Computers & Fluids, Vol. 1, pp. 73–100. Pergamon Press, 1973. Printed in Great Britain.

A NUMERICAL SOLUTION OF THE NAVIER–STOKES EQUATIONS USING THE FINITE ELEMENT TECHNIQUE

C. TAYLOR and P. HOOD
Department of Civil Engineering, University of Wales, Swansea, Wales

(*Received* 3 *August* 1972)

Abstract—The finite element discretisation technique is used to effect a solution of the Navier Stokes equations. Two methods of formulation are presented, and a comparison of the efficiency of the methods, associated with the solution of particular problems, is made. The first uses velocity and pressure as field variables and the second stream function and vorticity. It appears that, for contained flow problems the first formulation has some advantages over previous approaches using the finite element method[1, 2].

1. INTRODUCTION

The Navier–Stokes equations governing the flow of fluids, are known to have applications to a wide range of engineering problems. In the past, the numerical solution of the equations has mainly been achieved by the finite difference technique, where the resulting flow patterns have been shown to correspond, within acceptable tolerances, to experimental results[3]. The alternative finite element method, well proven in the structural mechanics field, has, in general, certain advantages over the finite difference techniques. These are the ease with which irregular geometries, non-uniform meshes and imposition of appropriate boundary conditions can be applied. In the field of fluid mechanics the use of the finite element method is a relatively new innovation, but it is anticipated that the method will again prove superior to the finite difference technique in the ways mentioned above. Indeed it is quite fair to say that for potential flow problems the lead of the finite element method has already been established[4, 5].

A limited attempt has already been made to solve the Navier–Stokes equations by the finite element method[1, 2]. These approaches seek to eliminate the continuity equation by using specialised element shape functions or other devices. The pressure field is then determined from a Poisson equation containing the instantaneous velocity. Unfortunately neither the pressure nor its normal gradient on the solid boundaries is known until the velocity field is determined. This difficulty is avoided with the present method, which uses a direct coupling of all the equations to obtain a solution. There is a requirement therefore for the pressure to be specified only at the inlet to or exit from the fluid domain.

The major part of the paper is devoted to a formulation involving the variables of velocity and pressure. However an alternative formulation using stream function and vorticity is also presented. The velocity–pressure formulation is shown to be the more efficient of the two methods on a computer time comparison for some contained flow problems. These problems are in the low and intermediate range of Reynolds number, some of which include separated flow. Since isoparametric elements[6] are used it is possible to analyse flow in regions of quite arbitrary shape.

The formulation can be modified to cover a number of different situations. The most natural extension is to turbulent flow where the variables are replaced by averaged values, and it is hoped to present this in a future paper. Perhaps easier applications are found in other branches of mechanics, since the Navier–Stokes equations are similar in form to many other equations. For instance the equation governing tidal hydraulics or dispersion phenomena, can both be obtained from the Navier–Stokes equations. In these instances the same computer program can be used with only minor modifications to effect a solution.

2. SOLUTION OF PARTIAL DIFFERENTIAL EQUATIONS BY FINITE ELEMENTS

The finite element method is based on the use of series expansions within subdomains or elements V^e, into which the domain of interest, V, is divided (Fig. 1). The approximate representation $\phi^{(a)}(x, y)$ of any function $\phi(x, y)$ will be written in terms of unknown parameters ϕ^i, within each element

$$\phi^{(a)}(x, y) = \sum_{i=1}^{l} N^i(x, y)\phi^i \tag{1}$$

where there are l unknown parameters and $N^i(x, y)$ are the shape functions. The desirable properties of the shape functions and the order of inter-element continuity required is discussed in refs[2, 7–9]. To avoid complicated algebra, the fact that the unknown

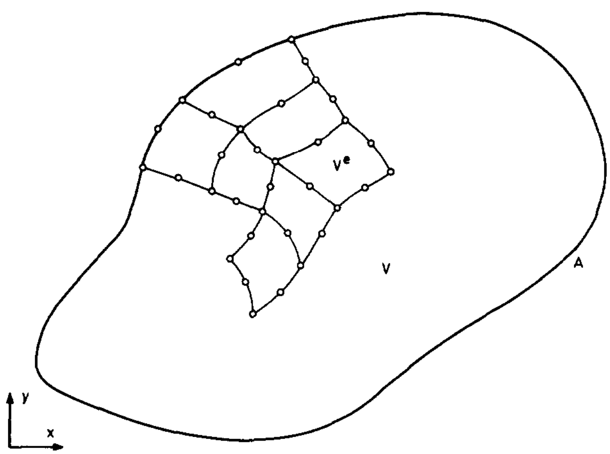

Fig. 1. Division of a domain into finite elements.

parameters ϕ^i may include derivatives at nodes will be implicitly assumed. These unknown parameters are determined generally in three ways:

(i) Minimisation of a functional whose associated Euler equations are the partial differential equations to be solved.

(ii) The method of weighted residuals (particularly the Galerkin method) in which the error resulting from the substitution of the approximate functions $\phi^{(a)}(x, y)$ into the governing differential equations, is distributed over the whole domain. This distribution is such that, on average, the partial differential equations are obeyed.

(iii) The direct method in which the $\phi^{(a)}$ are substituted directly into an integral equation. The integral equation typically consists of the energy integrated over the whole domain.

Other methods include the use of quadratic programming with linear constraints. A fuller discussion of this and the other methods in this section is presented in ref.[16]. The method used here is the Galerkin technique[10, 13–16].

The direct method[1, 11] and a quasi-variational approach[12, 22] can also be employed but the final set of equations obtained is the same in all cases. The solution of such a set of equations can be effected in three ways:

(i) *Direct coupling method*

In this method all the finite element equations are assembled directly into a single matrix. The method is extremely simple to apply but the bandwidth of the resulting matrix is rather large. This technique is used with reference to the velocity–pressure formulation in a subsequent section.

(ii) *Uncoupled method*

The method can be used where each of the equations in the system can be solved independently. When this cannot be done it is sometimes possible to eliminate or modify one set of equations (e.g. the continuity equation typically) at element shape function level or using other techniques[1, 17, 18]. The remaining equations can then be uncoupled and solved independently. The bandwidth is always smaller than in the direct coupling method but the accuracy is usually less due to the satisfaction of an equation at element rather than domain level. It is also established[19] that using this method in connection with the wave equation introduces spurious reflected wave solutions. These are eliminated if the direct method is used.

(iii) *Iterative method*

When boundary conditions relating to one variable are not available it is not possible to use the two former methods. For such a situation the procedure used in this paper is to assume starting values for this variable and iterate until self-consistent fields are obtained. This situation arises in the stream function vorticity formulation and is demonstrated subsequently.

3. THE DYNAMICAL EQUATIONS

The derivation of the flow equations can be found in many texts e.g.[20]. From a consideration of conservation of mass the continuity equation is derived:

$$\frac{\partial \rho}{\partial t} + (\rho v_i)_{,i} = 0 \tag{2}$$

where ρ = fluid density

v_i = velocity component in the x_ith direction, and comma denotes partial differentiation with respect to the following suffix. The summation convention will be used unless otherwise stated.

With the assumption of incompressible flow, equation (2) reduces to:

$$v_{i,i} = 0. \tag{3}$$

The principle of conservation of momentum applied to a fluid element leads to a set of equations of which the j^{th} component is:

$$\rho a_j = \rho F_j + \sigma_{ij,\,i} \tag{4}$$

where

$$a_j = \frac{\mathrm{d}v_j}{\mathrm{d}t} = \frac{\partial v_j}{\partial t} + v_i v_{j,i}. \tag{5}$$

F_j is the j^{th} component of body force, and σ_{ij} is the symmetric stress tensor defined by the relation

$$S_i = \sigma_{ij} l_j \tag{6}$$

where a stress S_i is exerted in the x_i direction across a surface whose outward normal from the fluid has direction cosines l_j. For Newtonian fluids the constitutive relation for the stress tensor is

$$\sigma_{ij} = -p\delta_{ij} + 2\mu d_{ij} \tag{7}$$

where

$$d_{ij} = \frac{1}{2}\left(\frac{\partial v_j}{\partial x_i} + \frac{\partial v_i}{\partial x_j}\right) \tag{8}$$

$\mu =$ viscosity coefficient

$p =$ the pressure.

$$\delta_{ij} = \begin{cases} 0 & \text{if} \quad i \neq j \\ 1 & \text{if} \quad i = j. \end{cases}$$

The equation of conservation of energy can be written

$$\frac{\mathrm{d}}{\mathrm{d}t}(T + U) = W + Q \tag{9}$$

where T is the kinetic energy, U is the internal energy, W is the rate of work done on the fluid, and Q is the heat energy. These are further defined as:

$$T = \frac{1}{2}\int_V \rho v_K v_K \,\mathrm{d}V \qquad U = \int_V \rho E \,\mathrm{d}V$$

$$W = \int_V \rho F_K v_K \,\mathrm{d}V + \int_A S_K v_K \,\mathrm{d}A$$

$$Q = \int_V \rho H \,\mathrm{d}V + \int_A q_K n_K \,\mathrm{d}A \tag{10}$$

where E is the internal energy per unit mass. F and S are as defined previously, H is the heat per unit mass generated internally. q_K is the heat conducted per unit area in the x_K direction through the surface A with unit normal n.

