Arch. Mech., 42, 2, pp. 183—195, Warszawa 1990

Some methods for improving the convergence of iterative solutions of transonic flow equations

R. TRĘBIŃSKI (WARSZAWA)

SOME NEW METHODS have been devised for improving the convergence of iterative methods of solving the equations of transonic flow. They concern the algorithm of construction of a solution of the equations of steady flow based on explicit difference schemes approximating the equations of unsteady flow. The modifications which are proposed for the standard algorithm consist in the application of a multigrid procedure, a change of the sequence of operations and the use of a reconstruction procedure of the grid different from the standard procedure. The efficacy of the methods proposed is analysed by studying the results of solution of a test problem.

W pracy zaproponowano pewne nowe sposoby przyspieszania zbieżności iteracyjnych metod rozwiązania równań przepływu transdźwiękowego. Odnoszą się one do algorytmu konstrukcji rozwiązania równań przepływu ustalonego, opartego na jawnych schematach różnicowych, aproksymujących równania niestacjonarnego przepływu. Proponowane są następujące modyfikacje standardowego algorytmu: zastosowanie procedury wielosiatkowej, zmiana sekwencji obliczeń w danej iteracji oraz odmienna od standardowej procedura przebudowy siatki. Efektywność proponowanych sposobów przyspieszania zbieżności przeanalizowano na podstawie wyników rozwiązania testowego problemu.

В работе предложены некоторые новые способы ускорения сходимости итерационных методов решения уравнений трансзвукового течения. Они относятся к алгоритму построения решения уравнений установившегося течения, опирающегося на явных разностных схемах, аппроксимирующих уравнения нестационарного течения. Предлагаются следующие модификации стандартного алгоритма: применение многосеточной процедуры, изменение последовательности расчетов в данной итерации и отличающаяся от стандартной процедура перестройки сетки. Эффективность предлагаемых способов ускорения сходимости проанализирована на основе результатов решения тестовой задачи.

1. Introduction

IF WE CONSTRUCT solutions of external and internal problems of steady compressible flow, we are often concerned with a case in which regions of subsonic motion of the fluid are adjacent to those of supersonic flow, from which they are separated by shock-wave fronts or Mach lines. A flow in which regions of both types occur simultaneously is termed transonic flow. The positions of the boundaries between particular regions are not known beforehand and are determined in the course of constructing the solution of the flow equations. The character of the equations describing subsonic and supersonic flows being different, the construction of a solution of the equations of flow comes up against considerable difficulties of mathematical nature. In subsonic regions a problem of transonic flow constitutes a Cauchy problem for an elliptic set of equations [1]. This is an incorrectly stated problem, the solution being very sensitive to the boundary conditions, which may

be a source of considerable difficulties if it is attempted to construct a solution of the equations of transonic flow by difference methods. There are no such difficulties in regions of supersonic flow, which is described by hyperbolic equations, for which the Cauchy problem is correctly stated. It follows that the most natural and direct way for avoiding the difficulties connected with the mixed character of the set of equations of transonic flow is by replacing it with an equivalent set of equations of the hyperbolic type. It is only this equivalent set of equations that can be solved numerically, by difference methods.

A description of methods for constructing such an equivalent set of equations can be found in the literature quoted in the monograph [1]. The most popular method consists in introducing the time-dependence into the equations considered. Then, the equations become hyperbolic and steady flow is considered as a limit to which unsteady flow tends. The problem of solving a given problem of transonic flow is thus reduced to solving a problem of unsteady flow with appropriate boundary and initial conditions. Let us observe that the difference method for solving the equivalent problem has in this case an iterative sense. The initial condition plays the role of the initial approximation and the solutions for consecutive "time layers" are consecutive approximations to the solution sought for. The difference scheme is, in this case, an iteration formula. There being a great variety of difference methods for solving hyperbolic sets of equations, we can speak of a family of iterative methods for solving the equations of transonic flow.

The computation efficiency of an iterative method for solving equations of transonic flow is determined by the number of iterations necessary to obtain the steady state solution and the "computation cost" of a simple iteration. Disregarding the obvious influence of the choice of the initial approximation on the number of iterations necessary to obtain a solution, this number will be determined by the magnitude of the time step. For this reason it is advised in a number of papers to use implicit schemes, which enable computation with a Courant number $\nu > 1$ (see, for instance, [2]). However, the computation cost of a single iteration is, as a rule, higher for an implicit than for an explicit method. This fact is connected with the necessity of inverting large multi-diagonal matrices. Moreover, although the stability condition for linearized conditions does not impose limitations on the time step in the implicit method, nonlinear instability or the way of approximating the boundary conditions make it often impossible to perform the analysis for $\nu \gg 1$. This is the cause of the fact that explicit difference schemes are those which are more often suggested in the literature as a basis for iterative methods of solving the transonic flow equations.

