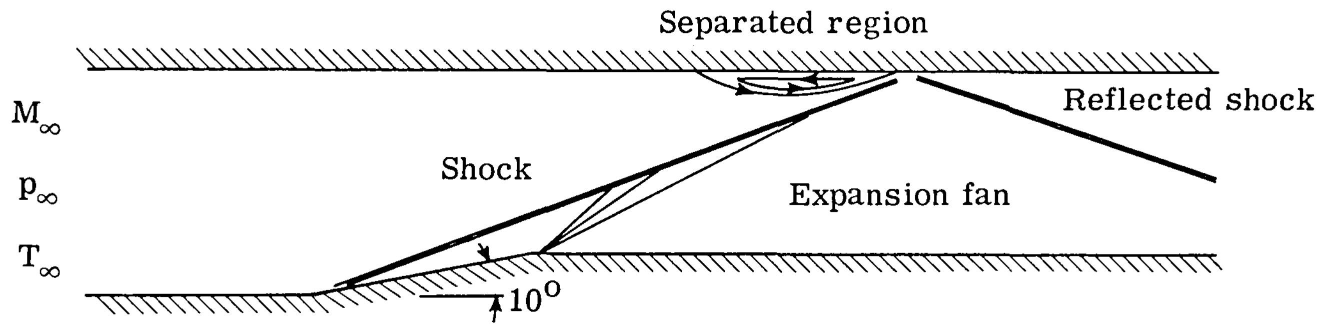
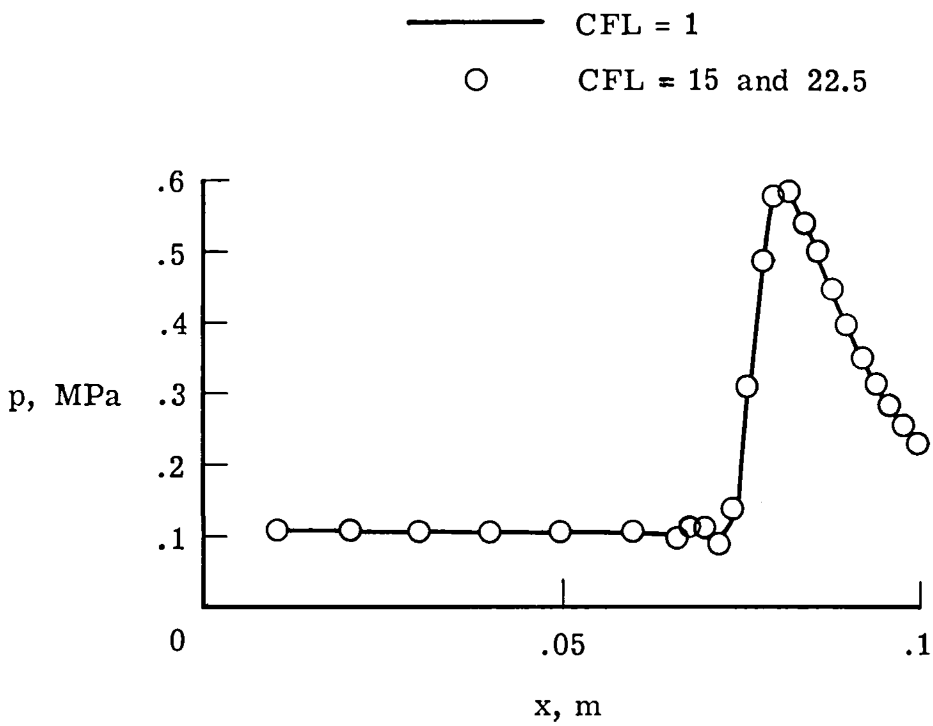
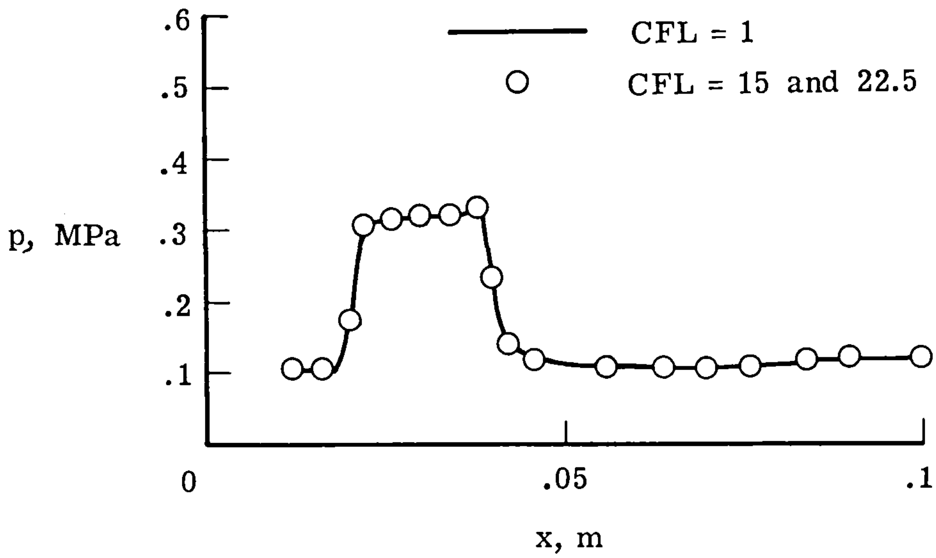


