The person charging this material is responsible for its return to the library from which it was withdrawn on or before the Latest Date stamped below.

Theft, mutilation, and underlining of books are reasons for disciplinary action and may result in dismissal from the University.

UNIVERSITY OF ILLINOIS LIBRARY AT URBANA-CHAMPAIGN
EVALUATION OF NUMERICAL VISCOSITY EFFECTS IN TRANSONIC FLOW CALCULATIONS

By
S. Rajan

January 1973
EVALUATION OF NUMERICAL VISCOSITY EFFECTS IN TRANSONIC FLOW CALCULATIONS

By

S. Rajan

Center for Advanced Computation
University of Illinois at Urbana-Champaign
Urbana, Illinois 61801

January 1973

This work was supported in part by the Advanced Research Projects Agency of the Department of Defense and was monitored by the U. S. Army Research Office-Durham under Contract No. DAHC04-72-C-0001.
ABSTRACT

The inviscid transonic compressible flow around a circular cylinder is calculated using a time dependent formulation that yields the steady state solution as its asymptotic long time limit. The conservation equations in polar co-ordinates are solved in finite difference form using the explicit, two-step variation of the Lax-Wendroff scheme. The total numerical viscosity of the difference approximation is varied by altering the value of an explicit artificial viscosity parameter incorporated in the finite difference scheme. The far field boundary condition is satisfied exactly by using an inverse transformation in this region. Two different finite difference formulations of the cylinder wall boundary condition are studied in the presence of different amounts of artificial viscosity. The influence of numerical viscosity as it interacts with aerodynamic and numerical criteria is discussed. In transonic flow regimes, where the shock structure is weak, artificial viscosity effects are shown to influence both the strength and location of the shock. At higher Mach numbers, results show that a higher threshold value of the artificial viscosity is required. The importance of choosing an optimum value for the viscosity parameter consistent with accuracy and stability requirements is discussed.
Nomenclature

\( p \) \hspace{1cm} \text{pressure}

\( u, v \) \hspace{1cm} \text{radial and tangential components of velocity}

\( u_\infty \) \hspace{1cm} \text{free stream velocity}

\( M_\infty \) \hspace{1cm} \text{free stream Mach number}

\( r, \theta \) \hspace{1cm} \text{radial and tangential co-ordinates}

\( t \) \hspace{1cm} \text{time}

\( \phi \) \hspace{1cm} \text{artificial viscosity parameter}

\( s \) \hspace{1cm} \text{entropy}

\( r_0 \) \hspace{1cm} \text{reference radius}

\( \Delta R, \Delta \theta \) \hspace{1cm} \text{finite difference increments in radial and tangential directions}

\( \Delta t \) \hspace{1cm} \text{incremented time step}

Subscripts

\( \infty \) \hspace{1cm} \text{free stream condition}

\( 0 \) \hspace{1cm} \text{values on the cylinder body surface}
TABLE OF CONTENTS

| I.  | INTRODUCTION .................................................. 1 |
| II. | THE PROBLEM................................................... 5 |
| III.| THE GOVERNING EQUATIONS..................................... 7 |
| IV. | THE FINITE DIFFERENCE APPROXIMATION....................... 9 |
| V.  | THE INITIAL AND BOUNDARY CONDITIONS....................... 12 |
| VI. | METHOD OF CALCULATION...................................... 14 |
| VII.| DISCUSSION OF RESULTS....................................... 16 |
| VIII.| CONCLUSIONS................................................ 20 |