In the survey paper [3] it was observed that certain modifications of iterative methods for solving equations of transonic flow are possible. If the aim of the analysis is not to determine the path leading to a steady state but only the fields of flow parameters for that state, it is not necessary for the consecutive approximations to correspond to consecutive stages of the physical process of the flow becoming steady. This fact enables us to change the form of secular terms in the equations of flow so as to simplify the computation and to improve the convergence to a steady state. Nor it is required for the difference scheme to constitute an accurate counterpart of the equations of unsteady flow. Those ideas were used for constructing solutions of the equations of transonic potential flow. A survey

of those methods can be found in [3] and in the more recent paper [4]. How far the methods for transonic potential flow have attained the high degree of development, the methods for rotational flow have been paid much less attention. The method of replacement of the unsteady equation of conservation of energy by the Bernoulli equation (for homo-energetic flows), the method of different time steps used for different space cells and the method of introducing the terms of the type of artificial viscosity (which damp the transition processes), are mentioned in [3] as all the existing methods for improving the convergence of iteration. Some new concepts have been introduced in [5] to [8]. They consist in each iteration being divided into a number of sub-iterations, every subsequent iteration giving a solution for a mesh of larger size. This enables us to transfer the influence of the disturbances produced in various parts of the region over the entire region. This gives an effect which is equivalent to a certain acceleration of wave motions, which make the flow become steady as a result of a process modelled by non-steady equations.

Some new methods for accelerating the iteration process of solving the equations of transonic flow will be proposed in the present paper, a numerical example being used to study the problem of reduction of the number of iterations necessary to obtain a satisfactory solution.

2. Description of the modification of the standard algorithm

The methods which will be proposed for improving the convergence of an iterative procedure for solving the equations of transonic flow may be used for two- or three-dimensional motions as well, the case of two-dimensional motion of a perfect fluid being used here as an example. Thus, let us consider the algorithm for constructing a solution of the set of equations.

$$\oint_C a\,dr - b\,dx = -\iint_S \frac{\alpha}{r^\alpha} f\,dx\,dr, \tag{2.1}$$

where

$$a = \begin{bmatrix} \varrho u \\ p+\varrho u^2 \\ \varrho uw \\ \varrho u h_0 \end{bmatrix}, \quad b = \begin{bmatrix} \varrho w \\ \varrho uw \\ p+\varrho w^2 \\ \varrho w h_0 \end{bmatrix}, \quad f = \begin{bmatrix} \varrho w \\ \varrho uw \\ \varrho w^3 \\ \varrho w h_0 \end{bmatrix}$$

and C is the contour bounding a region S in the x, r-plane, u, w denote the velocity components, ϱ — density, p — pressure, h_0 — total enthalpy of slowing down and α — coefficient of symmetry ($\alpha = 0$ or 1).

The total enthalpy of slowing down is

$$h_0 = \frac{u^2+w^2}{2} + h(p, \varrho), \tag{2.2}$$

where h is the total enthalpy of the fluid.

A solution of the set of equations (2.1) is sought for in the quadrangular region Ω bounded by the lines C_x^-, C_x^+, C_r^- and C_r^+ (Fig. 1), on which some boundary conditions are prescribed, the form of which depends on the type of the problem to be studied. The solution process starts from dividing the region Ω into cells Ω_{ij} by using a certain algorithm. We prescribe now initial values for the flow parameters in particular cells or nodes. They constitute the initial condition or, in other words, the initial approximation of the iterative method. Further approximations will be constructed by means of a difference approximation to the equations of unsteady flow

$$\oint_S \oint \omega\,dx\,dr + a\,dr\,dt + b\,dx\,dt = -\int\int\int_V \frac{\alpha}{r^\alpha} f\,dx\,dr\,dt, \tag{2.3}$$

where

$$\omega = \begin{bmatrix} \varrho \\ \varrho u \\ \varrho w \\ \varrho h_0 \end{bmatrix}$$

and S is any surface bounding a volume V in the x, r, t-plane.

