INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 6, 749–761 (1986)

USE OF PSEUDO-CONCENTRATIONS TO FOLLOW CREEPING VISCOUS FLOWS DURING TRANSIENT ANALYSIS

ERIK THOMPSON

Department of Civil Engineering, Colorado State University, Fort Collins, Colorado 80523, U.S.A.

SUMMARY

Creeping viscous flows are followed through finite element meshes by use of pseudo-concentrations which define material position. The concentrations, assumed to be transported only by convection, serve as material markers. Illustrations are presented related to industrial forming processes and the slow deformation of geological structures.

KEY WORDS Finite Element Creeping Flow Forming Processes

INTRODUCTION

The versatility offered by the finite element method for adjusting nodal point co-ordinates during an analysis has made it an ideal tool for following creeping viscous and viscoplastic flows through finite deformations.[1–5] To do so, it is only necessary to determine the instantaneous velocity field for a given position and then increment each nodal point co-ordinate to obtain the new position of the mesh and material. Although the method has been successfully applied to a wide range of problems, it has the disadvantage that the mesh often becomes so distorted that continued analysis is no longer possible. When this occurs the analyst must define a new mesh and interpolate new nodal point values.

The procedure presented in this paper assigns a pseudo-concentration throughout the mesh in such a manner that its value indicates the presence or absence of the creeping viscous material. In regions where the material is present, the appropriate effective viscosity is used during the assembly of the stiffness matrix. In those regions of the mesh where the value of the concentration indicates that the material has not yet penetrated, an artificially low value of viscosity is used so as not to affect the velocity of the higher viscosity material. The velocity field thus determined, the pseudo-concentration is allowed to follow the fluid flow through the next increment of time by setting its material derivative equal to zero.

Using concentrations to follow the movement of an interface between two immiscible fluids is well established, particularly in the simulation of fluid flows associated with oil reservoirs.[6] However, in these applications the concentrations are real, and they often develop steep fronts which cause numerical instabilities. For the method presented in this paper, the concentrations are fictitious and can be defined as smooth functions, thus making it possible to follow their movement without the difficulties encountered when simulating real concentrations. In the event that a pseudo-concentration does develop a steep front during the course of an analysis, a new, smoother

0271–2091/86/100749–13\$06.50
© 1986 by John Wiley & Sons, Ltd.

Received 30 September 1985
Revised 15 April 1986

concentration field can be assigned, provided that it defines the same location to the material. This usually means preserving a critical contour which separates the actual material from the artificial material. A procedure for automatically redefining a concentration field while preserving the location of a given contour has been developed and is illustrated later in the paper.

PROCEDURE

Governing equations

The governing equations for the slow (creeping) motion of both the real and the artificial fluids are;

momentum:

$$\frac{\partial \sigma_{ij}}{\partial x_j} + X_i = 0, \tag{1}$$

incompressibility:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{2}$$

stress–deformation-rate relationship:

$$\sigma_{ij} = 2\mu d_{ij} - p\delta_{ij}, \tag{3}$$

velocity–deformation-rate relationship:

$$d_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right), \tag{4}$$

boundary conditions:

$$\sigma_{ij} n_j = T_i^*, \quad \text{on } S_T, \tag{5}$$

$$u_i = u_i^*, \quad \text{on } S_u, \tag{6}$$

where σ_{ij} is the stress tensor, p is the pressure, X_i is the body force per unit volume, u_i is the velocity, μ is the viscosity, d_{ij} is the deformation-rate tensor, n_i is the unit outward normal vector to the surface S, u_i^* is the specified velocity on S_u and T_i^* is the specified surface traction on S_T.

The governing equation for the pseudo-concentration, M, is obtained by setting its material derivative equal to zero. Hence

$$\frac{\partial M}{\partial t} + u_i \frac{\partial M}{\partial x_i} = 0 \tag{7}$$

Because the above equation is of first order, boundary values for M should be specified only along that part of the boundary where the fluid enters the control volume (7). If M is not specified, the material will enter the control volume with a gradient normal to the boundary currently specified. For completely enclosed regions, boundary values for M need not to be specified.