Figure 1.- Sketch of the test problem.

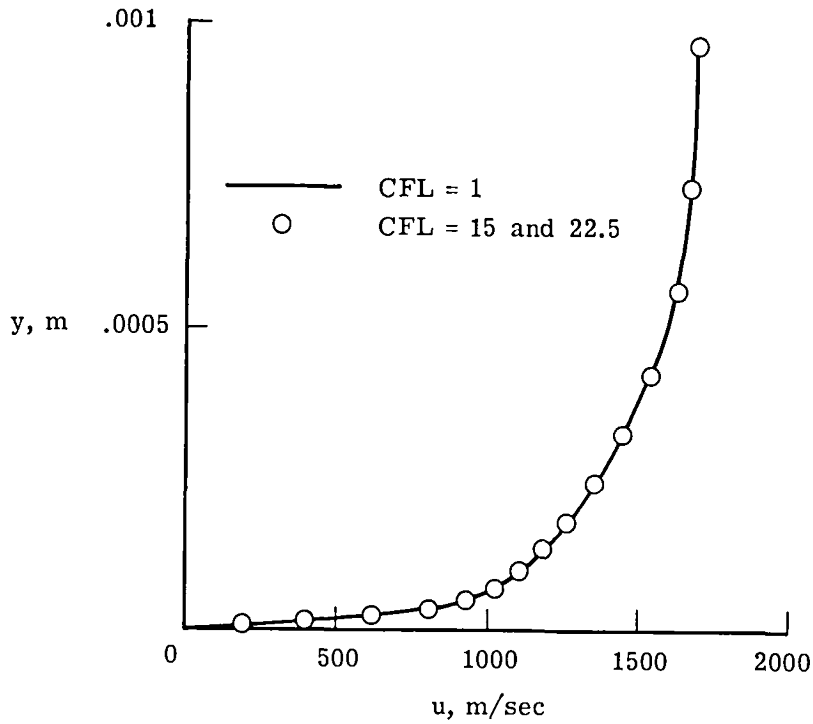


(a) Upper wall.

