MILITARY TECHNICAL COLLEGE

CAIRO - EGYPT

Calculation Of One-Dimensional Transonic Flows Using Micro-Computers

M.M.ABDELRAHMAN*, A.O.SHERIF**

ABSTRACT

The use of micro-computers to investigate new solutions, techniques and for training purposes, proved to be a cost effective solution. In this work the potential of using micro-computers to solve one dimensional transonic flow is explored.
A newly developed implicit pseudo-unsteady finite difference method for solving Euler equations is considered. The equations are solved in a converging diverging nozzle. Solutions obtained are compared to the exact solution . Bounds on the different parameters affecting the solution process and time estimates for the algorithms considered are presented.

* Assistant Professor, Dpt. of Aeronautics, Cairo University, Egypt.
** Associate Professor, Dpt. of Aeronautics, Cairo University, Egypt.

INTRODUCTION

Transonic aerodynamics is the focus of strong interest at the present time, It is known that the transonic regime is one of the most efficient flight regimes. However, the analysis of transonic flows and the design problems in such range of flight speeds is more difficult than studying pure subsonic or supersonic flows. The difficulty being primarily associated with the mixed elliptic hyperbolic nature of the governing equations, and to the presence of discontinuities in the field of computation.

At fairly low supersonic speeds, $1<M<1.3$, the shock wave is quite weak and it is reasonable to replace it by a series of isentropic compression waves. This approximation should not be a source of serious errors. The reason is that the entropy generated by the shock wave is proportional to the third power of the shock strength or (M^2-1), where M is the Mach number before the shock. In this range, it is possible to obtain satisfactory results by solving the potential equation instead of solving the complete Euler's equations. However, Euler equations should be used for solving flows with strong shocks and flows with rotational effects.

In an effort to stimulate and encourage the application of computational methods, the Department of Aeronautics, Cairo University has decided upon a plan to explore the utilization of present day micro computers in such field. The plan is concerned with the study of computational methods for one, two and three dimensional flows.

In this paper the finite difference method has been used to solve the transonic one dimensional flow in ducts. The following sections summarise the problem formulation, the discretization of the equations and the results of the numerical experiment.

PROBLEM FORMULATION

For a quasi one dimensional flow in a duct and assuming a constant total enthalpy, the Euler equations may be expressed in the following nondimensional form [1] ;

$$\vec{F}_t + \vec{G} = 0 \qquad (1)$$

where

$$\vec{F} = \{F_0 , F_1\}^T$$

$$\vec{G} = \{G_0 , G_1\}^T$$

$$F_0 = \rho/\rho_{0in} \quad , \quad F_1 = F_0 V/a^*$$

$$G_0 = (F_1 S)_x / S$$

$$G_1 = \{R (F_0 + F_1^2/F_0)\}_x + S_x(F_1^2/F_0)/S$$

$$R = (\gamma+1)/2\gamma$$

ρ is the density of gas ,

V is its velocity ,

a* is the critical speed of sound ,

S is the nondimension cross-sectional area ,

γ is the specific heats ratio ,

$_{0}$in is a subscript denoting the total condition at the intake section ,

t and x are subscripts denoting the differentiation with respect to time and space respectively.

Here (F_0 , G_0) represent the continuity equation and (F_1 , G_1) represent the momentum equation .

The numerical solution of equation (1) is obtained by using a time marching method. The procedure solves the equation in the unsteady form then computes the changes in the physical values over time untill a steady state solution is found [2-3].

Several finite difference schemes have been developed to solve equation (1) in the transonic regime. Some of these methods are based on explicit discretization in time[4-5]. In such methods, the time step used in the computation is restricted by the stability requirements of the scheme. This leads to considerable increase in the computation time. The stability of the scheme is usually expressed in terms of the CFL number;

$$CFL = \frac{\Delta t}{\Delta x} \max(\lambda_1 , \lambda_2)$$

where $\lambda_{1,2}$ are the eigen values of the system of equations (1)

$$\lambda_{1,2} = R \frac{V}{a^*} \{ 1 \pm \sqrt{1+\frac{1}{\gamma R} (\frac{1}{M^2} -1)} \} ,$$

M is the local Mach number.
Other methods use implicit discretization [6-8] . The usually have better stability properties than the explicit methods ; i.e. larger time steps can be used.

In the next section, a new implicit scheme developed by the first auther [9] will be described and adapted to the purpose of this work.