REFERENCES .................................................. 21

FIGURES .................................................... 22
1. INTRODUCTION

Numerical solutions of flow problems involving discontinuities invariably employ some form of artificial viscosity to provide smooth and stable solutions. This artificial viscosity which is a purely numerical phenomenon, has an effect on the computed solution similar to a physical molecular viscosity, namely it smooths out the discontinuity over a small number of mesh intervals, which is still a small fraction of the total number of meshes in the computational space. An alternative to employing numerical, artificial viscosity in computing flows involving discontinuities such as a shock wave, contact surface, a slip stream or a vortex sheet, etc., is to locate the discontinuity exactly in a space-time domain and apply the actual jump conditions that apply at the discontinuity. However in all but the simplest of cases, this latter procedure is quite difficult and more often than not unfeasible, in a computation of any length. Thus one is obliged to resort to the introduction of an artificial viscosity into either the governing differential equations or the approximating finite difference equations. This expedient now circumvents the necessity of applying the exact jump conditions at the transition boundary, and enables the computation to proceed in the same manner in the region of discontinuity as in other smooth parts of the flow field.

Finite difference schemes introduce a truncation viscosity which is a natural consequence of approximating a piecewise continuous and single valued function with the aid of known values at discrete points. This truncation viscosity, while introducing errors
into the computation is nonetheless necessary to provide computational stability. In addition it oftentimes becomes necessary to add an explicit artificial viscosity term to provide stability.

In the calculation of shock discontinuities by Von Neumann and Richtmyer [1] this explicit artificial viscosity is represented as an additional quadratic term in the momentum and energy conservation equations. An artificial viscosity term which is linear in the velocity gradient, instead of quadratic as in [1] has been employed by Landshoff [2].

Another mode of introducing an explicit artificial viscosity into the computation is by an added term in the finite difference approximation of the governing partial difference equations as in the computations of Kentzer [3] and Magnus and Yoshihara [4]. This explicit artificial viscosity in the difference approximation is oftentimes required to provide computational stability although a linear analysis of the governing equations shows that the difference scheme without the artificial viscosity should be stable [5]. The effect of this artificial viscosity is to attenuate the short wave length components of the solution as the computation proceeds. To do this the pseudo-viscosity in the difference approximation should be dissipative.

It has been shown [5] that for fluid flows without any discontinuities the addition of artificial viscosity terms does not alter the stability of the difference equations. In practice, one assumes the same to be true for piecewise continuous flows. Should the computation prove unstable even in the presence of an artificial viscosity term it is fairly common to increase the amount of added
artificial viscosity until stability is assured. While it is generally known that this increased added artificial viscosity renders the computation more inaccurate, no attempt has been made to study systematically its interaction with the aerodynamic flow variables. In a series of numerical experiments Taylor, Mdefo and Masson [6] have studied the effect of the truncation viscosity on a rarefaction fan, a contact discontinuity and a shock wave. Their purpose was to compare the five difference schemes which they tested in regard to their suitability for calculating flows containing these types of discontinuities. Cameron [7] has shown that introducing an artificial viscosity term into the differential equation after the manner of Von Neumann and Richtmyer [1] or Landshoff [2] also introduces errors into the computation if the shock discontinuity encounters a change in material or mesh size. He has shown that the errors can be substantially reduced by a suitable choice of mesh for the second material or by modifying the definition of the artificial viscosity term.

Very little effort has been expended in actually evaluating the effects of artificial viscosity as it interacts with the aerodynamic flow variables, since the numerical viscosity has been looked upon as a necessary evil in the process of obtaining a stable solution. Especially in transonic flow regimes where the shock structure is weak and easily modified, the pseudo-viscosity effects take on added significance. It now becomes necessary to distinguish fluid dynamic effects from numerical effects. The present investigation was therefore undertaken to understand and evaluate the effects of artificial viscosity in a transonic flow situation. An evaluation of its inter-
action with other aerodynamic and numerical criteria in an actual physical situation serves as a guideline for future calculations involving artificial viscosity. Accordingly a purely mathematical approach has been abandoned in favor of a numerical experiment approach.
II. THE PROBLEM

The steady state transonic flow around a circular cylinder is computed using a time dependent finite difference formulation. An artificial viscosity term is introduced into the finite difference approximation of the governing differential equations, and its interaction with other aerodynamic and numerical criteria is studied. The total artificial viscosity is comprised of the implicit truncation viscosity plus the explicit numerical viscosity added to the finite difference formulation. The truncation viscosity is a basic property of the particular finite difference scheme used and cannot be changed without altering the difference scheme itself. However, the explicit artificial viscosity can be changed and this is the parameter that is varied in the present investigation. The well known two-step, second order accurate Richtmyer variation of the Lax-Wendroff explicit finite difference scheme is used.