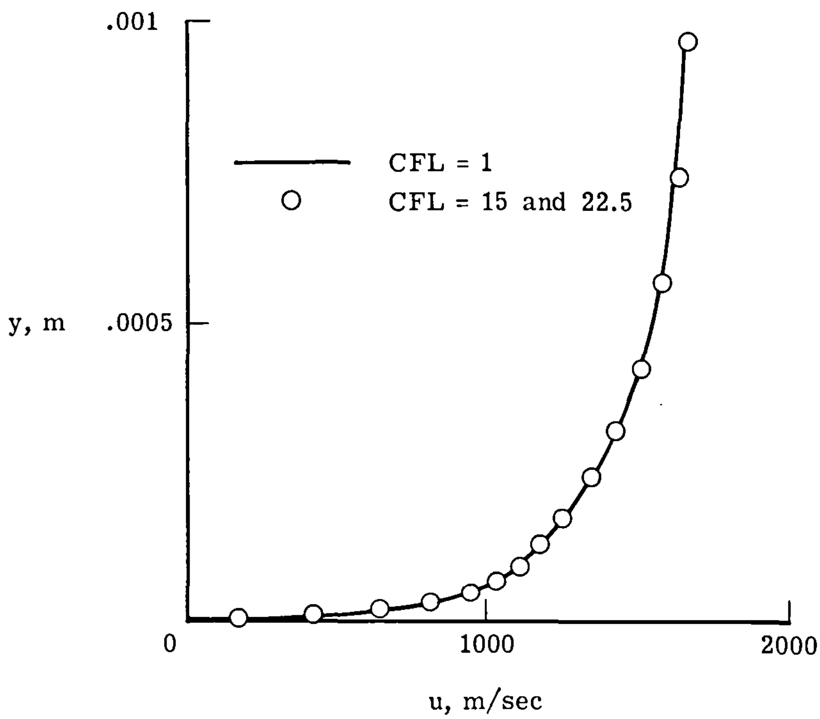


(b) Lower wall.

Figure 2.- Surface-pressure distributions.



(a) Upper wall.



(b) Lower wall.

Figure 3.- Velocity profiles at x = 0.05 m.