Solutions of the above equations by the finite element method are well documented in the literature. Because the algorithm being described involves nothing more than repeated use of these solution techniques, details are not presented here. The interested reader is referred to References 7 and 8.

The artificial fluid and free surfaces

In order to model the flow of a creeping viscous fluid using a pseudo-concentration, it is necessary to define the concentration throughout the entire mesh and have it transported by an artificial fluid in regions where the real fluid is not present. Implicit in the procedure is the assumption that it is possible to specify a viscosity for the artificial material low enough to ensure that its presence has no effect on the motion of the real fluid. It is, perhaps, obvious that this would be true under certain actual conditions, e.g. the atmosphere adjacent to the earth's crust as the latter slowly creeps over periods of geological time. However, if too low a value is specified for the viscosity of the artificial material, the finite element equations become ill-conditioned. Numerical tests were run which compared results obtained from two different values for the viscosity of the artificial fluid. The viscosities used were two and three orders of magnitude smaller than the viscosity of the real fluid. For these tests there was less than half a per cent difference in the calculated nodal point forces and no noticeable differences in the resulting flow patterns. Such tests should be made when applying the technique to specific problems.

The simulated free surface between the artificial fluid and the real fluid is assumed to be free from any surface traction. Work is under way to incorporate a procedure which would make it possible to assign arbitrary surface tractions on such surfaces. The algorithm will first identify the elements through which the surface passes and then calculate the equivalent nodal forces.

Smoothing

The need for a smooth function to represent the pseudo-concentration is important to ensure stability of the finite element equations. In this context, we refer to a smooth function as one which has a non-zero gradient, but with near zero values for all derivatives higher than the first. It is usually a simple matter to define an initially smooth concentration. However, as the flow progresses, the surface which this function represents can become distorted and develop regions having large curvature. Evaluation of the material derivative, therefore, becomes less accurate and instabilities usually ensue. When this occurs the analysis has reached a point similar to that in the incremental mesh formulation where it is necessary to re-mesh. However this usually occurs much further into the analysis than the point where re-meshing would be required, and is much easier to fix.

Because a proper specification for M requires nothing more than the ability to determine on which side of a critical contour a point lies, any two M-surfaces which have the same location for such a contour will both serve this purpose. Therefore, if an M-surface were to become so distorted as to prevent continued analysis, it would only be necessary to replace it with a smoother surface having the same location for the critical contour. Because the mesh would not have been altered by this procedure, all nodal values for other variables, such as temperature, would be left unaffected.

Several computer codes were developed and tested for automatically smoothing the M-surface while keeping a critical contour in place. The most successful was one which designates the new nodal values for M as equal to the critical contour value plus or minus the distance each node is from the contour. If the original M-value at a node is greater than the critical contour value, the distance is added, if less the distance is subtracted.

Moving boundaries

In many practical applications such as forging of metals, the boundaries of the control volume will have specified velocities. For these problems it is desirable to use a procedure similar to the arbitrary Eulerian–Lagrangian method[9,10] whereby the mesh is given a velocity independent of the fluid. This allows specification of a mesh velocity which is consistent with the boundary velocities and which maintains uniformity of the mesh in the interior of the control volume. This independent velocity, used only to increment the mesh between calculations of the fluid velocity, does not enter the Eulerian formulation of the momentum equation. That is, the momentum equation is still used to describe the velocity field at an instant of time within a fixed spatial mesh.

In order to obtain the governing equation for the pseudo-concentration with respect to the moving mesh, let its representation in a fixed Cartesian reference frame be

$$M = M(x_i, t), \tag{8}$$

and let the velocity of the mesh be given by

$$U_i = U_i(x_i, t). \tag{9}$$

The time rate of change of the concentration with respect to a point moving with the mesh is then given by

$$\frac{\mathrm{D}M}{\mathrm{D}t} = U_i \frac{\partial M}{\partial x_i} + \frac{\partial M}{\partial t}. \tag{10}$$