These constitute the basic equations to be considered from which alternative forms may be derived. For two dimensional flow two new variables can be introduced, they are the stream function ψ and the vorticity ω defined by

$$u = \frac{\partial \psi}{\partial y}$$

$$v = -\frac{\partial \psi}{\partial x}$$

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \tag{11}$$

where, dropping the tensor notation, u, v are the x, y components, respectively, of the fluid velocity. Rearranging equation (11) yields:

$$\nabla^2 \psi = -\omega \tag{12}$$

while the introduction of ψ and ω into equation (4) gives

$$\frac{\partial \omega}{\partial t} + \frac{\partial \psi}{\partial y}\frac{\partial \omega}{\partial x} - \frac{\partial \psi}{\partial x}\frac{\partial \omega}{\partial y} = \nu \nabla^2 \omega \tag{13}$$

where $\nu = \mu/\rho$.

The final form of equation to be considered in the paper, uses only the stream function as a variable, and is derived from equation (4) to give

$$\frac{\partial}{\partial t}(\nabla^2 \psi) + \frac{\partial \psi}{\partial y}\nabla^2\left(\frac{\partial \psi}{\partial x}\right) - \frac{\partial \psi}{\partial x}\nabla^2\left(\frac{\partial \psi}{\partial y}\right) = \nu \nabla^4 \psi. \tag{14}$$

All the above equations deal with transient fluid motion. Steady state motion ensues under certain conditions and it will be assumed for the purposes of discussion that formally dropping the transient terms will result in the appropriate governing equations. Due regard must be given to the problem of whether this state is a stable one or indeed if it is ever physically attainable. This question, however, is outside the scope of this paper.

4. FORMULATION

4.1. *Velocity–pressure*

As mentioned earlier there are at least three equivalent ways of deriving the finite element form of Navier–Stokes equations and at least two methods of solving the resulting coupled equations. The approach which will be followed here is the Galerkin method to form the equations and the direct coupling method for their solution. The difference between the direct coupling method and the earlier methods[1, 2] will be apparent later, when the relative merits of each approach are discussed.

Let the element interpolation functions be defined by the equation:

$$v_i^{(a)} = N^J v_i{}^J$$
$$p^{(a)} = N^J p^J \tag{15}$$

(summation over the l repeated indices J is understood), where l is the number of nodal parameters in each element. In addition let the approximate representation of the stress tensor be written $\sigma_{ij}{}^{(a)}$ in an element e. Note that for convenience the same shape functions are chosen for velocity and pressure. Since the body force F is for most problems a constant, it does not require element discretisation, although variation can be accommodated in the above manner.

Substitution of (5) into (4) and applying the Galerkin method yields a set of equations of which the component in the x_jth direction of the kth equation is

$$\sum_{e=1}^{M} \iiint_{V^e} N^K\left\{\rho\left(\frac{\mathrm{d}v_j{}^L}{\mathrm{d}t} N^L - F_j\right) - \sigma_{ij,i}^{(a)}\right\} \mathrm{d}V^e = 0 \tag{16}$$

where summation is over all the M elements which contain node K. (Using the direct method (16) is derived from (9) and (10)).

The application of boundary conditions determines the procedure to be adopted next. The determination of realistic boundary conditions at inlet and exit of contained flow problems is an extremely difficult task. If the velocities at these points are unknown then it would seem that there are two reasonable assumptions which might be made. Either the value of the stress at entrance and exit or a value for the normal derivatives of the velocity components can be assumed. Suppose first that the value of stress is known, the application of Gauss theorem to (16) yields

$$\sum_{e=1}^{M} \iiint_{V^e} N^K\rho\left(\frac{\mathrm{d}v_j{}^L}{\mathrm{d}t} N^L - F_j\right) + N_{,i}{}^K\sigma_{ij}{}^{(a)}\, \mathrm{d}V^e - \sum_{e=1}^{M} \iint_{A^e} N^K\sigma_{ij}{}^{(a)}l_i\, \mathrm{d}A^e = 0. \tag{17}$$

Using relation (6) the stress tensor is replaced by the stress S_j in the surface integral. The term $\sigma_{ij}{}^{(a)}$ under the volume integral can be expanded using (15), (7) and (8).

Equation (17) then becomes

$$\sum_{e=1}^{M} \left\{\iiint_{V^e} \rho N^K\left(\frac{\mathrm{d}v_j{}^L}{\mathrm{d}t} N^L - F_j\right)\right.$$

$$\left. + N_{,i}{}^K[-p^L N^L\delta_{ij} + \mu(v_j{}^L N_{,i}{}^L + v_i{}^L N_{,j}{}^L)]\, \mathrm{d}V^e - \iint_{A^e} N^K S_j\, \mathrm{d}A^e\right\} = 0. \tag{18}$$

The other type of boundary condition arises if a reasonable assumption can be made regarding the normal velocity gradients. Starting from equation (16) and using equations (3), (7), (8) and (15) gives

$$\sum_{e=1}^{M} \iiint_{V^e} N^K\left\{\rho\left(\frac{\mathrm{d}v_j{}^L}{\mathrm{d}t} N^L - F_j\right) + N_{,j}{}^L p^L - \mu N_{,ii}^L v_j{}^L\right\} \mathrm{d}V^e = 0. \tag{19}$$

Using Green's theorem on the last term gives the required form

$$\sum_{e=1}^{M} \iiint_{V^e} N^K\left\{\rho\left(\frac{\mathrm{d}v_j{}^L}{\mathrm{d}t} N^L - F_j\right) + N_{,j}{}^L p^L\right\} + \mu N_{,i}{}^K N_{,i}{}^L v_j{}^L\, \mathrm{d}V^e - \iint_{A^e} N^K N_{,i}{}^L \mu v_j{}^L l_i\, \mathrm{d}A^e = 0. \tag{20}$$

The surface integral can be evaluated if the velocity gradients $v_{j,n}^{(a)}$ normal to the surface are known, since by definition (15)

$$v_{j,i}^{(a)} = N_{,i}{}^L v_j{}^L.$$

On rigid boundaries the conditions of zero tangential and normal velocity components are applied. The value of surface integrals of (18) or (20) is then immaterial on these surfaces, since the nodal velocities are known here and consequently these equations are not required for their determination. To effect a solution, boundary conditions must be imposed on the inflow and outflow boundaries. These can be of three types, (i) velocity specified, (ii) stress values and (iii) normal gradients of velocity.

The question arises as to when either of the assumptions—stress or normal velocity gradient can be reasonably made. It would appear that with long straight parallel sided exit and entrance regions an assumption of normal velocity gradient could be made. The assumption of any particular stress at inflow or exit would seem harder to justify but might be known from experimental investigations for example.

The Galerkin method applied to the continuity equation (3) completes the required set of equations and yields typically for the K^{th} node.

$$\sum_{e=1}^{M} \iiint N^K N_{,i}{}^L v_i{}^L \, dV^e = 0. \tag{21}$$

The equations (20) and (21) will be used to illustrate the direct coupling method of solution, an analogous result occurs for equations (18) and (21) if stress boundary conditions are known.

Limiting the approach to two dimensions for simplicity, the equations resulting at each node are combined together directly so that the two momentum equations (in 2 dimensions) and one continuity equation feature for each node.

The variables associated with the K^{th} node are

$$X_K = \begin{pmatrix} v_1{}^K \\ p^K \\ v_2{}^K \end{pmatrix}.$$

Assembling contributions from every element the standard matrix equation results:

$$[A]X = B \tag{22}$$

where the non-linear terms are incorporated in the $[A]$ matrix and a solution obtained by using a suitable iterative procedure. The submatrix a_{KL} associated with the K^{th} node which acts on the L^{th} nodal variables is as follows

$$a_{KL} = \begin{bmatrix} a_1 & a_2 & 0 \\ a_3 & 0 & a_4 \\ 0 & a_5 & a_1 \end{bmatrix} \tag{23}$$

where

$$a_1 = \sum_{e=1}^{M} \iint\limits_{A^e} N^K N^L + \nu N_{,i}{}^K N_{,i}{}^L \frac{\Delta t}{2} + N^K N^i \bar{v}_j{}^i N_{,j}{}^L \frac{\Delta t}{2} \, dA^e$$

$$a_2 = \sum_{e=1}^{M} \iint\limits_{A^e} N^K N_{,1}{}^L \frac{\Delta t}{2\rho} \, dA^e$$

$$a_3 = \sum_{e=1}^{M} \iint\limits_{A^e} N^K N_{,1}{}^L \, dA^e$$

$$a_4 = \sum_{e=1}^{M} \iint_{A^e} N^K N_{,2}{}^L \, dA^e$$

$$a_5 = \sum_{e=1}^{M} \iint_{A^e} N^K N_{,2}{}^L \frac{\Delta t}{2\rho} \, dA^e$$

and where $\nu = \mu/\rho$ the kinematic viscosity

$\Delta t =$ the time increment

$dA^e =$ two dimensional elemental area

$\tilde{v}_j =$ value of v_j obtained from previous iteration within the time step.

Note that a mid difference scheme has been used to deal with the transient terms and that the total derivative of velocity has been decomposed by means of (5) since an Eulerian description is used. The reasons for choosing the above time stepping scheme are outlined in the Appendix.