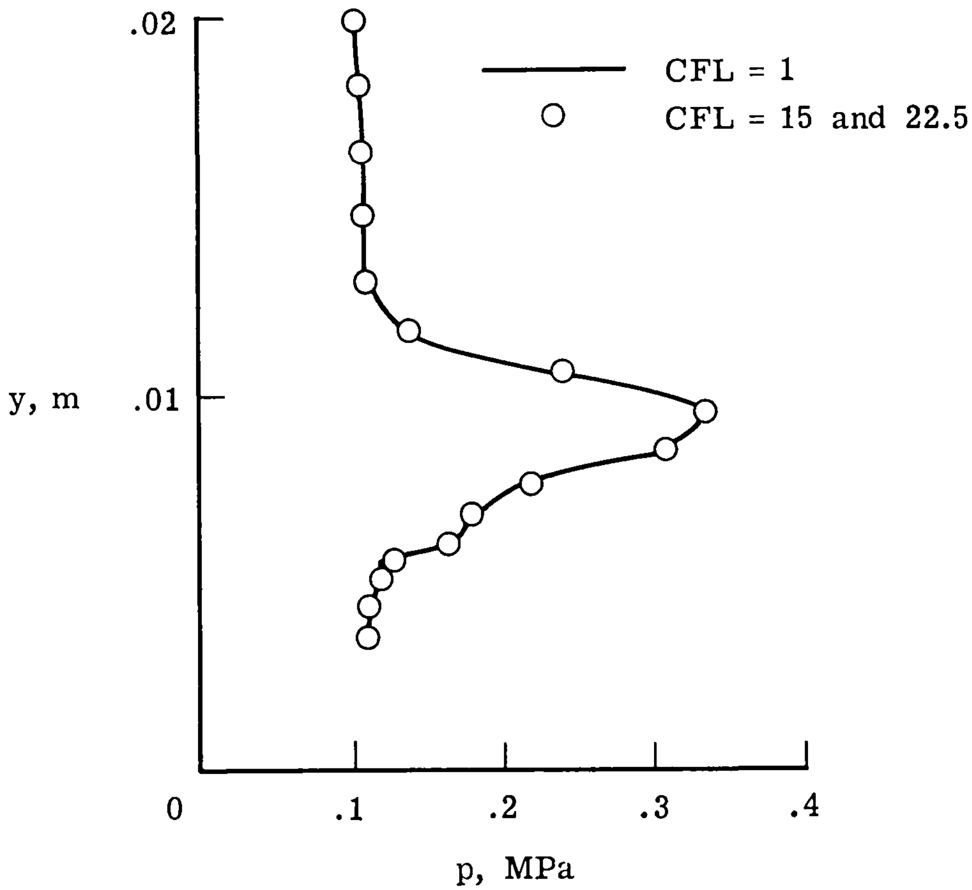
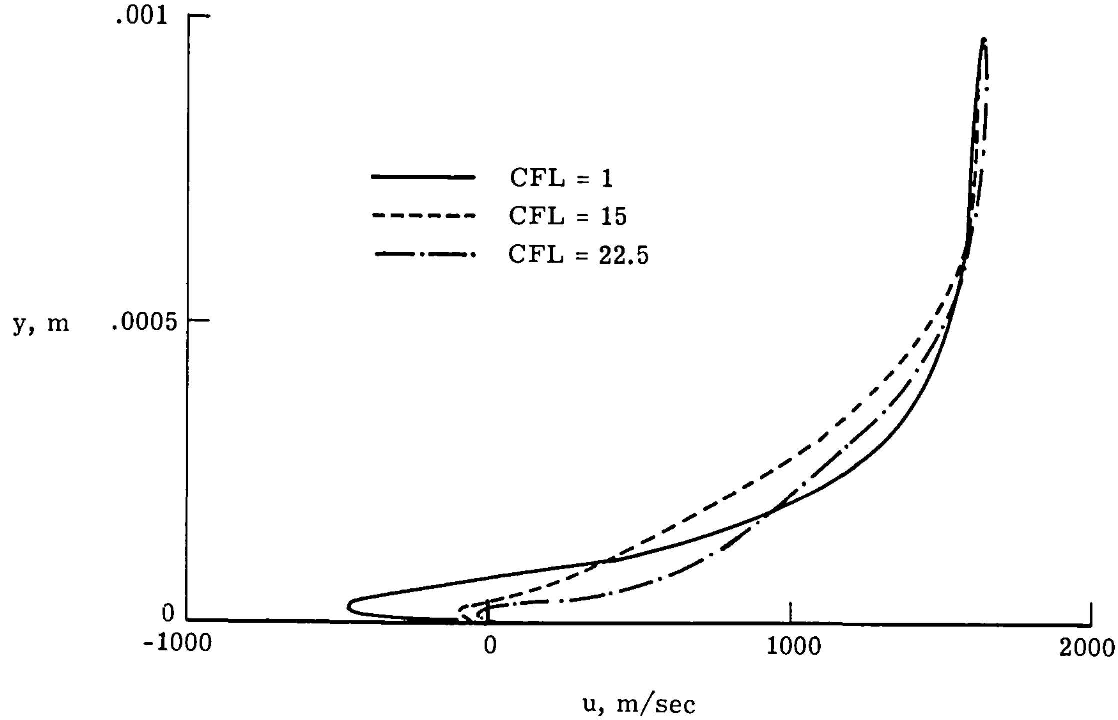