However, the time rate of change of the concentration with respect to a material point (a point moving with the velocity of the fluid, u_i) is zero; hence

$$u_i \frac{\partial M}{\partial x_i} + \frac{\partial M}{\partial t} = 0. \tag{11}$$

If this last expression is subtracted from the first, we obtain

$$\frac{\mathrm{D}M}{\mathrm{D}t} = (U_i - u_i)\frac{\partial M}{\partial x_i}, \tag{12}$$

which is the governing equation for the concentration with respect to points moving with the mesh, e.g. the nodes. It is clear from this expression that whenever the mesh velocity coincides with the fluid velocity, the rate of change of the concentration is zero, as it is for the quasi-Lagrangian formulation. Likewise, whenever the mesh velocity is zero, the time rate of change coincides with that used in the Eulerian formulation.

Fluid interfaces

Although pseudo-concentrations were developed to track free surfaces of creeping viscous fluids, they can also be used to track interfaces between two different materials. Such problems occur frequently in the study of the slow deformations of the Earth's crust. For these applications, several critical contours can be used to represent the interfaces between strata. For situations where the geometry of the interfaces is so complex that it would be difficult to describe with a single function, more than one concentration can be used. Such a procedure adds very little to the overall cost of the analysis since the same matrix equation is applicable for each concentration.

ILLUSTRATION

In the examples presented below, the inertial terms were assumed negligible. The velocity field and the pseudo-concentration were approximated with quadratic 6-node triangular elements. Seven-