We shall consider algorithms based on explicit difference schemes:

$$\omega_{ij}^{k+1} = F(\{\omega\}_s^k, \{x\}_s^k, \{r\}_s^k, \{x\}_s^{k+1}, \{r\}_s^{k+1}, \tau), \tag{2.4}$$

where $\{\omega\}_s^k$, $\{x\}_s^k$, $\{r\}_s^k$ are sets of values of the parameters of flow and the coordinates of the nodes on the pattern of the difference scheme in the k-th time layer, the symbol τ denoting the time step of the procedure. The standard algorithm for the construction of a solution includes the determination of the time step τ from the stability condition, the determination of the new locations of the nodes and the computation, from the iteration formula (2.4), of the new approximate values of the parameters of flow in all the meshes or at all the nodes. This algorithm is characterized by a rather slow convergence to the steady-state solution, what can be explained on the grounds of a physical interpretation. The physical process of the flow becoming steady is connected with wave motion, that is propagation of mechanical disturbances within the flow region. In order to reach the steady state, the disturbances produced in a certain region are subject to multiple reflections and pass many times the entire region. This process reflects in the construction of the solution of the equations of unsteady flow by means of explicit difference schemes. From the Courant–Friedrichs–Levi stability condition it follows that the minimum number of iterations necessary for a disturbance to pass between opposite boundaries of the region is equal to the number of meshes between those boundaries. Since the stability condition selects the least value of the time step for all the cells, the number of iterations necessary for the disturbance to pass may be much higher in reality. This unfavourable effect can be reduced by conducting the computation for each cell or node with a time step determined from the local stability condition. This modification of the standard algorithm makes the solution obtained for a subsequent time layer not coincide with the subsequent phase of physical evolution of the flow.

The relation between the number of iterations necessary to attain steady state and the number of cells, into which the flow region has been divided, makes us seek for a compromise between the requirement of accuracy of describing the flow structure and the desire to reduce the volume of computation. Some solution to this problem is furnished by the use of a variable number of cells. The analysis is started from the largest meshes. When the solution has been established, we pass to finer meshes. Such a passage may be performed several times. The sense of this procedure is clear if we observe that approaching the computation at finer meshes we start out from an approximation which is nearer the accurate solution than an approximation selected for coarse meshes.

The fundamental modification of the standard algorithm consists in a change of the sequence of operations in an iteration, that is to say the parameters ω_{ij}^k for a mesh or a node (i, j) are replaced by the new computed values of the parameters in the neighbouring meshes or nodes. To explain the nature of this modification in greater detail let us consider Fig. 1 and assume that the $(k+1)$-th iteration begins from the cell at the apex C of the flow region. Making use of the parameters in the neighbouring cells which were evaluated in the k-th iteration, we find, from (2.4), the new approximation ω_{41}^{k+1}. We replace now ω_{41}^k by ω_{41}^{k+1}. The new values ω_{41}^k are now used for computing the parameters in the cell (4,2) and, having computed ω_{42}^{k+1}, we substitute it for ω_{42}^k. They are involved, in turn, in the equation for computing the parameters in the cell (4,3) etc. Let us observe that, to find the new approximation to the values of the parameters in the cell (3,2), use is made of the new values of the parameters in the cells (3,1) and (4,2). Thus, the new approximation to the values of the parameters in the cell (1,3) adjacent to the corner A will depend in some measure on the new value of the parameters in the cell (4,1). Thus, a disturbance produced in the neighbourhood of the apex C travels through all the cells during a single iteration. With the standard algorithm this would require, approximately $I+J$ iterations, the symbols I and J denoting the number of cells in the x and r direction, respectively.

Following the analogy with the process of propagation of disturbances, we produce a reflection and a change of the direction of motion of the disturbance. This is realized in the algorithm by changing the direction of computation to make it opposite to that of the arrows in Fig. 1. After determining the new approximation to the values of the parameters in the cells from (1,3) to (4,1), we find, as a result of next iteration, the values of the parameters in the cells starting out from the cell (4,1) and ending at (1,3).

The next modification of the standard algorithm concerns the procedure of changing the location of nodes. The motion of the nodes is connected with that of the boundaries of the flow region. This is the case of boundaries constituting shock wave fronts or flow boundaries. Their location in a steady state is unknown, therefore their initial location must be prescribed and varied in the course of the solution procedure. The motion of the boundaries and the related motion of the nodes inside the region are accounted for in the stability condition and in the form of the difference scheme. In the standard algorithm the magnitude of displacement of the boundary nodes is equal to the product of the velocity of motion of the element of the boundary considered and the time step τ. The locations of the internal nodes are adjusted to the new position of the boundaries. Their displacements are of the same order of magnitude as those of the boundary nodes. Such a procedure of remodelling the grid corresponds to the physical evolution of the flow, but prolongs consider-

ably the computation time necessary to attain the steady state. It is reasonable, therefore, to consider an algorithm of remodelling the grid so as to accelerate the motion of the nodes.