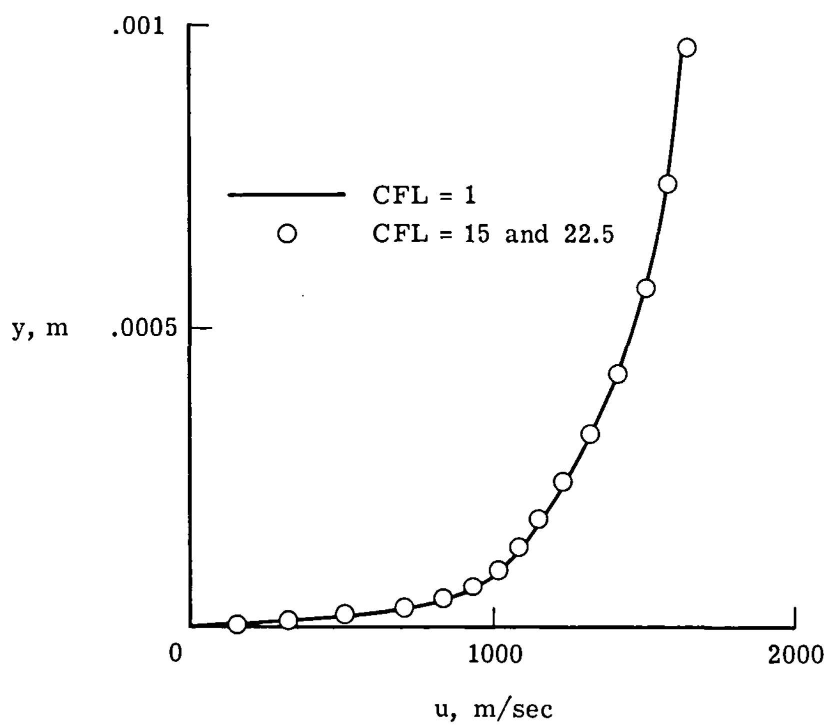


Figure 4.- Pressure profile at x = 0.05 m.