The B vector of equation (22) contains the prescribed surface gradients of velocity although using equations (18) and (21) it would contain the stresses. The $b_K{}^{\text{th}}$ subvector is as follows:

$$b_K = \begin{pmatrix} b_{K1} \\ 0 \\ b_{K2} \end{pmatrix}$$

where

$$b_{K1} = \sum_{e=1}^{M} \int_{l^e} N^K \nu v_{1,i} l_i \, dl^e + \sum_{e=1}^{M} \iint_{A^e} N^K \left\{ -N^i v_m{}^i N_{,m}{}^j v_1{}^j \frac{\Delta t}{2} + N^i v_1{}^i + F_1 \, \Delta t - \frac{N_{,1}{}^i p^i}{2\rho} \Delta t \right\}$$
$$- \nu N_{,j}{}^K N_{,j}{}^i v_1{}^i \, \Delta t/2 \, dA^e = 0 \tag{24}$$

where dl^e is the elemental length of the boundary line. The term b_{K2} is identical with b_{K1} except that the subscript 1 is replaced by 2 in (24). The values of v_i and p in the B vector are determined from initial conditions or the previous time cycle. The matrix equation for the steady state is obtained by putting $\Delta t = 2$, dropping the first term of a_1, i.e. $N^K N^L$, and putting the B vector zero.

The boundary conditions of velocity and pressure are applied directly to the matrix equation (22). If for example, a velocity component at node K on the boundary is known the corresponding component of the K^{th} momentum equation is deleted and replaced by a 1 on the diagonal and the prescribed value on the right hand side vector. If the pressure is known at the K^{th} node then the continuity equation at the K^{th} node is deleted and replaced as above. This choice is somewhat arbitrary and certainly for the steady state case it makes no difference if another choice of equations to be deleted is made. Provided that the choice is consistent with the matrix assembly.

Consideration of the order of derivatives in equation (22) will show that the order of continuity required is $C^{(0)}$. The choice of elements is thus vast but the most efficient element overall is probably the parabolic isoparametric element of[21]. Solutions obtained using equations (20) and (21) with the particular element seem to be satisfactory as will be subsequently shown.

The other method for solving the equations is that discussed in[1, 11] where one equation is eliminated. By choice of special shape functions for velocity the continuity equation can be satisfied at element level. The removal of the continuity equation from the set of equations means that insufficient equations are available for the determination of the pressures. The additional equations are provided by the Poisson equation relating pressures to instantaneous velocity:

$$p_{,jj} = \rho F_{j,j} + 2\mu d_{ij,ij}. \tag{25}$$

The corresponding boundary conditions required are pressure or its normal derivative at all points on the boundary. The pressure on inlet and exit to a contained region can be assumed but the pressure is not known generally on a rigid wall. The pressure gradient normal to the walls can be found from the equations (4) suitably rearranged, but unless a high order element is used the value of second derivatives $v_{i,jj}$ have to be assumed zero which normally would be valid for slow flowing fluids only. This would seem to be a distinct drawback of the method as compared with the direct coupling of equations where it is necessary to know the pressure only at inlet or exit. Although it is not suggested that the latter method does not have its drawbacks, for the $[A]$ matrix in equation (22) is unsymmetrical. Furthermore using direct coupling the bandwidth is three times greater than the method of [1, 11] where, correspondingly, the matrices involved can be symmetric.

4.2. *Stream function and vorticity formulation*

As an alternative to the previous formulation it is possible to use stream function and vorticity as variables. This time the continuity equation is implicitly satisfied and there are only the two governing differential equations (12) and (13). The solution is not as straightforward as it might appear since generally the vorticity is not known on the boundary walls, and the specification of boundary conditions on pressure except for the simplest problems is virtually impossible. As with the previous formulation the Galerkin method is used to obtain the finite element form of equations, while the solution of the equations is necessarily iterative. Another finite element formulation and solution of the problem has been presented[17], and uses a decomposition of the solution of (12) into a particular solution and complementary function.

The present method for solution of equations (12) and (13) uses an iterative method to obtain self-consistent stream function and vorticity fields, and the approach depends on whether the steady state or transient solution is required. Considering the steady state solution first, the equations to be solved are:

$$\nabla^2\psi = -\omega \tag{26}$$

$$\nu\nabla^2\omega = \frac{\partial\psi}{\partial y}\frac{\partial\omega}{\partial x} - \frac{\partial\psi}{\partial x}\frac{\partial\omega}{\partial y}. \tag{27}$$

The solution to (26) requires specification of ψ or its normal derivatives on all boundaries. It is therefore necessary to assume a velocity profile on entrance to and exit from the region, while on the boundary walls the velocities are zero. After fixing ψ at an arbitrary reference point on the boundary, the use of relations (11) provide the boundary ψ values. The vorticity is not known anywhere usually so an initial guess of zero vorticity is often appropriate, and equation (26) can then be solved for the stream function.

It now remains to solve (27) for the vorticity field. Suppose that ψ and ω are given by the element approximation functions

$$\begin{aligned} \psi^{(a)}(x, y) &= N^i(x, y)\psi^i \\ \omega^{(a)}(x, y) &= N^i(x, y)\omega^i \end{aligned} \tag{28}$$

where i is summed from 1 up to l, the number of nodes in an element. The boundary values of vorticity are obtained from the stream function field by use of relations (26) and (28). Differentiation of (28) gives the vorticity ω_K at the boundary node K.

$$\nabla^2\psi^{(a)}(x_K, y_K) = \nabla^2 N^i(x_K, y_K)\psi^i = -\omega_K. \tag{29}$$

When two or more elements meet at a node K the average of the values obtained by (29) is used. Instead of using (27) to find the vorticity field it can be obtained from:

$$\nabla^2\omega = 0 \tag{30}$$

with boundary conditions provided by equation (29), when the flow is not too irregular. The values of vorticity are substituted into (26) and the cycle is repeated until convergence is obtained. Once this is achieved the equation (27) is used in place of (30) to determine the vorticity field. The use of (30) rather than (27) has proved to speed convergence considerably, since the symmetric matrices resulting from (26) and (30) are identical and inversion is required only once at the beginning of the cycle. The procedure may be summarised as follows:

1. Determine suitable velocity profiles on inflow and exit

↓

2. From a reference stream function obtain boundary conditions on ψ

↓

3. Guess an initial ω distribution

↓

4. Solve $\nabla^2\psi = -\omega$

↓

5. Find the boundary ω values from $\omega_B = -\nabla^2\psi^{(a)}$

↓

6. Solve $\nabla^2\omega = 0$

convergence

↓

7. Solve $\nabla^2\psi = -\omega$ and find boundary ω values from $\omega_B = -\nabla^2\psi^{(a)}$

↓

Solve $\left(\nu\nabla^2 - \dfrac{\partial\psi}{\partial y}\dfrac{\partial}{\partial x} + \dfrac{\partial\psi}{\partial x}\dfrac{\partial}{\partial y}\right)\omega = 0$

convergence

↓

8. Solution obtained.

The finite element formulation of equations (26) and (30) is standard and may be found in [10] for example, while the formulation of (27) by finite elements requires further discussion. There are two choices essentially, either the term

$$\left(\frac{\partial \psi}{\partial y}\frac{\partial \omega}{\partial x}+\frac{\partial \psi}{\partial x}\frac{\partial \omega}{\partial y}\right)$$

is incorporated in the matrix for $\nabla^2\omega$ or it is put on the "right hand side" using the previously iterated ω values. The latter choice has proved unstable and therefore the former method was of necessity used. Thus applying the Galerkin method to (27) with element interpolation as in (28) results in a matrix equation:

$$[A]\{\omega\} = 0$$

where ω is the column vector of nodal vorticities, and the $A_{KL}{}^{\text{th}}$ term of the matrix A is given by:

$$A_{KL} = \sum_{e=1}^{M} \iint_{A^e} \left(\frac{\partial N^K}{\partial x}\frac{\partial N^L}{\partial x} + \frac{\partial N^K}{\partial y}\frac{\partial N^L}{\partial y}\right)\nu + N^K\left(\frac{\partial N^i}{\partial y}\psi^i\frac{\partial N^L}{\partial x} - \frac{\partial N^i}{\partial x}\psi^i\frac{\partial N^L}{\partial y}\right)\,\mathrm{d}A^e.$$

Here Green's theorem has been used to reduce the second derivatives occurring in (27) and since ω is known on the boundary the surface integrals have been dropped by the usual arguments.

The type of element required by this formulation is $C^{(0)}$ continuous and at least a third order polynomial, since second derivatives are taken of the shape functions. In this respect the cubic quadrilateral[21] has proven useful and versatile as is demonstrated later.

For transient flow problems again the major part of the computing time consists of obtaining self consistent vorticity and stream function fields, which has to be done at each time step. The iterative procedure starts with an initially prescribed vorticity field obtained from a finite difference approximation to the velocity field using (11). The corresponding initial ψ field is then found from equation (26). In the steady state case, ψ remains fixed on all boundaries, but in the transient case, where the total flow varies in time, the stream function may only be fixed on one boundary or a line of symmetry in the flow field.

The boundary conditions of zero velocity relative to the walls can only be interpreted, therefore, in terms of normal and tangential derivatives of ψ. This precludes the use of simple elements, since it is impossible to specify both these derivatives unless an element is used with first order derivatives included as nodal variables. This is a restriction on the usefulness of the method.

The boundary conditions for the ψ field are found by first fixing ψ at a suitable point within the flow domain or on the boundary. The velocity boundary conditions provide derivative boundary conditions on ψ. On the boundary ψ and the normal and tangential derivatives may all be specified. Clearly there might be considered to be too much information for the solution of Poisson's equation, but in fact, provided that this information is self-consistent, it has been found more accurate to over-specify conditions at the boundary rather than specify the minimum conditions.

Once the ψ field has been determined the boundary vorticity is obtained from equation (29). This interpolated value will be different from the initial vorticity value since the finite element solution of equation (26) averages out singularities in the initial vorticity field such as those associated with sudden acceleration. The vorticities on the rigid boundary are taken to be the average of the initial and interpolated values, while those internally are unaltered. The vorticity boundary conditions on inlet and exit of the fluid can either be obtained in the same way, or under some circumstances an assumption of normal vorticity gradient can be made. Equation (26) is resolved by adjusting the boundary vorticity until a self consistent stream function and vorticity field is obtained, which is normally about three iterations for a 1 per cent tolerance.