The boundaries of the flow region, which change their form and position in consecutive approximations, may have a nature of shock-wave fronts, or flow boundaries or sonic surfaces. For a steady-state those boundaries take the forms determined by the parameters at the boundary nodes or in the boundary cells. Thus, the inclination of various segments of the shock wave front to the x-axis is determined by the value of the shock disturbance angle at the corresponding node or cell, the inclination of the flow boundary is determined by the value of the turning angle of flow, and the inclination of the sonic line — by the value of the disturbance angle. Knowing the position of the relevant corner node (A, B, C or D) and the parameters at the boundary nodes or meshes, we can determine the form of the boundary. As an example let us consider the method of constructing a segment of the boundary in the case when node A (Fig. 1) is fixed and the adjacent boundary AD

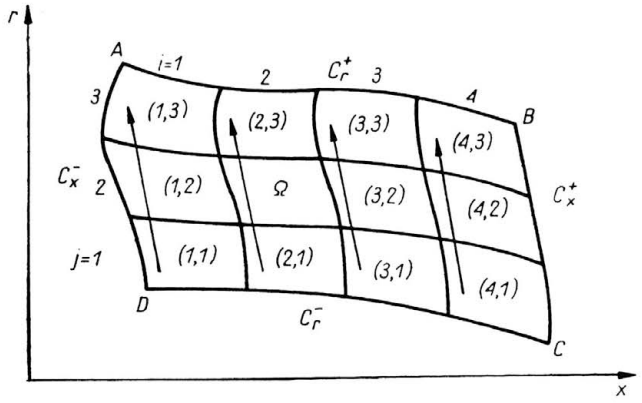

FIG. 1.

constitutes a shock wave front. Knowing the values of the flow parameters in the mesh (1,3) and the parameter of undisturbed flow before the wave front, we can determine the shock disturbance angle which is the angle between the segment of the wave front originating from the node A and the x-axis. By selecting in a definite manner the coordinate r_3 of the neighbouring node of A, we determine the coordinate x of that node (Fig. 1). This node becomes the origin for the next segment of the wave front, inclined to the x-axis at an angle determined from the values of the parameters at the cell (1, 2). Subsequent segments of the wave front are constructed in the same manner.

Modification of the procedure of changing the positions of the nodes consists in the procedure of constructing the boundaries of the flow region described above for each iteration. The location of the internal nodes is adjusted to the new position of the boundaries in the same manner as it was done in the standard algorithm. The modified procedure ensures agreement between the form of the boundary and the values of the parameters of flow obtained at the boundary nodes or cells. If the standard algorithm is used, this agreement is attained in an asymptotic manner. To explain the consequences of the difference between the standard and the modified procedures of constructing the variable boundaries

of flow, use will be made of the example which has already been considered above. Let us observe that the displacement of the node D determined by using the modified procedure depends on the displacements of all the nodes lying at the boundary AD starting out from the node next to A. In the case of the standard algorithm, it depends only on the flow parameters in the mesh (1,1) and on the time step. This reduces the magnitude of the displacement. On the one hand this means that the convergence of the method is slower and, on the other hand, that it prevents any significant reduction in the time step (from the stability condition it follows that the admissible value of the time step is lower for larger displacements of the nodes). Superposition of the displacements of boundary nodes in the modified procedure leads to relatively large displacements of nodes distant from the fixed nodes. This enables us to approach at a higher rate the steady state position of the boundaries, but the motion of the nodes may lead to a considerable reduction of the time step. To avoid this, new approximations are evaluated for fixed meshes. This means that the idea of modelling the physical exolution of the flow is partly abandoned, but we are able to construct a sequence of successive approximations convergent to a steady solution. Let us observe that the displacement of the nodes decreases as the steady state is approached. As a consequence, the error due to fact that the motion of the nodes has been disregarded in the difference scheme also decreases. It follows that the sequence of successive approximations constructed above approaches the solution sought for.

The non-local character of the algorithm of successive approximations, which is manifested by the dependence of the parameters in a given cell (or node) and the position of a node on the parameters in all the cells (nodes) and the position of all the other nodes, enables us to approach relatively rapidly the steady state solution. However, considerable oscillations of the solution are produced if this state is approached, what makes it difficult to reduce the error of the solution below a certain value. In the standard algorithm there are also oscillations about a steady state but, owing to the local character of the algorithm, their period is much longer and the amplitude smaller. Consequently, the standard algorithm may prove to be more efficient in the vicinity of the steady state. Another way to obtain the same result is that of final averaging of the parameters. This method will be discussed by considering a particular case.