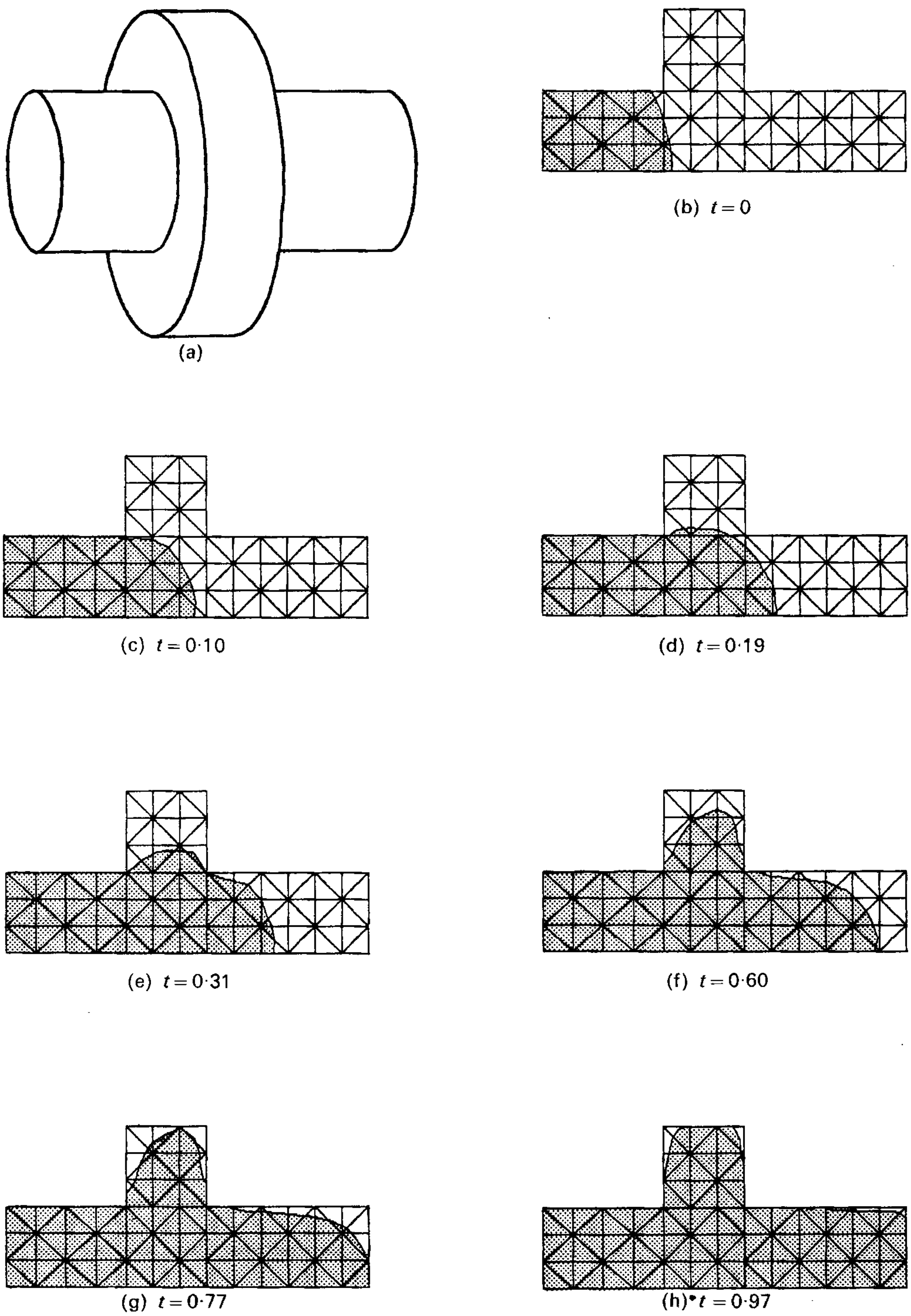

Figure 1. Axisymmetric analysis of the flow of fluid during injection moulding: (a) shape of mould; (b)–(h) profiles of the fluid as the mould is field

point Gaussian quadrature was used for the numerical integration. The pressure approximation, hence the degree to which the constraint of incompressibility was satisfied, was taken either as an independent linear function in each element, a constant in each element, or as a continuous piecewise linear function throughout the mesh

Injection moulding

The problem is to analyse the flow of a fluid as it is injected into a mould to form the shape shown in Figure 1(a). The fluid enters the mould through the left hand boundary at a constant rate and with a parabolic profile. The position of the material at the start of the analysis was assumed to be as shown in Figure 1(b). The viscosity of the artificial material was taken to be two orders of magnitude less than that of the material being simulated. An axisymmetric analysis was conducted using the upper half of the mould. The pressure was specified as a continuous, piecewise-linear function.

Because the mould is a closed container, it was necessary to make accommodation for the artificial material to either escape or become permanently compressed as the mould is filled. Both procedures were used for the analysis, with little difference except near the end of the analysis when the trapped voids occupied only small regions within single elements. In the case where the fluid was allowed to escape from the mould, the boundaries at the top of the flange and at the end of the shaft were left open, with the other boundaries assumed smooth. No-slip boundary conditions were prescribed at all nodes in contact with the real material.

The progression of the fluid as it fills the mould is shown in Figures 1(b)–(h). The non-dimensional time indicated in these Figures represents the ratio of the simulated time to the actual time needed to fill the mould based on its volume and the rate of injection. The material contacted the final node at a time equal to 1·1, which indicates a ten per cent error associated with the mesh and time step used for this analysis.

Compression of a cylindrical specimen

The second example illustrates the use of the method for applications which have moving boundaries. The problem, although relatively simple in concept, represents a rather difficult problem to analyse. A cylindrical specimen is compressed between two rough platens of the same diameter as the specimen. As the compression progresses, the material flows out and over the edges of the platen.

The problem's geometry is shown in Figure 2(a), where the vertical axis represents an axis of symmetry and the horizontal axis represents a plane of symmetry. The platens were assumed to have constant velocities. A mesh velocity was assigned to each node proportional to its distance from the horizontal axis. The velocity was in the vertical direction and coincided with the platen velocity at the contacting nodes. The pressure was taken as constant within each element, and the artificial viscosity was taken to be three orders of magnitude less than that specified for the real material.