The value of vorticity at the next time step is obtained from equation (13). Since the stream function appears in this equation it is possible to use only a forward difference time stepping scheme unless the non-linear terms are ignored.

An alternative to this scheme is possible with repeated iteration between the stream function and vorticity fields. This process is rather involved and will have to be omitted from the discussion. Having obtained the vorticity from (13) it is then possible to evaluate the new stream function field. The above process can be summarised below:

1. Initial vorticity field and velocity $B.C$ prescribed

↓

2. a. Reference value or line for stream function chosen

↓

b. Derivative $B.C$ on ψ obtained.

↓

3. $\nabla^2\psi = -\omega$ Solved for ψ field

↓

4. Interpolated boundary vorticity ω_1 found (29)

↓

5. Boundary vorticity obtained from $\omega_B = \omega_0 + \frac{1}{2}(\omega_1 - \omega_0)$

↓

convergence

↓

6. Vorticity field updated from (13)

↓

7. Finish

where ω_0 = initial or previously iterated vorticity value.

Using the Galerkin method and element interpolation (28) with a forward difference time scheme equation (13) results in

$$[A]\{\omega\} = \{B\}$$

where the $A_{KL}{}^{\text{th}}$ term of the matrix A is

$$A_{KL} = \sum_{e=1}^{M} \iint_{A^e} N^K N^L \, \mathrm{d}A^e$$

$$B_K = \sum_{e=1}^{M} \iint_{A^e} N^K N^j \omega^j - \Delta t \left[N^K \frac{\partial N^i}{\partial x} \psi^i \frac{\partial N^j}{\partial y} \omega^j - N^K \frac{\partial N^i}{\partial y} \psi^i \frac{\partial N^j}{\partial x} \omega^j \right.$$
$$\left. + \nu \left(\frac{\partial N^K}{\partial x} \frac{\partial N^i}{\partial x} \omega^i + \frac{\partial N^K}{\partial y} \frac{\partial N^i}{\partial y} \omega^i \right) \right] \mathrm{d}A^e.$$

The terms ω^i and ψ^i in the B vector are the values at the beginning of each time step while the column vector $\{\omega\}$ contains the vorticities at the end of the time steps.

4.3. *Stream function formulation*

For the sake of completeness mention must be made of an alternative method which is applicable to creeping flow problems. This approach uses equation (14) as its basis, and the field variable of stream function alone. Probably the first examples using this method were by [23, 24] who considered the solution of the biharmonic equation governing creeping flow and a variational approach. The most practical advantage of the approach is that the programs for solution of plate bending problems and creeping flow equations are identical, and so existing programs can be used[25, 31]. The number of applications of creeping flow are limited however, and the formulation using the Galerkin method[26] rather than the functional approach has application to higher Reynolds number flows. The functional approach itself can be modified to cover the case of certain non-Newtonian fluids using the method of[27], and this has been considered in connection with a finite element solution[28].

Practical solutions to problems using equation (14) are now available[43], but it is clear that the main disadvantage of the method is the requirement for high order elements, while the advantage is that only one equation instead of three describes the fluid behaviour.

5. NUMERICAL EXAMPLES

5.1. *Introduction*

There are several idealised flow situations for which there exist analytic solutions[20]. These are used as a basis for a test of the accuracy of the finite element approach. The effectiveness of the method is subsequently demonstrated by obtaining the solution to some separated flow problems for which there is no analytic solution. Obviously space does not permit the coverage of individual problems in great detail, but it is anticipated that a complete discussion will be available shortly[35].

The units used in the following problems have been omitted since any set of units consistent with the governing equations can be inserted. It was also considered useful to give an idea of computation time for each problem. The machine on which most of the problems were run was an ICL 1905E, the remainder being run on the Science Research Council Atlas Machine. In fact times will refer to computing time on the ICL machine.

Several forms for displaying the results were considered, and it was decided that for some problems a stream function and vorticity plot was preferable to a velocity vector plot. This being the case, it was necessary to develop a separate program to interpret the velocity distribution, produced by the finite element program, into compatible stream function and vorticity fields. The procedure adopted is described in Ref. [35, 44] where some details of the automatic contour plotting program are also given.

5.2. *Couette flow*

Consider the fully developed plane parallel flow, between infinite parallel plates, in the x-direction, Fig. 2, under a pressure gradient $\frac{\partial p}{\partial x}$. One plane moves with relative velocity U. With the boundary conditions $u\left(x, -\frac{h}{2}\right) = 0$, $u\left(x, \frac{h}{2}\right) = U$ the flow at any point

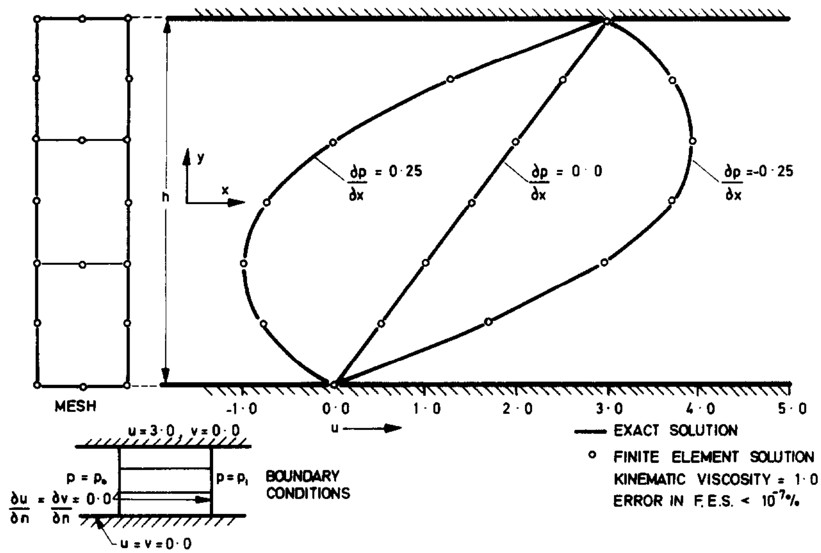

Fig. 2. Couette flow between flat plates.

is given by:

$$u(x, y) = \frac{U}{h}\left(y + \frac{h}{2}\right) - \frac{1}{2\mu}\frac{\partial p}{\partial x}\left(\frac{h^2}{4} - y^2\right)$$

where u, v are the x, y components, respectively, of velocity. Using the velocity/pressure approach the boundary conditions are the same as above. Since the flow is plane and parallel the appropriate governing equations are (20, 21) with zero normal velocity gradients on the inlet and outlet boundaries. The mesh, using parabolic isoparametric elements[21] of dimension 2×2 units, is illustrated in Fig. 2.

Both the density and kinematic viscosity of the fluid were taken as unity for this problem. For all subsequent examples the density is also unity. The error in the numerical solution was less than 10^{-7} per cent which is gratifyingly low. Other examples were tried in the range of Reynolds numbers 0–10^5 and all displayed corresponding accuracy. Execution time on the computer was about 30 sec for each example.

The second method involved the stream function and vorticity approach based on equations (26) and (27). Boundary conditions on ψ are shown in Fig. 3(b), and for the particular test problem "A" was chosen to be 2·25 units, "h" was 6 units and the viscosity 0·1 units. The mesh discretisation, which uses cubic quadrilateral isoparametric elements is illustrated in Fig. 3(a). The double row of elements merely served as a check that the method gave the correct vorticity distribution at the centre nodes. The iterative procedure described earlier was used to obtain the vorticity and stream function fields. To obtain an accuracy of better than 10^{-5} per cent error in solution required an execution time of 1000 sec, which is significantly greater than the velocity and pressure formulation used.

Poiseulle flow can be studied using an axisymmetric form of Navier–Stokes equations. These can be placed in a finite element form, analogous to the equations (20, 21) in cartesian coordinates. Since the equations in polar coordinates are very similar to equations

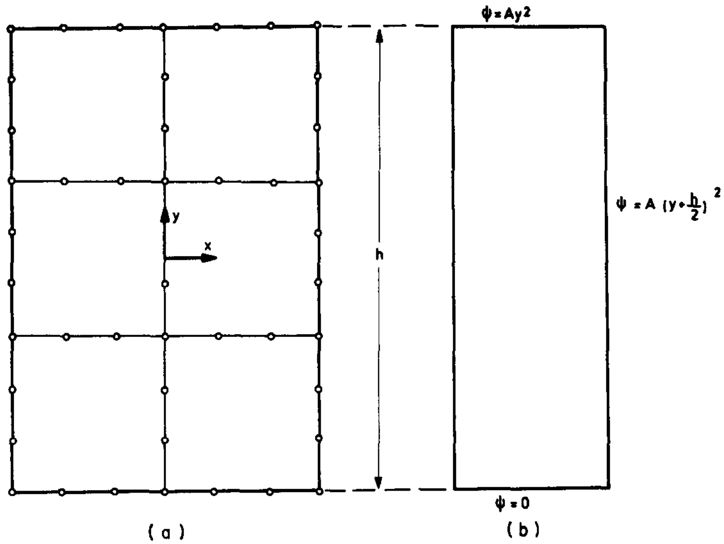

Fig. 3. Couette flow stream function/vorticity formulation. (a) Mesh discretisation. (b) Boundary conditions.