DISCRETE MODEL

The system of equations (1) is discretized in two time levels such that

$$F_i^{n+1} = F_i^n - \Delta t \, G_i^{n+1} + O(\Delta t^2) \qquad (2)$$

where F_i^n is the numerical solution at the point $(t=n\Delta t , x=i\Delta x)$

The nonlinear implicit term G^{n+1} is written as

$$G^{n+1} = G^n + \beta f + O(\overline{\Delta t}^2) \qquad (3)$$

where

$$\beta f = (\partial G/\partial F)^n f + (\partial G/\partial F_x)^n f_x$$

and

$$f = F^{n+1} - F^n$$

The elimination of G^{n+1} between equation (2) and equation (3) leads to

$$\{ I + \Delta t \, \beta \} f = - \Delta t \, G^n \qquad (4)$$

where I is a unit matrix.

Now, centered space discretization is used for the system of equations (4)

$$f_{0i} + \frac{\mu}{S_i}\{ (Sf)_{1i+\frac{1}{2}} - (Sf)_{1i-\frac{1}{2}} \} = -\Delta t G^n_{0i} \qquad (5\text{-a})$$

$$f_{1i} + \mu R\{ (1-v^2)_{i+\frac{1}{2}} f_{0i+\frac{1}{2}} - (1-v^2)_{i-\frac{1}{2}} f_{0i-\frac{1}{2}} \} - S'_i v_i^2 f_{0i}$$
$$+ 2\mu R(v_{i+\frac{1}{2}} f_{1i+\frac{1}{2}} - v_{i-\frac{1}{2}} f_{1i-\frac{1}{2}}) + 2S'_i v_i f_{1i} = -\Delta t G^n_{1i} \qquad (5\text{-b})$$

where

$$\mu = \Delta t/\Delta x \quad , \quad v = V/a^* \quad \text{and} \quad S'_i = \mu(\Delta S/S)_i$$

This system of equations is a block tridiagonal system. In order to use a standard tridiagonal solver, equation (5-a) is substituted into equation (5-b) and the later is solved. The resulting system is defined by

$$- b_i \, f_{1i-1} + c_i \, f_{1i} - d_i \, f_{1i+1} = e_i \qquad (6)$$

where

$$b_i = \mu^2 R\,(1-v^2)_{i-\frac{1}{2}} \frac{S_{i-1}}{S_{i-\frac{1}{2}}} + \mu\, R\, v_{i-\frac{1}{2}} + \frac{1}{2}\mu S'_i\, v_i^2 \frac{S_{i-\frac{1}{2}}}{S_i},$$

$$c_i = 1 + \mu^2 R\,\{(1-v^2)_{i+\frac{1}{2}} \frac{S_i}{S_{i+\frac{1}{2}}} + (1-v^2)_{i-\frac{1}{2}} \frac{S_i}{S_{i-\frac{1}{2}}}\} + 2S'_i v_i$$

$$+ \mu R\,(v_{i+\frac{1}{2}} - v_{i-\frac{1}{2}}) + \frac{1}{2} S'_i v_i^2 (S_{i+\frac{1}{2}} - S_{i-\frac{1}{2}})/S_i$$

$$d_i = \mu^2 R\,(1-v^2)_{i+\frac{1}{2}} \frac{S_{i+1}}{S_{i+\frac{1}{2}}} - \mu R\, v_{i+\frac{1}{2}} - \frac{1}{2}\mu\, S'_i v_i^2 \frac{S_{i+\frac{1}{2}}}{S_i}$$

$$e_i = -\Delta t\, G^n_{1_i} - \tau S'_i\, v_i^2\, G^n_{0_i} + \mu R\{(1-v^2)_{i+\frac{1}{2}} G^n_{0_{i+\frac{1}{2}}} - (1-v^2)_{i-\frac{1}{2}} G^n_{0_{i-\frac{1}{2}}}\}$$

To ensure the existence and uniqueness of the numerical solution of equation (6) at supersonic speeds, a mixed differences must be introduced into the momentum equation. This is introduced by using backward spatial difference in the supersonic zone as described in [1] and [9].

$$F^{n+1}_{1i} = F^n_{1i} - \Delta t\alpha\,\{(1-\varepsilon_+)\, G^{n+1}_{1i} + \varepsilon_- G^{n+1}_{1i-1}\} \qquad (7)$$

where $\varepsilon = 0$ when $M<1$

$\varepsilon = 1$ when $M>1$

The subscripts +,- correspond to the points $(i+\frac{1}{2})$ and $(i-\frac{1}{2})$ respectively.

$\alpha = 1$ every where

$\alpha = 1/2$ at the shock point

The scheme described by equations (5-a) and (7) is unconditionally stable in the sense of Von Neuman [1] ;i.e. it is independent of the time step selected.