3. Numerical example

To analyse the methods proposed here aimed at improving the convergence of iterative methods of solution of the transonic flow equations, let us consider the problem of compression of a column of barotropic liquid by a pressure pulse propagating at a constant supersonic velocity D along the wall of the column. The motion of the liquid is stationary in a system connected with the pressure pulse. In this system the undisturbed flow moves in the direction of the x-axis at a supersonic speed D (Fig. 2).

The shock wave generated by the action of the pressure pulse slows down the motion of the liquid. As a result, a region of subsonic flow is formed. Now the expansion of the flowing medium due to the drop in pressure acting on the boundary of the flow region results in the motion being accelerated up to a supersonic speed. This means that a successive region of supersonic flow appears.

The solution of the equations of flow is constructed in the region bounded by the segment DC of the symmetry axis (Fig. 2), the shock wave front AD, the segment AB of the boundary of the flow region and the segment BC of supersonic flow. The boundary conditions at the shock wave front AD has a form of the Hugoniot conditions, the pressure is prescribed at the boundary AB of the flow region, the radial component of the velocity vanishes at the axis of flow, and a condition of non-reflecting boundary is prescribed at the line BC.

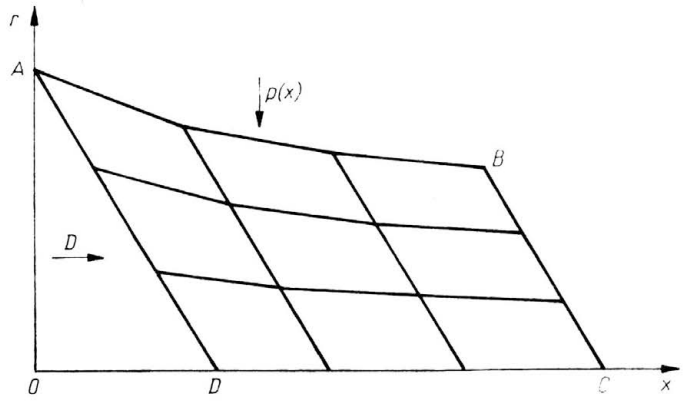

FIG. 2.

The problem considered is a model of the process occurring in a cylindrical specimen under the action of the pressure, exerted by the detonation products of an explosive, moving along the lateral surface of the specimen. A similar problem was solved in [9] by using the standard algorithm. The results of computation corresponding to the experimental data described in [10] and [11] will be presented. In those experiments detonation products were acting on cylindrical specimens of a radius $r_0 = 12{,}5$ mm, made of plexiglass. The velocity of the detonation wave was $D = 6750$ m/s. The distribution of pressure acting on the sample was determined in [11] by means of manganine gauges. Plexiglass subjected to shock pressure was modelled by a barotropic liquid. The form of the relation between the density ϱ and the pressure p was assumed to be

$$\varrho = \frac{2\lambda^2 \varrho_0 p}{2\lambda(\lambda-1)p - \varrho_0 a^2 + a\sqrt{(\varrho_0 a)^2 + 4\lambda\varrho_0 p}}, \tag{3.1}$$

where $\varrho_0 = 1185$ kg/m^3, $a = 2600$ m/s and $\lambda = 1.5$.

The first stage of constructing the solution comprises the establishment of the initial form of the boundaries AB, BC and AD, the construction of the grid and the choice of the initial values of the flow parameters in the meshes. The rectilinear segment AD of the wave front was assumed to be inclined to the axis at an angle equal to the disturbance angle for a wave with a pressure amplitude equal to the initial value of the pressure of the loading pulse. The segment AB of the boundary of the flow region was subdivided into segments corresponding to even steps Δx. Making use of the boundary condition, the mean value of the pressure acting on each of the boundary segments was determined. From the relations for an oblique shock wave the velocity components of flow were determined. Knowing the turning angle of flow, radial positions of the nodes lying at the boundary

were found. Then, the meshes were constructed. To this end rectilinear segments parallel to the boundary AD were drawn starting from each node on the boundary AB. Each of them was divided into equal segments, their ends indicating the positions of the nodes. By connecting nodes lying on neighbouring segments quadrangular meshes are obtained. In the individual rows of meshes parallel to the boundary AD we assume the same values of parameters as those determined for the corresponding segments of the boundary AB. Thus, we have determined the boundaries of the region, the location of the nodes and the initial values of the flow parameters in all the meshes.