(20, 21), they have been omitted. This approach has been used to obtain solutions to plane parallel pipe flow (Poiseulle Flow) for various Reynolds numbers. The particular example illustrated in Fig. 4 is for a pressure gradient of $-1/2$ and kinematic viscosity of 1·0. The error of less than 10^{-7} per cent was again recorded, when compared with the analytic solution. Computation time was about 30 sec for this problem.

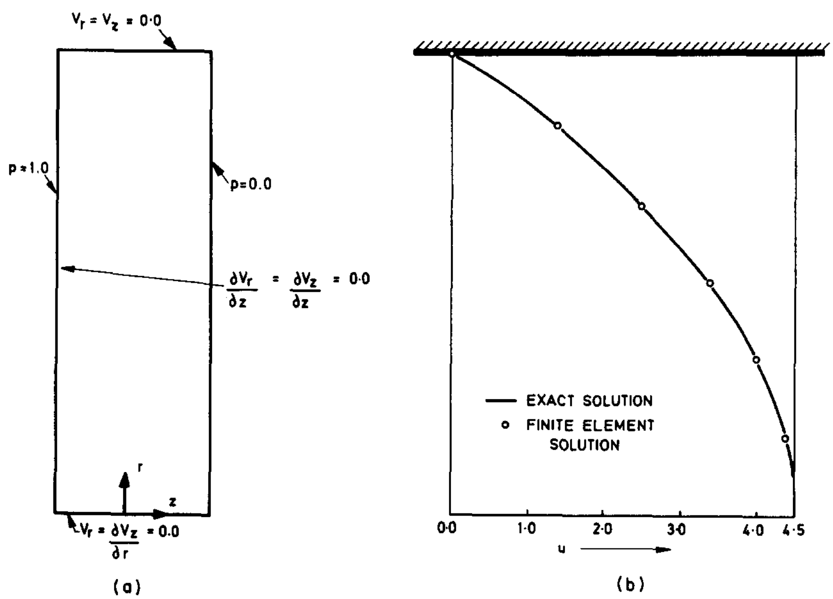

Fig. 4. (a) Boundary conditions. (b) Velocity profile–Poiseulle flow.

5.3. *Couette flow development*

The transient development of Couette flow between parallel plates, generated for example by sudden acceleration of one plane, provides an example to test the accuracy of the transient programs. Suppose initially the fluid is at rest under zero pressure gradient, and one wall is suddenly accelerated to a velocity U. The fluid will gradually gain momentum until the uniform velocity profile in Fig. 2 is reached.

Using the velocity and pressure formulation the problem has been studied with the simple mesh in Fig. 2. The results and physical conditions are illustrated in Fig. 5, where the high accuracy of the method is apparent. Typical computation time was seven seconds per time step.

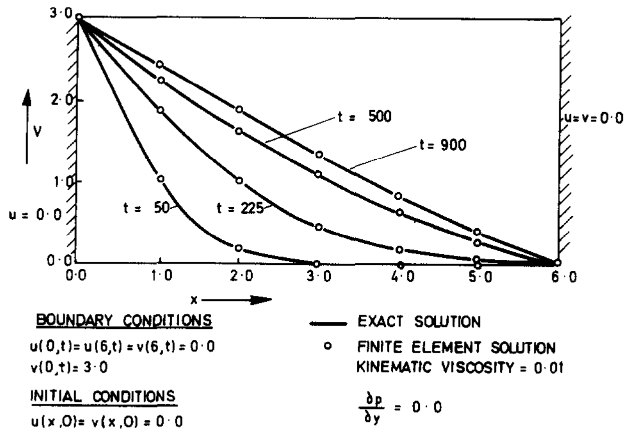

Fig. 5. Transient problem–couette flow formation

The stream function and vorticity approach was used to study the same problem. The element used was a plate bending type element with four nodal degrees of freedom

$$\psi, \frac{\partial \psi}{\partial x}, \frac{\partial \psi}{\partial y}, \frac{\partial^2 \psi}{\partial x \partial y}$$

at each of the four corners of a rectangle[36]. The mesh and boundary conditions used are shown in Fig. 6. The vorticity field was initially zero everywhere except on the moving boundary, and the iterative procedure to obtain stream function and vorticity fields was described earlier. Using two iterations per time cycle gave an accuracy tolerance of 5 per cent for the relative change of stream function within each cycle. In other words if ψ_{new} is the latest iterated value and ψ_{old} is the previous one,

$$\left| \frac{\psi_{\text{new}} - \psi_{\text{old}}}{\psi_{\text{new}}} \right| < 5 \text{ per cent.}$$

Computation time with this accuracy tolerance was about 50 sec per time step.

A comparison of the accuracy of the two methods used, is shown in Tables 1 and 2. For this test a time step of one time unit was used, the kinematic viscosity was 0·1 and the channel width h was six units. The velocity of the moving wall was three units after zero time. The analytic solution[20] is obtained from an error function series expansion. The series was summed on the computer, and it is not possible to obtain

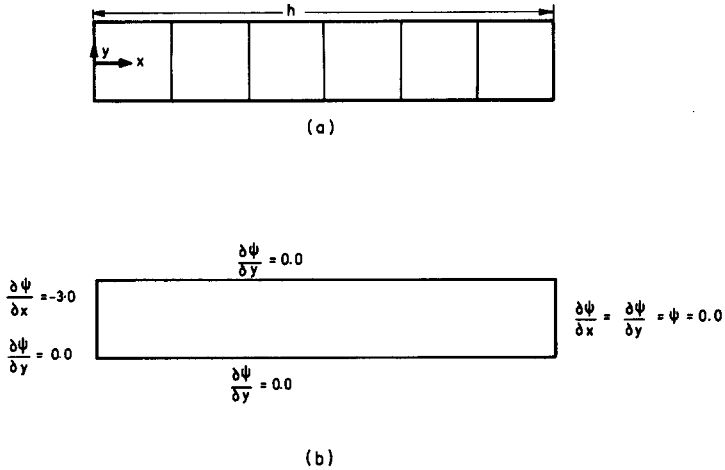

Fig. 6. Couette flow development–stream function/vorticity method. (a) Mesh discretisation. (b) Boundary conditions.

very accurate answers near the origin of the error function. For this reason these are omitted from the Tables. The two times chosen are fairly representative of the two phases in transient problems. The first time is at the initial instigation of motion, when errors are comparatively large. The second time is at a point when the finite element solutions start to approximate actual behaviour very accurately. In order to give some idea of the time scale involved the flow reaches within 0·5 per cent of the steady value after 100 time units.

Table 1. Couette flow development at 4·0 time units

Distance (x)	Analytic solution	Finite element solution: Velocity/pres.	Finite element solution: S.F/vorticity
0·0	3·0000	3·0000	3·0000
1·0	0·7905	0·9259	0·9921
2·0	0·0708	0·0352	0·1275
3·0	—	−0·0254	0·0096
4·0	—	0·0078	0·0006
5·0	—	−0·0003	−0·0003
6·0	0·0000	0·0000	0·0000

Table 2. Couette flow development at 40·0 time units

Distance (x)	Analytic solution	Finite element solution: Velocity/pres.	Finite element solution: S.F./vorticity
0·0	3·0000	3·0000	3·0000
1·0	2·1710	2·1713	2·3937
2·0	1·4385	1·4404	1·6760
3·0	0·8665	0·8646	0·9788
4·0	0·4719	0·4583	0·4229
5·0	—	0·1914	0·0881
6·0	0·0000	0·0000	0·0000

From a comparison of the accuracy of the two methods discussed it is fairly obvious that the velocity pressure formulation is far more efficient for a given amount of computer time than the stream function vorticity formulation. For this reason no further examples will be included on the stream function and vorticity method. Attention will now be given to practical problems for which there are no analytic solutions.

5.4. *Slider bearing*

The geometry of a typical slider bearing is shown in Fig. 7. The lower plane BC moves with a velocity u_0 relative to the upper plane AD. Some assumption has to be made about the velocity gradients or stress distribution at inlet and outlet boundaries. The assumption was, in one case, that the normal derivatives of velocity were zero, and in the other case that the stresses were zero. The pressure is also assumed to be zero at either end of the bearing. The viscosity was 0·0008 units, $l = 0{\cdot}36$ units, $h_1 = 0{\cdot}0008$ units, $h_2 = 0{\cdot}0004$ units, and $u_0 = 30{\cdot}0$ units. The resulting pressure profile is shown in Fig. 8, and shows the general parabolic trend in the distribution of pressures. The difference between the results of the two cases of zero stress or velocity gradient was in

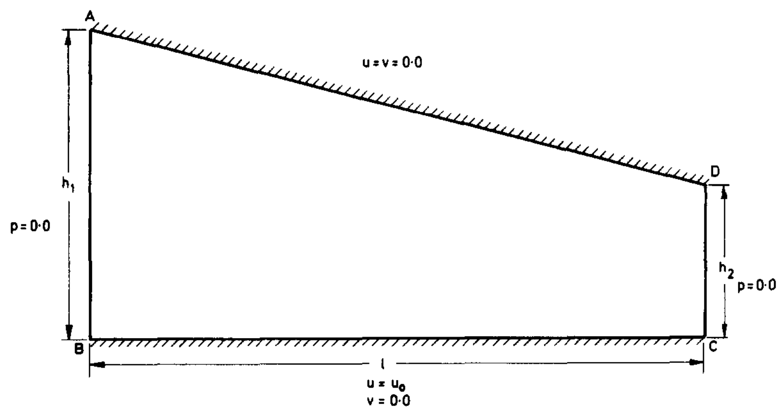

Fig. 7. Lubrication in a slider bearing-boundary data.