In computing the inviscid compressible flow around a circular cylinder it is convenient to employ polar co-ordinates. The initial-boundary value problem yields the steady state flow field as the long time limit of the temporal formulation. Accordingly the solution marches forward in time until the change in the flow variables with time is less than a small specified amount. This is the condition used to approximate the steady state.

The computational mesh layout is shown in Figure 1. The discretised values of the flow variables $p$, $u$, $v$ and $\rho$ are computed at each mesh point. The mesh size is varied to secure greater accuracy in regions of steep gradients in the flow variables, such
as for example near the forward stagnation point. A coarser mesh is employed in the far field region away from the cylinder surface, as the change in the flow variables in this region is small. The free stream Mach number of the flow is varied to examine the influence of the artificial viscosity on the flow in the transonic regime. The effect of different finite difference formulations of the wall boundary condition in the presence of artificial viscosity is studied. The results are compared with those obtained by Holt and Masson [8] by the method of integral relations.
III. THE GOVERNING EQUATIONS

The partial differential equations governing the inviscid compressible flow around a circular cylinder are the conservation laws for mass, momentum and energy expressed in polar co-ordinates, and can be expressed as the following.

Conservation of mass:

\[
\frac{\partial p}{\partial t} + u \frac{\partial p}{\partial r} + \frac{v}{r} \frac{\partial p}{\partial \theta} + \rho \frac{\partial u}{\partial r} + \frac{\partial v}{\partial \theta} + \frac{\partial u}{r} = 0
\]  

Conservation of momentum:

r direction:

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial r} + \frac{v}{r} \frac{\partial u}{\partial \theta} = \frac{v^2}{r} + \frac{1}{\rho} \frac{\partial p}{\partial r} = 0
\]  

θ direction:

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial r} + \frac{v}{r} \frac{\partial v}{\partial \theta} + \frac{1}{\rho} \frac{\partial p}{\partial \theta} = 0
\]  

Conservation of energy:

\[
\frac{\partial e}{\partial t} + u \frac{\partial e}{\partial r} + \frac{v}{r} \frac{\partial e}{\partial \theta} = 0
\]  

The equation of state:

\[
p = \rho RT.
\]  

The physical variables are normalized, using

\[
\sqrt{\frac{p_\infty}{\rho_\infty}}, \quad \frac{r_0}{\sqrt{p_\infty/\rho_\infty}} \quad \text{and} \quad \frac{p}{\rho_\infty}
\]

as reference values for velocity, time and temperature respectively.

Also the normalized entropy is given by

\[
s = \ln \left(\frac{\rho}{\rho_\infty}\right) - \gamma \ln \left(\frac{p}{p_\infty}\right)
\]
The conservation equations (1) - (4) can also be written in matrix vector form as

\[ F_t + AF_r + BF_{\theta} + C = 0 \]  

(7)

where,

\[
F = \begin{bmatrix} P \\ u \\ v \\ s \end{bmatrix}, \quad A = \begin{bmatrix} u & \gamma & 0 & 0 \\ p/\rho & u & 0 & 0 \\ 0 & 0 & u & 0 \\ 0 & 0 & 0 & u \end{bmatrix}, \quad B = \frac{1}{r} \begin{bmatrix} v & 0 & \gamma & 0 \\ 0 & v & 0 & 0 \\ p/\rho & 0 & v & 0 \\ 0 & 0 & 0 & v \end{bmatrix}
\]

\[
C = \begin{bmatrix} \gamma u \\ -v^2 \\ uv \\ 0 \end{bmatrix}
\]