To find the next approximation use is made of the S.K. Godunov's difference scheme [12]. The parameters on the lateral walls of a mesh (except the boundary of the region) are determined by making use of the acoustic approximation to the solution of the problem of disintegration of an arbitrary discontinuity. In the next iteration, in agreement with the algorithm described in the foregoing section, the sequence of computation of the flow parameters in individual meshes is changed. After each iteration the meshes are rebuilt. Starting out from the node A, segments inclined to the axis at an angle equal to that of the shock disturbance are drawn from successive nodes at the boundary AD. The radial coordinates of the nodes lying at the wave front do not undergo any change, contrary to the case of axial coordinates. Further segments of the boundary of the flow region are determined in a similar manner. In this case their inclination angle coincides with the turning angle of flow in the meshes adjacent to the boundary. The x-coordinates of the nodes located at the boundary remain unchanged. Inside the region the meshes are constructed so that for a fixed j all the segments are parallel to the corresponding segment of the AD wave front and the even division along the r-coordinate is preserved.

The multigrid procedure is used for the construction of the solution. The initial approximation is determined for coarse meshes. After the solution has been established for a given grid we pass to finer meshes by doubling the number of meshes in both directions. The parameters in each of the four meshes formed from an original mesh are assumed to be equal to those of the original mesh. The following quantity is used as a measure of the degree of stabilization of the solution obtained for the given meshes:

$$E = \frac{1}{r_0}\sqrt{\sum_{k=1}^{s}\frac{(x_D^k-\bar{x}_D)^2}{s}}, \tag{3.2}$$

where x_D^k is the coordinate of the node D as determined in the k-th iteration, and $\bar{x}_D$ is the mean value of x_D resulting from s iterations. If the condition $E < \varepsilon$ is satisfied, the procedure is repeated for finer meshes or the analysis is finished. In the former case the quantity ε decreases.

Figures 3 to 6 represent diagrams illustrating the variation of the relative deviation $\Delta = (x_D - x_{DU})/r_0$ of the position of the node D from its steady location x_{DU}, as a function of the number of iterations N. Diagram 1 in Fig. 3 concerns the standard algorithm. It has been drawn for the case in which the $ABCD$ region has been divided into 32×16 meshes. Diagram 2 illustrates the convergence of the solution to the steady state for the multigrid procedure. The initial condition was prescribed on 8×4 grid. After 44 iterations we have passed to the 16×8 meshes and, after further 106 iterations — to 32×16 meshes.

Confronation of the diagrams 1 and 2 shows that there is no essential difference between the rate of convergence for the single-grid and multigrid algorithms. It should be borne in mind, however, that the computation cost of one iteration with 8×4 meshes is one-

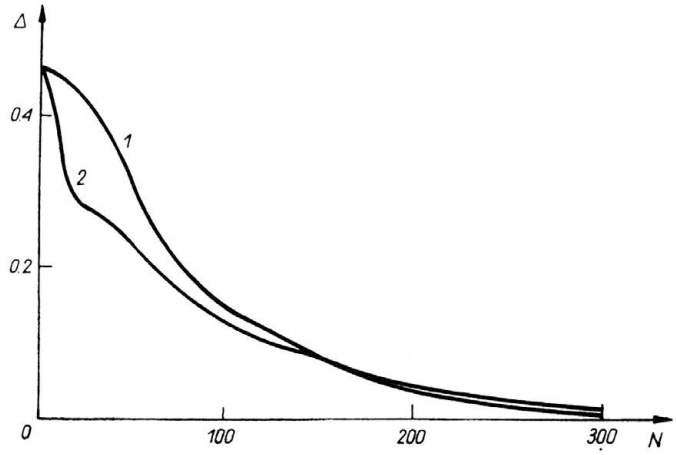

FIG. 3.

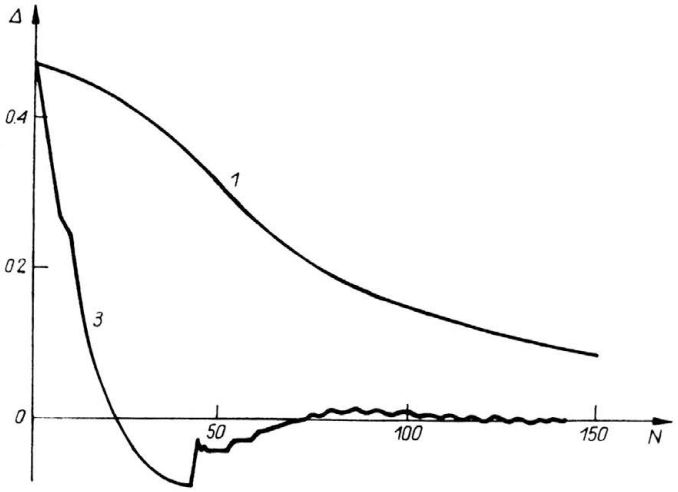

FIG. 4.

sixteenth and with 16×8 meshes one quarter of that for 32×16 meshes. This is the principal advantage of the multigrid procedure.