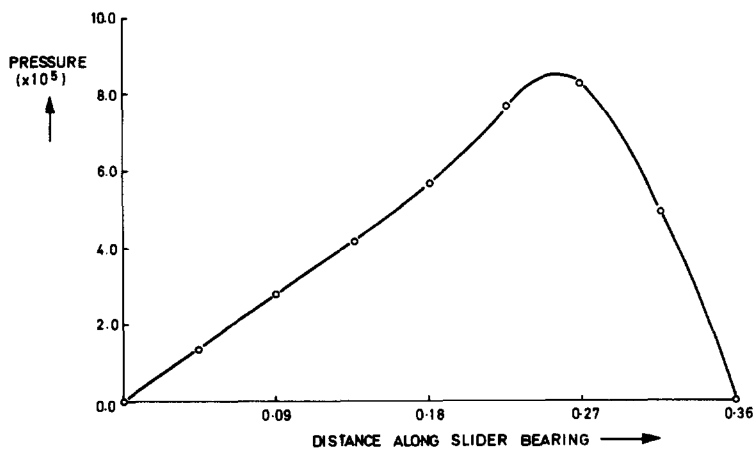

Fig. 8. Pressure on a slider bearing face.

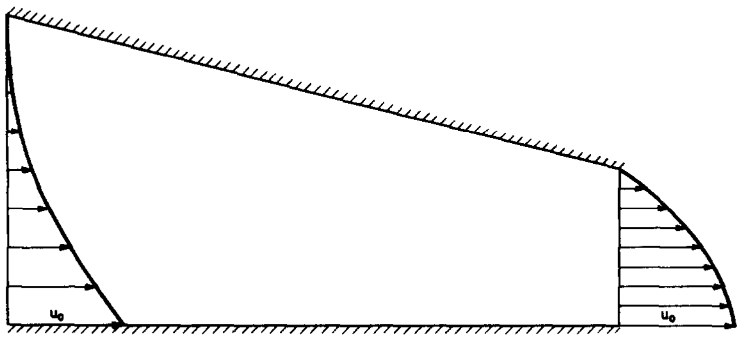

Fig. 9. Velocity vector plot for slider bearing problem.

fact less than 1 per cent. The velocity vectors at the extreme boundaries are illustrated in Fig. 9. Assuming a zero initial velocity distribution three iterations for the non-linear terms in equations (20, 21) were necessary to achieve a tolerance of 1 per cent. The total computation time was 450 sec.

5.5. *Flow past a square cavity*

Flow round a square cavity, which is the first problem considered where separation occurs, was studied for Reynolds numbers of 1, 100, 400 and 600 respectively. The results can be applied to groundwater problems where soil voids are assumed to be rectangular[32]. Although with the finite element method there is no reason why a better approximation to a soil void could not be made. It also has applications in connection with polymer extruders, paper production, and tooth gears.

The case of Reynolds number 600 was used to test the effect of elongating entrance and exit regions on the flow pattern. The boundary conditions for this case are shown in Fig. 10 and the velocity vector plot in Fig. 11. The long entrance and exit in fact produced no significant alteration in flow behavior.

For the other examples, in this category, the mesh and boundary conditions used are illustrated in Fig. 12. The stream function and vorticity plots are shown in Fig. 13–18 and were obtained using the automatic contour plotting program.

The results of the program compare well with existing finite difference solutions[33, 34]. A detailed comparison will be made in a further publication[35]. Computation time was about 2000 sec for a Reynolds number 600.

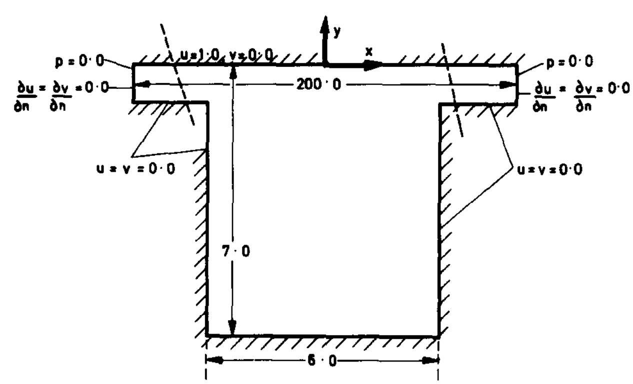

Fig. 10. Flow past a cavity boundary data.

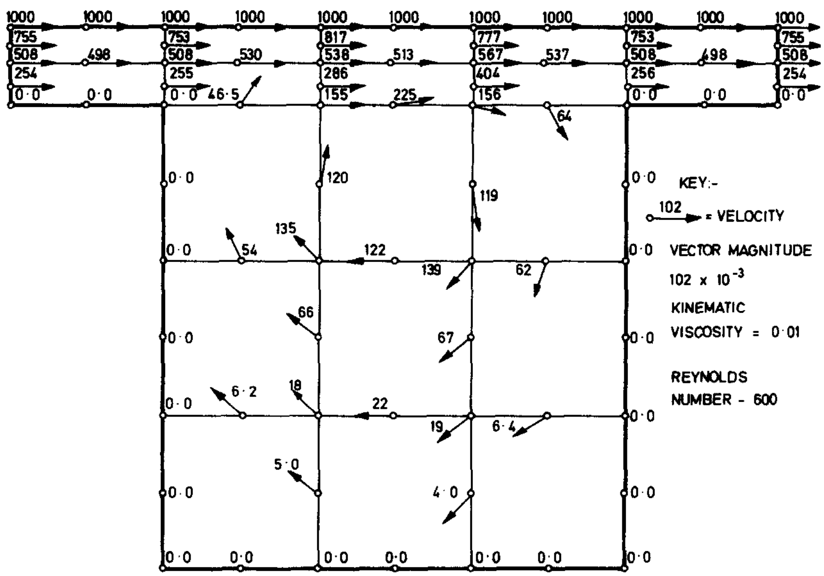

Fig. 11. Viscous incompressible flow past a cavity.

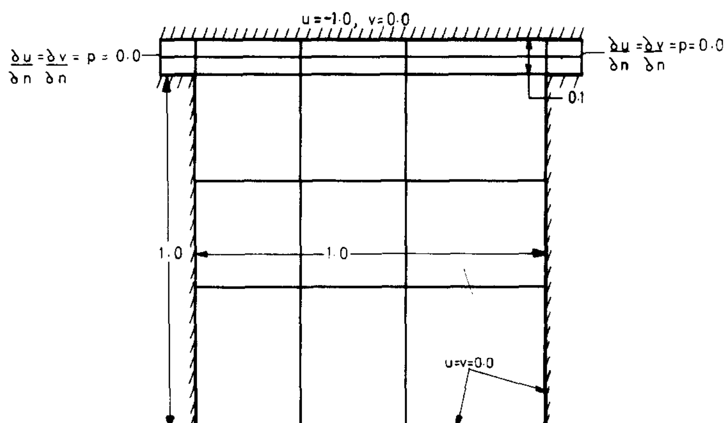

Fig. 12. Boundary data for the unit cavity.

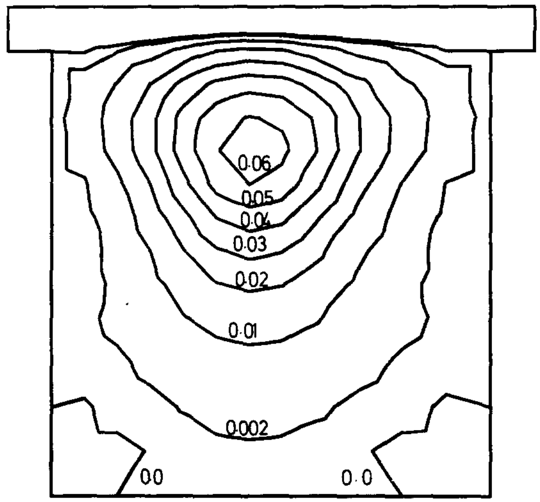

Fig. 13. Re = 1, streamlines.

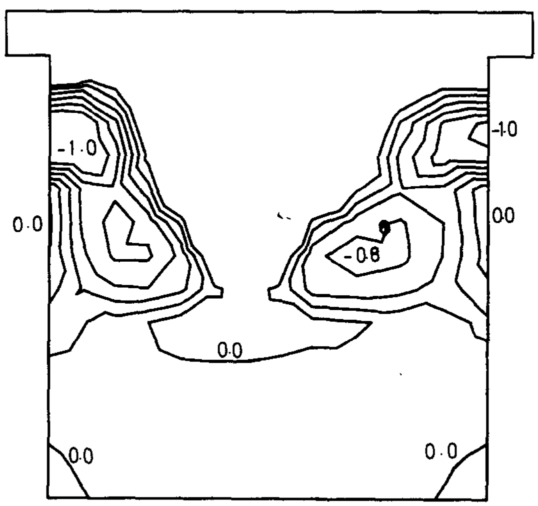

Fig. 14. Re = 1, vorticity.

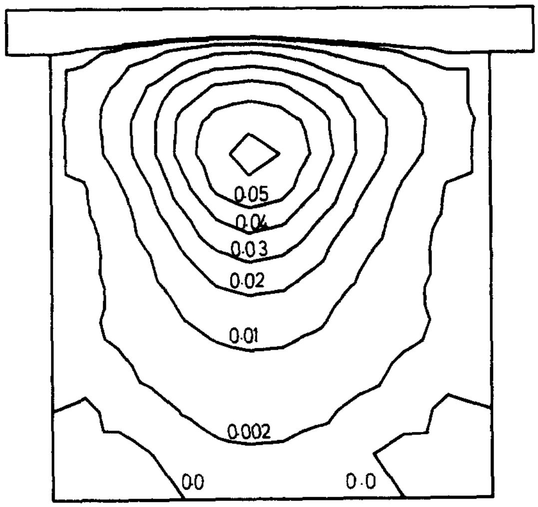

Fig. 15. Re = 100, streamlines.