Here \( P = \ln \left( \frac{\rho}{\rho_{\infty}} \right) \)  

(8)
IV. THE FINITE DIFFERENCE APPROXIMATION

Finite difference equations corresponding to the governing differential equations are employed at the mesh points in the computational space. The solution progresses from known specified values at an initial time level to computed values at the next time level, separated by a time increment $\Delta t$. A linear stability analysis of the particular variation of the two step Lax-Wendroff scheme used in the present analysis indicates that the scheme is stable if the Courant-Friedrichs-Lewy criterion is satisfied. The value of the time step $\Delta t$ used is therefore given by

$$\Delta t = \frac{\Delta r}{1.5 \left( |u| + |v| \right)}$$  \hspace{1cm} (9)

where a safety factor of 1.5 is used.

Values of the physical variables $p$, $u$, $v$ and $\rho$ are computed in the first step at time level $(n+\frac{1}{2}) \Delta t$. For the differential equation

$$F_t + AF_x + BF_y + C = 0$$

the intermediate values are given by

$$F^n_{\lambda,m} = \frac{1}{4} \left[ F^n_{\lambda+1,m} + F^n_{\lambda-1,m} + F^n_{\lambda,m+1} + F^n_{\lambda,m-1} \right]$$

$$- \frac{\Delta t}{2 \Delta r} F^n_{\lambda+\frac{1}{2},m} \left[ F^n_{\lambda+1,m} - F^n_{\lambda+\frac{1}{2},m} \right] - \frac{\Delta t}{2 \Delta \phi} F^n_{\lambda,m}$$

$$- \frac{\Delta t}{2} C^n_{\lambda,m}$$

$$\left[ F^n_{\lambda,m+\frac{1}{2}} - F^n_{\lambda,m-\frac{1}{2}} \right]$$

$$- \frac{\Delta t}{2} C^n_{\lambda,m}$$

Final values at the next time level $(n+1) \Delta t$ are computed from the difference equation
A nine point lattice is involved in the computation of the variables at one point at the next higher time level. The parameter \( \phi \) is introduced into the difference scheme at the second step. It provides additional numerical viscosity as it assumes values between 0 and 1, and provides a means of varying the total numerical viscosity of the computation. The generation of an artificial viscosity by the parameter \( \phi \) is illustrated using the simplified equation

\[
F^{n+1}_r + A F^r = 0
\]

in one dimension, without any loss of generality. Let \( F^n_\ell \) denote the value of \( F \) at a mesh point \((\ell \Delta r, n \Delta t)\) in the computational space. Assuming \( A \) is constant, we have

\[
\frac{n+1}{2} = \frac{1}{2} \left[ F^n_{\ell+2} + F^n_{\ell} \right] - A \frac{\Delta t}{2 \Delta r} \left[ F^n_{\ell+2} - F^n_{\ell} \right] \tag{13}
\]

\[
\frac{n+1}{2} = \frac{1}{2} \left[ F^n_{\ell} + F^n_{\ell-2} \right] - A \frac{\Delta t}{2 \Delta r} \left[ F^n_{\ell} - F^n_{\ell-2} \right] \tag{14}
\]

\[
F^{n+1}_\ell = (1 - \phi) F^n_\ell + \phi \left[ F^n_{\ell+1} + F^n_{\ell-1} \right] - A \frac{\Delta t}{2 \Delta r} \left[ F^n_{\ell+1} - F^n_{\ell-1} \right] \tag{15}
\]
Let \( D_p \) denote equation (15) equated to zero. Assuming that the function \( F \) is continuous at \(( \Delta \rho, \Delta t)\) and has continuous derivatives, we obtain values of \( F \) in the vicinity of \(( \Delta \rho, \Delta t)\) by Taylor series expansions around that point. Then

\[
\frac{\partial F}{\partial t} + A \frac{\partial F}{\partial r} - D_p = \phi \Delta \rho \frac{\partial^2 F}{\partial r^2} + \Delta t \left[ A \frac{\partial^2 F}{\partial r^2} - \frac{1}{2} \frac{\partial^2 F}{\partial t^2} \right] \tag{16}
\]