(a) Upper wall.



(b) Lower wall.

Figure 5.- Velocity profiles at $x = 0.074$ m.

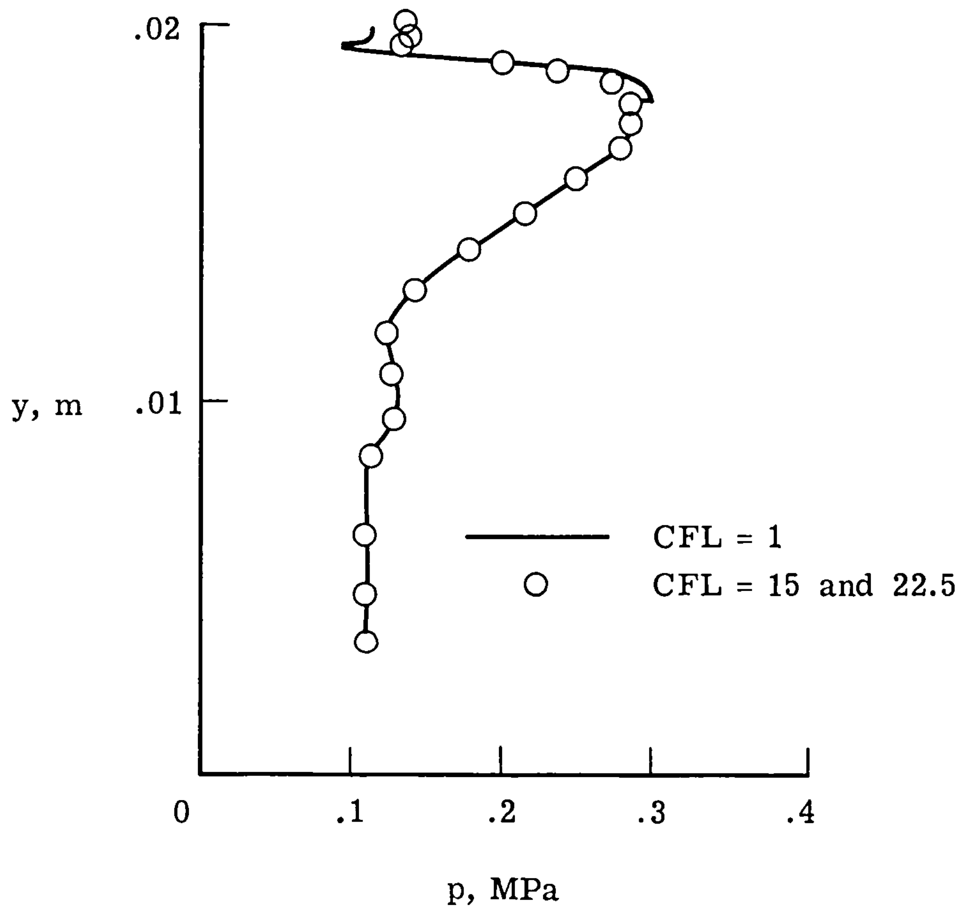
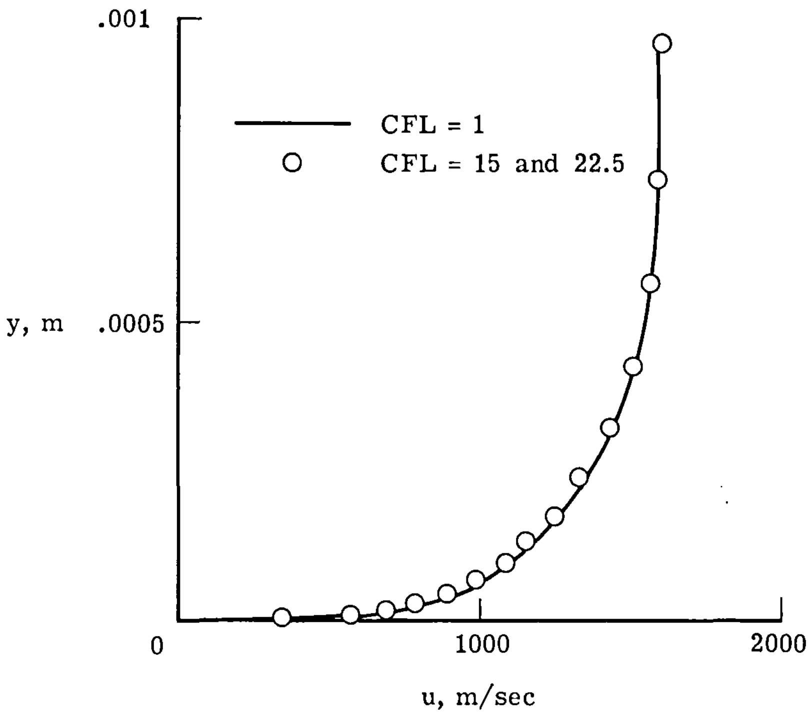
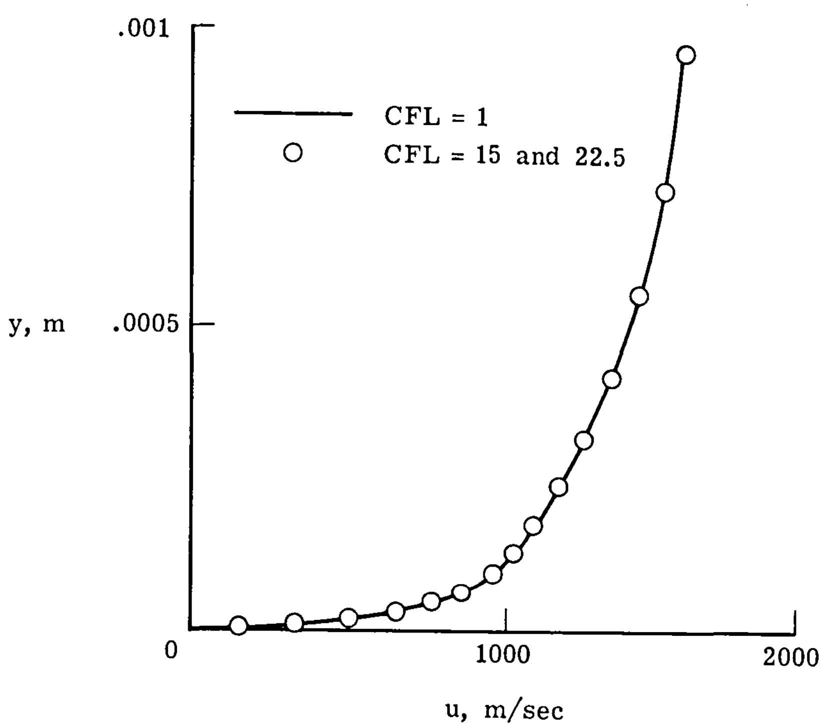


Figure 6.- Pressure profile at x = 0.074 m.



(a) Upper wall.



(b) Lower wall.

Figure 7.- Velocity profiles at $x = 0.088$ m.