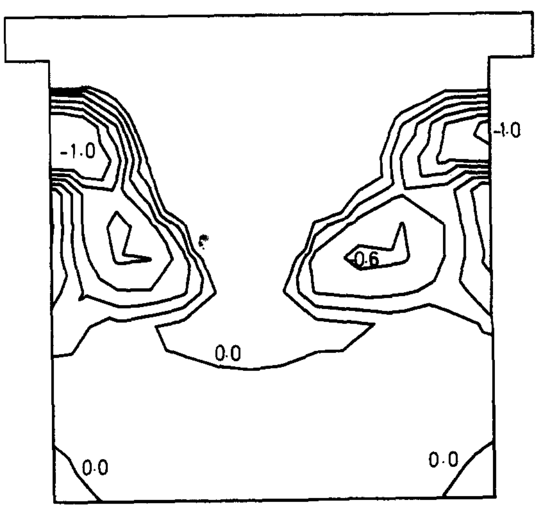

Fig. 16. Re = 100, vorticity.

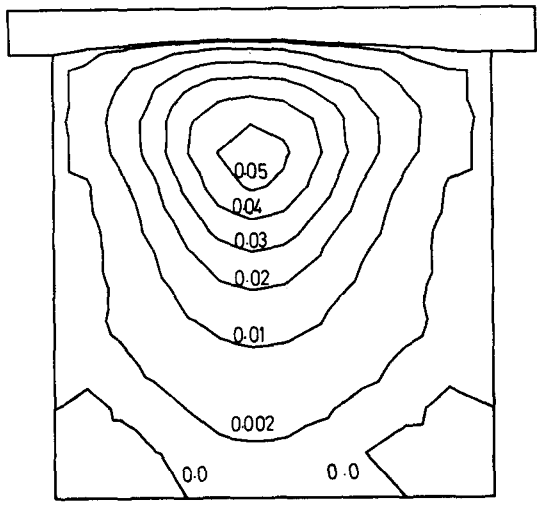

Fig. 17. Re = 400, streamlines.

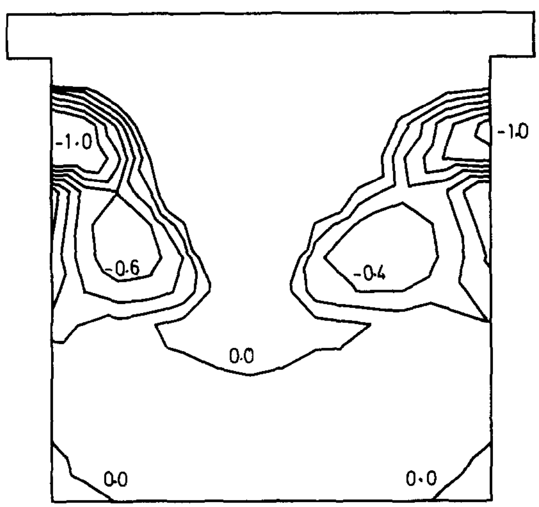

Fig. 18. Re = 400, vorticity.

5.6. *Flow past a cylinder*

The flow past a cylinder, perhaps the best documented[37–42] of all problems, provides a very useful practical model. The mesh used for the study is illustrated in Fig. 19 and boundary conditions in Fig. 20. For this example, only the upper half-plane is considered, although some flow-asymmetry may be expected to arise at Reynolds numbers approaching 100. The flow was analysed for Reynolds numbers of 5, 7, 10, 20, 30, 40, 50, 100 and the

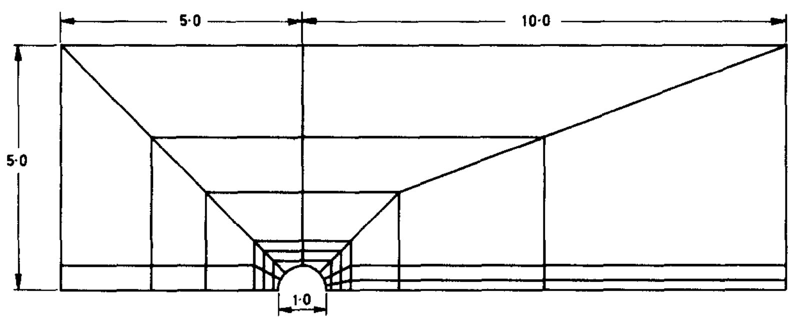

Fig. 19. Mesh used for flow round a cylinder.

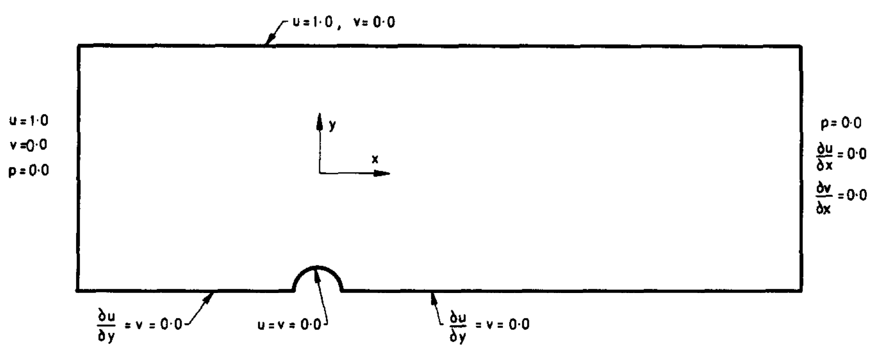

Fig. 20. Boundary conditions–flow round a cylinder.

results are shown in Figs 21–28, in terms of stream functions. Rather than adjust the velocity for each problem the kinematic viscosity was successively increased, the results from one program being the initial values for the next. This procedure considerably speeds convergence of the higher Reynolds number flows, although even for Reynolds number of 100 it is possible to start the iteration for the non-linear terms with a zero velocity field. In such a case the computation time is about 6000 sec. However, using the previously converged values corresponding to a lower viscosity this time was reduced to about 1500 sec.

Fig. 21. Re = 5.

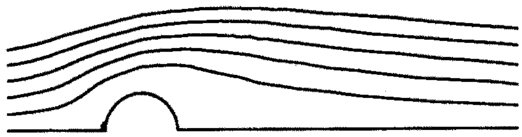

Fig. 22. Re = 7.

Fig. 23. Re = 10.

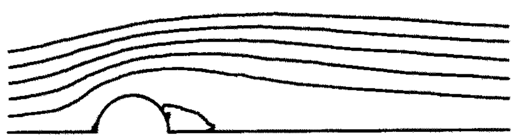

Fig. 24. Re = 20.

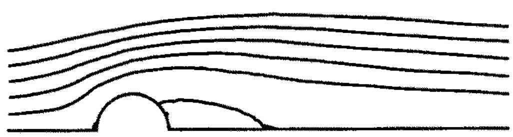

Fig. 25. Re = 30.

Fig. 26. Re = 40.

Fig. 27. Re = 50.

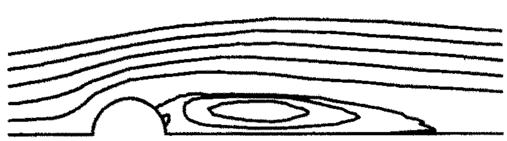

Fig. 28. Re = 100.

6. CONCLUSIONS

In this paper some of the concepts relating to finite element formulation of the Navier–Stokes equations have been discussed. Of the two methods introduced the velocity-pressure formulation proved to be the more viable. The advantage of the method, over earlier finite element formulations, is most certainly the fact that the pressure boundary conditions can be specified directly without the need to assume a "reasonable velocity field" initially. For free surface problems the choice of other variables can cause immediate difficulties. The method also has the advantages that the value of stress or normal velocity gradients on the boundary can be incorporated into the equation matrix very easily.

In addition, the velocity–pressure formulation facilitates the use of the method in three dimensions. The use of stream function for example requires the use of plate bending type elements, and consequently cannot easily be extended to three dimensional work. With the present method, since isoparametric elements are used, three dimensional analysis produces no more difficulty than two dimensional case. Furthermore, curved boundaries can be defined more efficiently by isoparametric elements than any other existing element.

Obviously for conservation of space, a great deal of material had to be omitted. For instance a fuller discussion of axisymmetric and transient problems, as well as a comparison of iteration schemes (in particular the Newton Raphson method) have to be left as a subject for a further paper.

ACKNOWLEDGEMENTS—The authors would like to acknowledge the encouragement of Professor O. C. Zienkiewicz, Mr. B. M. Irons, Dr. R. F. Allen and Mr. J. Davis, and the financial assistance of the S. R. C. in the form of a research studentship.