We thus have two additional terms on the right hand side of (16) which represent a diffusion effect. The first diffusion term contains a second order space derivative with a viscosity coefficient proportional to \( \phi \). The second component is also comprised of second order derivatives, but with a coefficient of viscosity proportional to \( \Delta t \). Thus it is seen that in any computation where all other factors remain the same, the variation of \( \phi \) enables us to enhance or decrease the numerical viscosity effects on the flow field. It is also seen that larger values of \( \phi \) imply larger truncation errors.

Since transonic flows contain only a weak shock structure the influence of a large artificial viscosity parameter can result in serious errors in the computed solution. Also transonic flows can be considered as the limit situation of smooth flows. Since the introduction of an artificial viscosity into finite difference calculation of smooth flows has been shown [5] to leave the stability of the scheme unaltered, the same is assumed to hold true in transonic flow calculations.
V. THE INITIAL AND BOUNDARY CONDITIONS

The steady state solution computed as the long time limit of the time dependent formulation is independent of the initial conditions. Accordingly, the initial values of $p$, $u$, $v$ and $\rho$ that are specified at each mesh point in the computational field, are the free stream values $p_\infty$, $u_\infty$, $v_\infty$ and $\rho_\infty$. On the cylinder body surface the radius $r_0$ is taken as unity. At time $t = 0$, at the start of the computation, the radial component $u_0$ of the cylinder surface velocity is impulsively reduced to zero.

The flow is assumed to be symmetrical about the $\theta = 0$ and $\theta = \pi$ boundaries (Figure 1). Therefore, for mesh points situated at $\pm \Delta \theta$ from these boundaries we have

\begin{align*}
  p(\Delta \theta) &= p(-\Delta \theta) & p(\pi - \Delta \theta) &= p(\pi + \Delta \theta) \\
  u(\Delta \theta) &= u(-\Delta \theta) & u(\pi - \Delta \theta) &= u(\pi + \Delta \theta) \\
  v(\Delta \theta) &= -v(-\Delta \theta) & v(\pi - \Delta \theta) &= -v(\pi + \Delta \theta) \\
  \rho(\Delta \theta) &= \rho(\Delta \theta) & \rho(\pi - \Delta \theta) &= \rho(\pi + \Delta \theta)
\end{align*}

for all $r$. Also $v(0) = v(\pi) = 0$.

On the cylinder body surface, the radial velocity is zero.

Two finite difference formulations on the wall boundary condition were experimented with to study the effect of this numerical criterion on the computed flow field in the presence of various amounts of artificial viscosity. In one case a centered scheme was employed to compute the dependent variables at time $t + \Delta t$, and in the other a one-sided finite difference formulation was used. For the two step Lax-Wendroff scheme, a fictitious row of reflection points below the body surface are required if a centered scheme is used. Therefore, in this case we have
\[ p(r_0 + \Delta r) = p(r_0 - \Delta r) \]
\[ u(r_0 + \Delta r) = u(r_0 - \Delta r) \]
\[ v(r_0 + \Delta r) = v(r_0 - \Delta r) \]
\[ \rho(r_0 + \Delta r) = \rho(r_0 - \Delta r) \]

for all \( \theta \).