Figure 4 shows diagrams of $\Delta(N)$ for the standard algorithm (diagram 1) and for the modified algorithm in which the local value of the time step is used, the sequence of computation in an iteration is changed and the modified procedure of rebuiliding the meshes is used (diagram 3). Confrontation of the two diagrams shows that the modifications introduced result in the fact that the steady state is approached at a much higher rate than in the case of the standard algorithm. The same conclusion can be drawn from the

confrontation of the diagrams in Fig. 5. Diagram 2 has been drawn for the standard algorithm and the variable mesh procedure and diagram 4 — for the modified algorithm, the multigrid procedure being also used. Comparison of the diagram 3 of Fig. 4 with the diagram 4 of Fig. 5 shows that also in the case of the modified algorithm, the use of the multigrid procedure does not change in any essential manner the number of iterations necessary to obtain a solution.

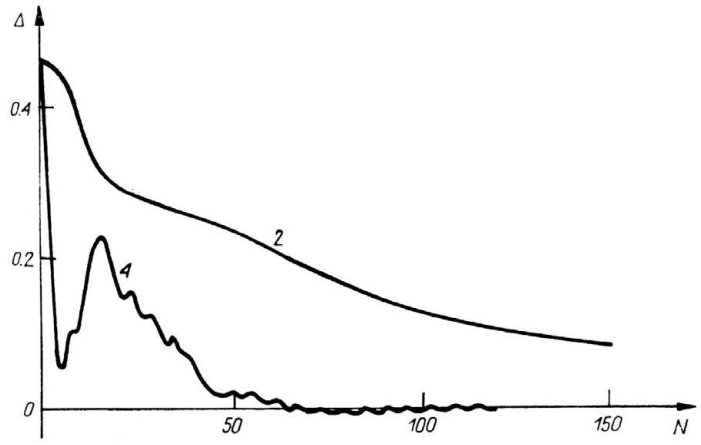

FIG. 5.

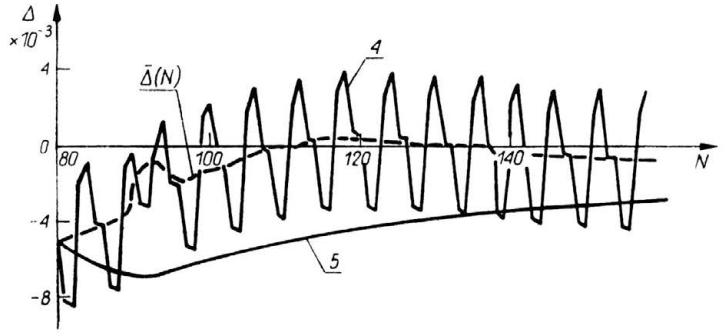

FIG. 6.

As observed in the foregoing section, the non-local character of the modified algorithm results in the solution becoming oscillatory, what makes it difficult to obtain the required accuracy. In the vicinity of the steady state the oscillations become regular, with constant amplitude and period. This is illustrated in the diagram 4 of Fig. 6, which is an enlarged fragment of the diagram 4 of Fig. 5. To obtain the required accuracy it is necessary to pass to the standard algorithm. Diagram 5 illustrates the convergence history of the solution, the standard algorithm was introduced after 80 iterations performed according to the modified algorithm. It is seen that the convergence to the steady state is monotonic. However, the return to the standard algorithm does not allow us to make full use of the advantages of the modified algorithm. For, if we confront diagram 5 with the diagram of

$\bar{A}(N)$ obtained by averaging the successive extremes of diagram 4, it is found that the averaged solution tends to the steady state at a much higher rate. To use the modified algorithm it is necessary to introduce a certain procedure of final averaging of the solutions. The condition of convergence of the solution must also be changed. The quantity E determined by Eq. (3.2) is replaced with E_1 which is found from the formula

$$E_1 = \frac{1}{r_0}\sqrt{\sum_{k=1}^{s}\frac{(\bar{x}_D^k - \bar{x}_D)^2}{s}}, \tag{3.3}$$

where $\bar{x}_D^k$ denotes the mean value of the successive extremes of the function $x_D(N)$. The condition of attaining the steady state has the form $E_1 < \varepsilon$.