REFERENCES

1. J. T. Oden, The finite element method in fluid mechanics, Lecture for NATO Advanced Study Institute on Finite Element Methods in Continuum Mechanics, Lisbon (1971).
2. P. Hood, A finite element solution of the Navier–Stokes equations for incompressible contained flow, M. Sc. Thesis, University of Wales, Swansea, (1970).
3. S. I. Cheng, Numerical integration of Navier–Stokes equations, *AIAA Jl*, **8**, 2115–2122 (1970).
4. O. C. Zienkiewicz, and Y. K. Cheung, Finite elements in the solution of field problems, *The Engineer*, **220**, 507–510 (1965).
5. G. de Vries, and D. H. Norrie, The application of the finite element technique to potential flow problems, *Trans. ASME appl. Mech. Div., Paper No.* 71-APM-22, 798–802 (1971).
6. J. Ergatoudis, B. M. Irons, and O. C. Zienkiewicz, Curved isoparametric, quadrilateral elements for finite element analysis, *Int. J. Solids Struct.*, **4**, 31–42 (1968).
7. B. M. Irons, A conforming quartic triangular element for plate bending, *Int. J. num. Meth. Engng.* **1**, 29–45 (1969).
8. A. Zenisek, Interpolation polynomials on the triangle, *Numer. Math.*, **15**, 283–296 (1970).
9. M. J. L. Hussey, R. W. Thatcher, and M. J. M. Bernal, On the construction and use of finite elements, *J. Inst. Maths. Applics.*, **6**, 263–282 (1970).
10. O. C. Zienkiewicz, *The Finite Element Method in Engineering Science*. McGraw-Hill, New York, (1971).
11. J. T. Oden, A general theory of finite elements II Applications, *Int. J. num. Meth. Engng*, **1**, 247–259 (1969).
12. P. Tong, The finite element method for fluid flow, Paper US5-4, *Japan–U.S. Seminar on Matrix Methods in Structural Analysis and Design* (Edited by R. H. Gallagher). University of Alabama Press (1970).
13. B. A. Finlayson, and L. E. Scriven, The method of weighted residuals and its relation to certain variational principles for the analysis of transport processes, *Chem. Engng Sci.*, **20**, 395–404 (1965).
14. B. A. Finlayson, and L. E. Scriven, The method of weighted residuals—a review, *Appl. Mech. Rev.*, **19**, 735–748 (1966).
15. B. A. Finlayson, and L. E. Scriven, On the search for variational principles, *Int. J. Heat Mass Transfer*, **10**, 799–821 (1967).

16. J. Davis, and P. Hood, Finite element formulation with reference to fluid dynamics, To be published.
17. A. J. Baker, Finite element theory for viscous fluid dynamics, Rep. No. 9500-920189, Bell Aerospace Company (1970).
18. A. J. Baker, Finite element computational theory for three dimensional boundary layer flow, AIAA Reprint No. 72-108, Presented at the *AIAA 10th Aerospace Sci. Mtg*, San Diego, California (1972).
19. J. Davis, and C. Taylor, Finite element solution of the tidal hydraulics equations, To be published.
20. H. Schlichting, *Boundary Layer Theory*. McGraw-Hill, New York (1960).
21. R. L. Taylor, On completeness of shape functions for finite element analysis, *Int. J. num. Meth. Engng*, **4**, 17–22 (1972).
22. E. Spreeuw, Fiesta: finite elements stress and temperature analysis, *Reactor Centrum Nederland Rep.* RCN-149 (1971).
23. C. C. H. Card, Ph. D. Thesis, University of Wales, Swansea (1968).
24. B. Atkinson, C. C. H. Card, and B. M. Irons, Application of the finite element method to creeping flow problems, *Trans. Instn Chem. Engrs*, **48**, T276–T284 (1970).
25. B. Atkinson, M. P. Brocklebank, C. C. H. Card, and J. M. Smith, Low Reynolds number developing flows, *A.I.Ch.E. Jl*, **15**, 548–553 (1969).
26. O. C. Zienkiewicz, and C. Taylor, Weighted residual processes in F. E. M. with particular reference to some coupled and transient problems, Lecture for NATO Advanced Study Institute on Finite Element Methods in Continuum Mechanics, Lisbon (1971).
27. R. B. Bird, New variational principle for incompressible non-Newtonian flow, *Phys. Fluids*, **3**, 539–541 (1960).
28. B. Atkinson, Private compunication, University of Wales, Swansea (1972).
29. F. Kikuchi, and Y. Ando, A finite element method for initial value problems, *Proc. 3rd Conf. on Matrix Methods in Structural Mechanics*, Wright Patterson Air Force Base, Ohio (1971).
30. O. C. Zienkiewicz, and C. J. Parekh, Transient field problems: two dimensional and three dimensional analysis by isoparametric finite elements, *Int. J. num. Meth. Engng*, **2**, 61–71 (1970).
31. P. Tong, and Y. C. Fung, Slow particulate viscous flow in channels and tubes-application to biomechanics, *Trans. ASME appl. Mech. Div., Paper No.* 71-APM-R, 721-728 (1971).
32. K. P. Stark, A numerical study of the non-linear laminar regime of flow in an idealised porous medium, International Association for Hydraulic Research Symposium on the Fundamentals of Transport Phenomena in Porous Media, Hafia, Israel (1969).
33. R. D. Mills, Numerical solution of the viscous flow equations for a class of closed flows, *Jl R. aeronaut. Soc.*, **69**, 714–718 (1965).
34. O. R. Burggraf, Analytic and numerical studies of the structure of steady separated flows, *J. Fluid Mech.*, **24**, 113–151 (1966).
35. P. Hood, Ph. D. Thesis, University of Wales, Swansea. To be submitted.
36. F. K. Bogner, R. L. Fox, and L. A. Schmit, The generation of interelement compatible stiffness and mass matrices by the use of interpolation formulas, The Conference of Matrix Methods in Structural Mechanics, Wright Patterson Air Force Base, Ohio (1965).
37. S. Taneda, Experimental investigation of the wakes behind cylinders and plates at low Reynolds numbers, *J. phys. Soc. Japan*, **11**, 302–307 (1956).
38. M. Kawaguti, and P. Jain, Numerical study of a viscous fluid flow past a circular cylinder, *J. phys. Soc. Japan*, **21**, 2055–2062 (1966).
39. S. C. R. Dennis, and G. Z. Chang, Numerical solutions for steady flow past a circular cylinder at Reynolds numbers up to 100. *J. Fluid Mech.*, **42**, 471–489 (1970).
40. H. Takami, and H. B. Keller, Steady two-dimensional viscous flow of an incompressible fluid past a circular cylinder, *Phys. Fluids, Suppl. II*, II-51-II-56 (1969).
41. P. C. Jain and K. S. Rao, Numerical solution of unsteady viscous incompressible flow past a circular cylinder, *Phys. Fluids, Suppl. II*, II-57-II-64 (1969).
42. D. C. Thoman, and A. A. Szewczyk, Time dependent viscous flow over a circular cylinder, *Phys. Fluids, Suppl. II*, II-76-II-86 (1969).
43. M. D. Olson, A variational finite element method for two-dimensional steady viscous flows, McGill University Engineering Institute of Canada, *Conf. on Finite Element Methods in Civil Engineering*, McGill University, Montreal, Quebec (1972).
44. D. Kan, Mesh and Contour plot for triangle and isoparametric elements, University of Wales, Swansea, Dept. Civil Engineering, Computer Rep. No. CNME/CR/39, (1970).

APPENDIX

Time marching schemes

The main time stepping techniques applicable to the finite element method are the finite difference schemes, finite elements in time, or the Runge Kutte method. The stability of time elements is of the same order or better than a conventional mid-difference scheme, but the accuracy is generally slightly inferior showing damping effects[16]. However, for linear equations, time elements can save up to 50 per cent in computer time per time cycle when there are no right hand sides to reform at each time step.

Finite difference schemes in time are well documented e.g.[29] and for most purposes mid-difference schemes are the best. Forward differences are sometimes useful for development work particularly where non-linear terms can be avoided by its use, but the method is too unstable for any general application. Backward difference methods are the most stable but less accurate than mid-difference (except for the initial time steps when the reverse is sometimes true). A mid-difference method seems to be the most viable all round scheme of the three and is equivalent in accuracy and efficiency to the Runge–Kutte method. The types of mid-difference scheme which the author uses is a trapezoidal rule. Reference to the one dimensional diffusion equation[30] should make the application of the trapezoidal rule clear.

The equation:

$$\frac{\partial \phi}{\partial t} = \frac{\partial^2 \phi}{\partial x^2} \tag{31}$$

with suitable initial and boundary conditions is considered at an instant t in time. The assumption is made that

$$\left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t+\Delta t/2} = \frac{1}{2}\left(\left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t+\Delta t} + \left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t}\right).$$

Equation (31) becomes

$$\begin{aligned} \frac{\phi|_{t+\Delta t} - \phi|_t}{\Delta t} &= \left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t+\Delta t/2} \\ &= \frac{1}{2}\left(\left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t+\Delta t} + \left.\frac{\partial^2 \phi}{\partial x^2}\right|_{t}\right). \end{aligned}$$

Substitution of the element trial functions[1] using the Galerkin method gives

$$[A]\{\phi\}_{t+\Delta t} = [B]\{\phi\}_t \tag{32}$$

where

$$A_{KL} = \sum_{e=1}^{M} \int_{x^e} \frac{N^K N^L}{\Delta t} - \frac{N^K}{2}\frac{\partial^2 N^L}{\partial x^2}\, \mathrm{d}x^e$$

$$B_{KL} = \sum_{e=1}^{M} \int_{x^e} \frac{N^K N^L}{\Delta t} + \frac{N^K}{2}\frac{\partial^2 N^L}{\partial x^2}\, \mathrm{d}x^e.$$

Integration by parts would normally be used to reduce the order of inter-element continuity required for the formation of the A and B matrices. The right hand side is known from the initial conditions or the previous time step. The procedure is identical in principle for non-linear equations except that some form of iteration is required to find the value of the non-linear terms at time $t + \Delta t$.