Since it would be necessary to specify a large number of mesh points to be able to compute deep into the far field, an inverse transformation is used in this region away from the cylinder surface. The circular computational space external to a given radius is mapped by an inverse transformation such that infinity in the physical space becomes the origin in the transformed space. The transformed conservation equations (1) - (4) are applied in finite difference form in this far field calculation. The conditions at infinity are applied on a circle of small radius in the transformed space. Sufficient overlapping of points in the physical and transformed spaces ensures continuity between the two regions. In this way no approximations enter into the calculation of the transonic flow field in fulfilling the far field boundary conditions.
VI. METHOD OF CALCULATION

Initial values of the dependent variables equal to the free stream values are specified at all points in the computational space. The points just above and below the $\theta = 0$ and $\theta = \pi$ lines are made to correspond to the reflection boundary conditions specified. The radial velocity of all points on the cylinder body surface is set to equal zero. Starting with the initial values at time $t = 0$, the dependent variables $p$, $u$, $v$ and $\rho$ are calculated at time $t + \Delta t$ on the circumferential row of mesh points corresponding to the cylinder surface; i.e., $r = r_0$. For the two step explicit finite difference scheme used, intermediate values at time level $\frac{\Delta t}{2}$ are first computed before final values at time level $\Delta t$ can be obtained. A centered finite difference formulation of the wall boundary condition ($u = 0$), and also a one-sided formulation have been used. On obtaining the dependent variables at time level $\Delta t$, it is usually found that the computed value of the radial velocity $u$ at the wall is not zero. A correction is then applied in the form of an impulsive wave to force the radial velocity of the wall to zero.

The radial increment $\Delta r$ is increased and the dependent variables $p$, $u$, $v$ and $\rho$ are computed at time level $\Delta t$ for the row of circumferential mesh points situated at radius $r + \Delta r$. Computation thus proceeds outwards with increasing radius until it is time to begin the far field calculations extending to a specified large radius. Dependent variables at mesh points in the far field are computed from the finite difference analogue of the transformed governing equations. Since an inverse transformation has been used, the radial increment $\Delta r'$ in this region is negative.
proceeds in essentially the same manner in the transformed space as in the physical space, with values of \( p, u, v \) and \( \rho \) being computed on a row of circumferential mesh points at time level \( \Delta t \), until a specified small radius in the transformed space is reached. Here the free stream conditions \( p_\infty, u_\infty, v_\infty \) and \( \rho_\infty \) are held to remain constant with time, so that this last boundary row of circumferential points is not computed.

Using these computed values at time level \( \Delta t \) as initial values, the computational field is traversed again beginning at the cylinder surface \( r = r_0 \), to compute the dependent variables at all the mesh points at time level \( 2\Delta t \). This marching procedure in time is continued until the steady state is reached.

For an initially specified free stream Mach number in the transonic range and viscosity parameter \( \phi \), the shock forms naturally on the rearward side of the cylinder. Computation of the transonic flow field is made for different values of the free stream Mach number, for different values of the viscosity parameter \( \phi \) and for two different finite difference formulations of the wall boundary condition.
VII. DISCUSSION OF RESULTS

Figures 2, 3 and 4 show the computed values of cylinder surface velocity, surface pressure and density respectively for a free stream Mach number of 0.425. The artificial viscosity parameter \( \phi \) is varied from 0.05 to 0.175. The computed values of the pressure jumps at the recompression shock decrease with an increase in the amount of artificial viscosity. Therefore not only does the artificial viscosity smear out the shock over a larger number of meshes, it affects the shock strength, as can be readily observed from Figures 2, 3 and 4. This in turn strongly influences the pressure coefficient. It is seen that for the supercritical Mach number of 0.425 and a value of \( \phi = 0.05 \), there is still some velocity fluctuation persisting in the flow on the downstream side (Figure 2). Increasing the artificial viscosity, not only reduces the strength of the recompression shock but also influences the shock location. A variation of \( \phi \) from 0.05 to 0.175 at a free stream Mach number of 0.425, moves the recompression shock upstream about 10 degrees on the cylinder arc.