RESULTS AND DISCUSSION

The method described in the previous section of this article has been programmed on an HP45 micro computer system using BASIC language. The example considered is the flow in a parabolic Laval nozzle. The cross sectional area is given by

$$S(x) = .01 + .04778\ (x-0.5)^2 \qquad 0<x<1$$

At the outflow, a constant pressure value was imposed that will ensure subsonic flow there. The back pressure imposed resulted in a shock at x=.75 which corresponds to a ratio of the output static pressure to the input total pressure of 0.813 and a Mach number before the shock equal to 1.6.

To start the calculations, the stagnation values for the pressure and the density was prescribed every where with zero velocity.

Fig.1 shows the evolution of the flow, expressed in terms of pressure, with the iteration. The CFL number was 20.
Fig.2 shows the behaviour of the rate of convergence of the iteration as the CFL number is varied. It is quite clear that the rate of convergence increase as the CFL number increased.

For explicit schemes, the upper bound of the CFL number is determined by the stability condition. For the implicit scheme it is determined by the actual computation. In this case the maximum CFL number is 100 . CFL as high as 150 has been obtained using a super main frame (CYBER 170-750).

Fig.3 illustrate the pressure distributions associated with three values of CFL numbers used in computations after 100 iterations.

The following table summarizes the actual experience gained from such experiment

CFL No	No of time steps	Reduction in Residual	$-\log_{10}$(Res.) for (100 it.)
5	100	1	2.8
20	67	6	10.1
50	47	6	11.4

Note that more than 100 iterations are needed to reach the steady state solution for CFL number as low as 5.
The mesh used has 51 points and the computer time taken was 0.4 second/iteration/point.

In conclusion, the results obtained are in good agreement with the exact solution. It indicated that micro computers are suitable for such kind of computations. However the 0.4 sec./point/iteration is a very high figure. But in the authors opinion is due to the slowness of the HP system. A typical figure for two-dimensional transonic computation 10 was 4 minutes per iteration in a mesh of 33X17 points.

The authors beleive that this figure will be lower on modern systems such as IBM (PC or XT) with mathematical coprocessor. A typical figure for 2-D incompressible flow problem was approximately .01 sec./point/iteration. The computation was done using double precision arithmetic, the program was in FORTRAN on an IBM PC with 640 KB.

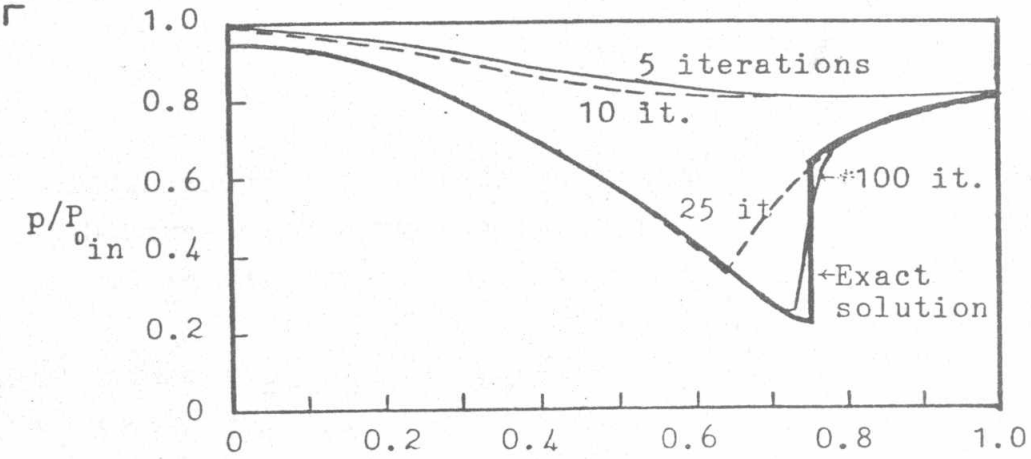

Fig.1. Evolution of the pressure ratio with iterations.