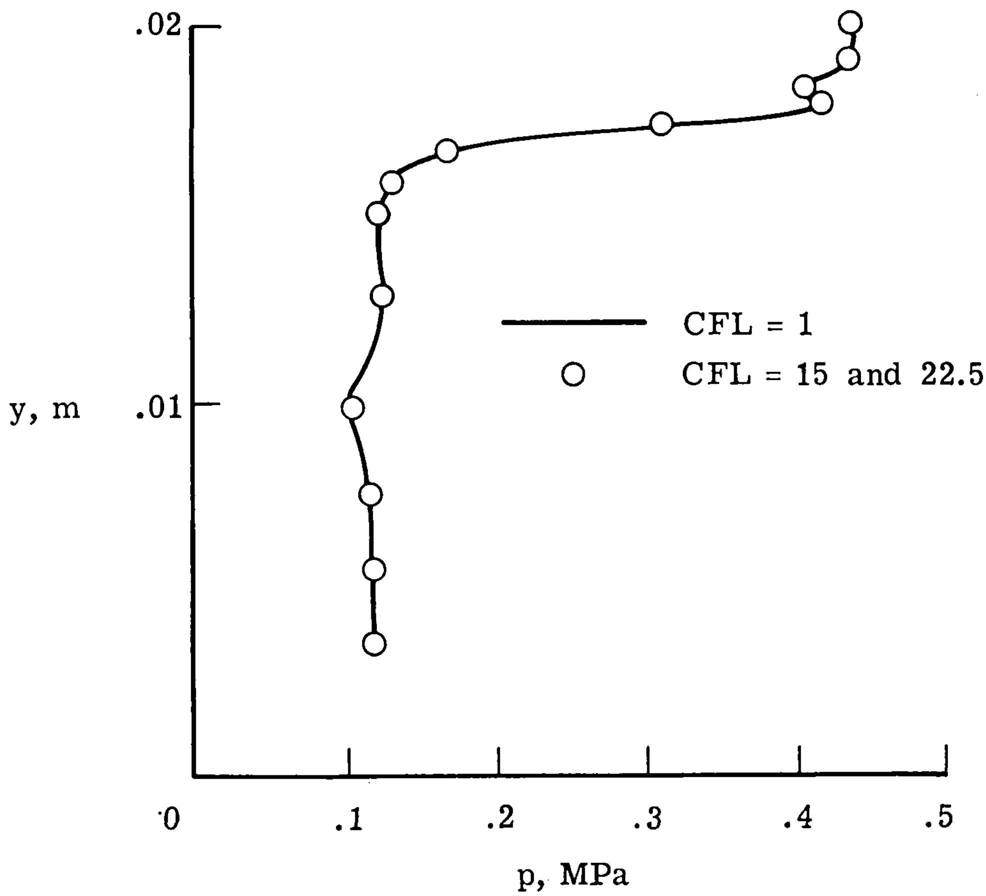


Figure 8.- Pressure profile at x = 0.088 m.

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TP-1934		

4. Title and Subtitle SOME OBSERVATIONS ON A NEW NUMERICAL METHOD FOR SOLVING THE NAVIER-STOKES EQUATIONS	5. Report Date November 1981
	6. Performing Organization Code 505-32-93-01
7. Author(s) Ajay Kumar	8. Performing Organization Report No. L-14746
	10. Work Unit No.
9. Performing Organization Name and Address NASA Langley Research Center Hampton, VA 23665	11. Contract or Grant No.
	13. Type of Report and Period Covered Technical Paper
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546	14. Sponsoring Agency Code

15. Supplementary Notes

16. Abstract

A new method for solving Navier-Stokes equations has been studied. The method is an explicit-implicit technique and is much less complex than other implicit methods. It is used to solve a complex, two-dimensional, steady-state, supersonic-flow problem. A discussion is given of the computational efficiency of the new method and of the quality of the solution obtained from it at high Courant-Friedrich-Lewy (CFL) numbers. Modifications are discussed and certain observations are made about the method which may be helpful in using it successfully.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Finite-difference methods Explicit-implicit method Supersonic flow Navier-Stokes equations	Unclassified - Unlimited Subject Category 02

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	17	A02

For sale by the National Technical Information Service, Springfield, Virginia 22161

NASA-Langley, 1981

National Aeronautics and
Space Administration

Washington, D.C.
20546

Official Business
Penalty for Private Use, $300

SPECIAL FOURTH CLASS MAIL
BOOK

Postage and Fees Paid
National Aeronautics and
Space Administration
NASA-451

U.S.MAIL

NASA

POSTMASTER: If Undeliverable (Section 158
Postal Manual) Do Not Return