From the results it is seen that the optimum value of the added explicit artificial viscosity as expressed by the parameter \( \phi \) lies somewhere between 0.05 and 0.075. At values of \( \phi \) below 0.05, oscillations in the velocity, pressure and density profiles became stronger with time, so that it became difficult to obtain a steady state solution. It is also seen from Figures 2, 3 and 4 that at the high value of \( \phi = 0.175 \), the recompression shock has been smeared out to such an extent as to make it almost indistinguishable.
Figure 5 shows the cylinder wall pressure for a super-critical free stream Mach number of 0.45, with values of $\phi$ ranging from 0.075 to 0.175. Even at a value of $\phi = 0.075$ the oscillations on the downstream side of the shock still persist, while below $\phi = 0.075$ it was difficult to obtain stable steady state solutions. It is possible to obtain stable, smooth solutions at $\phi = 0.1$ as seen from Figure 7. Thus for a free stream Mach number of 0.15, the optimum value of $\phi$ to obtain smooth stable solutions lies between 0.05 and 0.075, while at a free stream Mach number of 0.45, the optimum value of $\phi$ lies between 0.075 and 0.1. Therefore we can conclude that there is a threshold optimum value of $\phi$ for a given particular value of the free stream Mach number, and that this threshold optimum value increases with higher free stream Mach numbers.

Comparing Figures 3 and 5, it is seen that at higher Mach numbers, the artificial viscosity does not influence the shock location as strongly as at lower Mach numbers.

Figure 6 shows the influence of the finite difference formulation of the wall boundary condition on the computed cylinder wall pressure. A value of $\phi = 0.10$ ensures a smooth solution at a Mach number of 0.45. It is seen from Figure 6 and also from Figure 7 that the shock location is transported a little downstream with a one-sided scheme at the wall. The peak pressure is higher with a one-sided scheme.

Figure 7 compares the cylinder wall velocity profiles obtained from the present method with that obtained by Holt and Masson [8] by the method of integral relations. It is seen that the
one-sided finite difference formulation of the wall boundary condition gives a closer approximation to the result of [8]. While the velocity jump is essentially the same with a one-sided scheme and a centered scheme at the wall, the peak velocity is higher with a one-sided scheme. The curve obtained from the present calculation with $\phi = 0.075$ agrees more closely with that of reference [8] on the upstream side of the recompression shock that on the downstream side.

Figure 8 shows the number of iterations required for convergence for different values of $\phi$ at free stream Mach numbers of 0.425 and 0.45. One possible explanation for the fact that at a higher Mach number the number of iterations decreases with an increase in artificial viscosity is that the shock is now steeper. This reduces the severity of the convergence criterion in a point by point application. The convergence criterion employed was that the computed solution at each point should not differ by a total of one percent for ten successive time iterations.

The present work differs from the work of Emery [9] in that it applies to the transonic regime and the method of computation is entirely different. Emery's calculations of a Mach 3 shock with different finite difference schemes containing different amounts of artificial viscosity, does not study the influence of artificial viscosity as such. As compared with the work of Kentzer [3], the present method of computing the transonic flow around a circular cylinder is shown to yield meaningful results with low enough values of the artificial viscosity, even when the free stream Mach number is fairly high in the supercritical range. The time dependent
formulation of computing the transonic flow discussed herein can be readily extended to include more complicated geometries such as airfoil sections.
VIII. CONCLUSIONS

The method of computing the steady state transonic flow around a circular cylinder using a time dependent formulation is shown to yield reliable results even at low values of explicit artificial viscosity. For a given free stream Mach number, there is an optimum value of artificial viscosity which enables a stable solution to be realized. This optimum value increases with an increase of free stream Mach number. Increasing the added artificial viscosity beyond this optimum value merely increases the inaccuracies consequent to the introduction of higher truncation errors. Results obtained by the present method compare favorably with those obtained by other investigators. The study shows that in the transonic regime the artificial numerical viscosity influences both the location and strength of the recompression shock, thus altering the pressure coefficient. This influence is seen to be mitigated at higher Mach numbers in the transonic range.