If it is established that the condition of the solution becoming steady is satisfied, we determine the parameters in the meshes for two consecutive extrema of the function $x_D(N)$. As a final solution we assume the mean value of the results obtained for the iterations corresponding to those points. The differences between the solution thus obtained and the solution obtained by means of the standard algorithm are contained within the limits of the required accuracy of solution. At the same time the number of iterations necessary to obtain a solution reduces to a number several times smaller than before.

Figure 7 represents the results of solving a test problem. The form of the shock wave front ADA and the lateral surface of the compressed specimen AC are shown. The boundaries of the regions of sub- and supersonic flows are shown as well as some selected isobars. Chain lines show the form of the shock wave front which was obtained experimentally in [9]. It is seen that this form does not differ much from that obtained by computation.

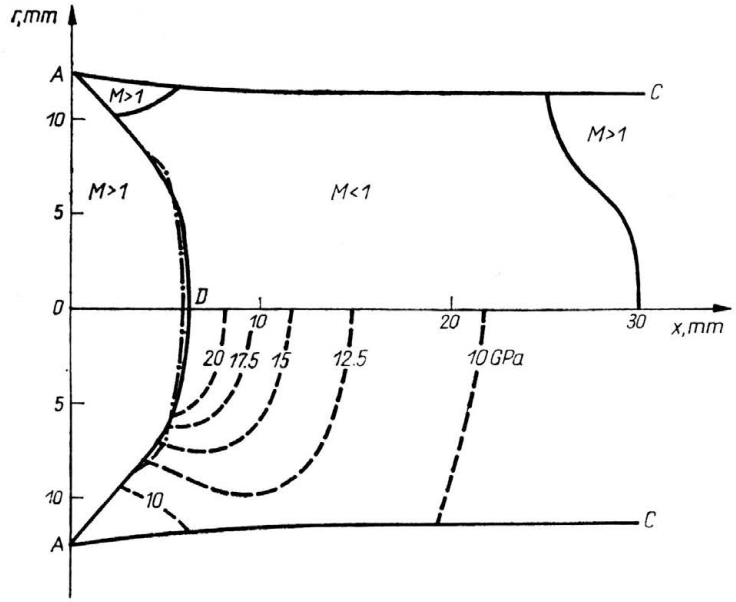

FIG. 7.

References

1. W. J. PROSNAK, *Fluid mechanics*, PWN, Warszawa 1971 [in Polish].
2. M. JA. IWANOW, P. Z. NIGMATULIN, *Implicit S. K. Godunov scheme of improved accuracy scheme for numerical integration of the Euler equation*, **27**, 11, 1987 [in Russ.].
3. H. VIVIAND, *Pseudo-unsteady methods for transonic flow computations*, Lect. Notes Phys., **141**, 44, 1981.
4. M. YA. IVANOV, W. W. KORECKIJ, *Flow analysis in two-dimensional and spatial nozzles by the method of approximate factorization* ŻWMMF, **25**, 9, 1985 [in Russ.].
5. RON-HO NI, *A multiple-grid scheme for solving the Euler equations*, AIAA J., **20**, 11, 1982.
6. R. V. CHIMA, G. M. JOHNSON, *Efficient solution for of the Euler and Navier–Stokes equations with a vectorized miltiple-grid algorithm*, AIAA J., **23**, 1, 1985.
7. W. K. ANDERSON, J. L. THOMAS, *Multigrid acceleration of the flux-split Euler equations*, AIAA J., **26**, 6, 1988.
8. D. J. MAVRIPILIS, *Multigrid solution of the two-dimensional Euler equations on unstructured triangular meshes*, AIAA J., **26**, 7, 1988.
9. W. G. LOBANOV, *Numerical modelling of flows in cylindrical specimens under compression by a grazing detonation wave*, PMTE, 5, 1975 [in Russ.].
10. G. A. ADADUROW, A. N. DREMIN, G. I. KANEL, *Parameters of Mach reflection in plexiglass cylinders*, PMTF, 2, 1969 [in Russ.].
11. A. N. DREMIN, G. I. KANEL, W. D. GŁUZMAN, *Experimental study of pressure profiles under irregular reflection of a conical shock wave in a plexiglass cylinder*, FGW, **8**, 1, 1972 [in Russ.].
12. *Numerical solution of multidimensional problems of gasdynamics*, ed. by S. K. Godunov, Moskva 1976 [in Russ.].

MILITARY TECHNICAL ACADEMY, WARSZAWA.

Received April 20, 1989.