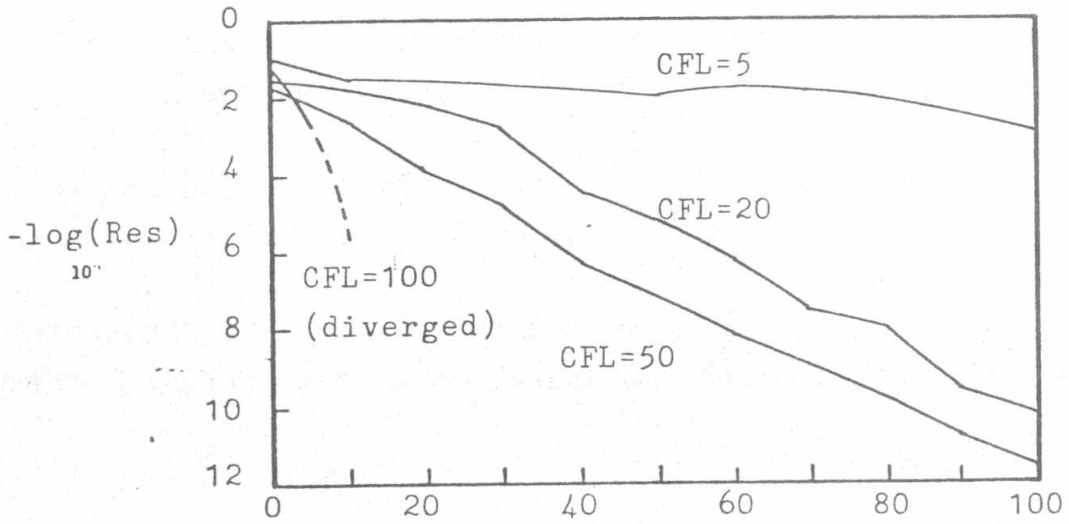

Fig.2. Convergence histories for different CFL numbers.

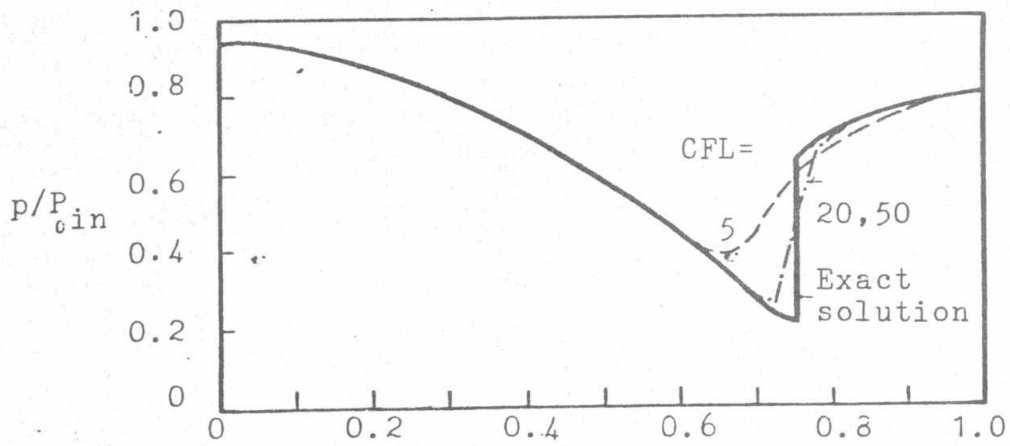

Fig.3. Distribution of the pressure ratio after 100 iterations for different CFL numbers.

REFERENCES

1. Abdelrahman, M.M., "Methode de Resolution des Ecoulements Transsoniques en vue de l'Application aux Compresseurs Centrifuges", D.Sc. Thesis. University of Pierre &Marie Curie, Paris 6, France (1984).
2. Magnus, R. and Yoshihara, H.,"Steady Inviscid Transonic Flows over planar airfoils. A research for a simplified procedure", NASA-CR N°186 (1972).
3. Viviand, H. and Veuillot, J.P., "Méthodes Pseudo-Instationnaires pour le calcul d'écoulements Transsonique", ONERA T.P. N°4, France (1978).
4. MacCormack, R.W., "Numerical solution of the interaction of a shock wave with a laminar boundary layer", Lecture Notes in Physics, vol.8,Springer-Verlag, (1971).
5. Lerat, A., "Sur le calcul des Solutions faibles des systèmes Hyperboliques de lois de Conservation à l'aide de Schémas aux Différences", ONERA P N° 1, France (1981) .
6. Beam, R.M. and Warming, R.F., "An Implicit Factored Scheme for the Compressible Navier-Stokes equations", AIAA Journal , vol. 16 , (1978) .
7. Moretti, G., "The λ scheme", Computer and Fluids, vol.7,191-205,Sept. (1979) .
8. Lerat, A., "Implicit Methods of Second-Order Accuracy for the Euler Equations", AIAA paper 83-1925 (1983) .
9. Abdelrahman, M.M., "Contribution to the Calculation of Transonic Flows using an Implicit Pseudo-Unsteady Method" , Rech. Aérosp., N° 1981-6 (1981) .
10. Sherif, A.O., "A Modified Algorithm for computing Plane Transonic Flows using the Stream Function Equation", 19th. Int. Conf. for Statistics, Computer Science and Operations Research ISSR, Cairo (1985).