The method described herein for transonic flow calculations can be readily applied to more complicated geometries such as ellipsoids of revolution and airfoils.
REFERENCES

[1] VonNeumann, J., and Richtmyer, R. D., "A Method for the Numerical Calculation of Hydrodynamic Shocks", J. Applied Physics, Vol. 21, March 1950.

[2] Landshoff, R., Report No. 1930, Los Alamos Scientific Laboratory, 1955.

[3] Kentzer, C. P., "Computations of Time Dependent Flows on an Infinite Domain", A.I.A.A. Paper No. 70-45, 1970.

[4] Magnus, R. and Yoshihara, H., "Inviscid Transonic Flow over Airfoils", A.I.A.A. Journal, Vol. 8, No. 12, December 1970.

[5] Richtmyer, R. D. and Morton, K. W., Difference Methods for Initial-Value Problems, InterScience Publishers, N. Y., 1967.

[6] Taylor, T. D., Ndefo, E., and Masson, B. S., "A Study of Numerical Methods for Solving Viscous and Inviscid Flow Problems", J. of Computational Physics, 9, pp. 99-119, 1972.

[7] Cameron, I. G., "An Analysis of Errors Caused by Artificial Viscosity Terms to Represent Steady-State Shock Waves", Journal of Computational Physics, 1, 1966.

[8] Holt, H. and Masson, B. S., "The Calculation of High Subsonic Flow past Bodies by the Method of Integral Relations", Proc. Second International Conference on Fluid Dynamics, Berkeley, Springer-Verlag, N. Y., 1971.

[9] Emery, A. F., "An Evaluation of Several Differencing Methods for Inviscid Fluid Flow Problems", Journal of Computational Physics, 2, 1968.
Figure 1. Computational Mesh System
Figure 2. Influence of artificial viscosity on cylinder wall velocity at $M_\infty = 0.425$. 

CYLINDER SURFACE VELOCITY $v_0/a$
Figure 3. Influence of artificial viscosity on shock strength and location for a free stream Mach number of 0.425.
Figure 4. Influence of artificial viscosity parameter $\phi$ on cylinder surface density.
Figure 5. Influence of artificial viscosity on cylinder wall pressure at $M_\infty = 0.45$. 
Figure 6. Influence of finite difference formulation of wall boundary condition on cylinder wall pressure for $\phi = 0.10$. 

$M_\infty = 0.45$

$\phi = 0.10$

CENTERED SCHEME

ONE-SIDED SCHEME
Figure 7. Comparison of present results with those of Holt and Masson for different values of \( \phi \) and different formulations of wall boundary condition.
Figure 8. Effect of viscosity parameter on convergence rates at different free stream Mach numbers.
The inviscid transonic compressible flow around a circular cylinder is calculated using a time dependent formulation that yields the steady state solution as its asymptotic long time limit. The conservation equations in polar co-ordinates are solved in finite difference form using the explicit, two-step variation of the Lax-Wendroff scheme. The total numerical viscosity of the difference approximation is varied by altering the value of an explicit artificial viscosity parameter incorporated in the finite difference scheme. The far field boundary condition is satisfied exactly by using an inverse transformation in this region. Two different finite difference formulations of the cylinder wall boundary condition are studied in the presence of different amounts of artificial viscosity. The influence of numerical viscosity as it interacts with aerodynamic and numerical criteria is discussed. In transonic flow regimes, where the shock structure is weak, artificial viscosity effects are shown to influence both the strength and location of the shock. At higher Mach numbers, results show that a higher threshold value of the artificial viscosity is required. The importance of choosing an optimum value for the viscosity parameter consistent with accuracy and stability requirements is discussed.
| KEY WORDS                                      | LINK A | LINK B | LINK C |
|-----------------------------------------------|--------|--------|--------|
| Inviscid transonic compressible flow          |        |        |        |
| circular cylinder                             |        |        |        |
| artificial viscosity                          |        |        |        |