Figures 2(c)–(f) show the resulting deformations and the initial and final mesh configurations. Figure 2(b) illustrates the final shape of the specimen. For comparison, the same problem was analysed using the quasi-Lagrangian procedure. Figure 3 shows the mesh as it deforms with the flow. The axial forces were calculated throughout the deformation for both procedures and are shown in Figure 4, where there magnitudes are given in terms of the initial force at zero deformation. The two methods give almost identical results until approximately 30 per cent compression. At that point, the mesh for the quasi-Lagrangian case apparently becomes distorted

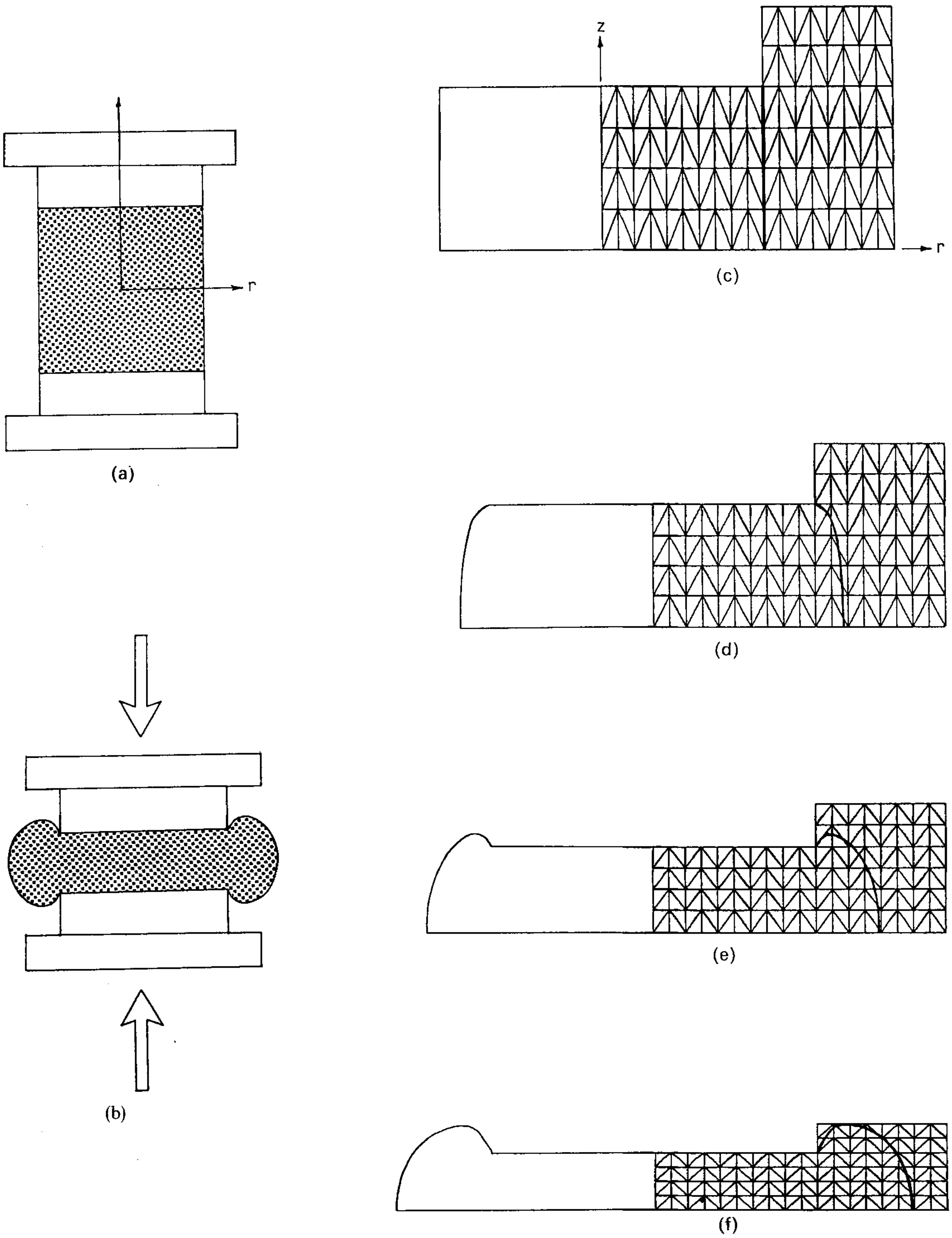

Figure 2. Compression of a cylindrical specimen: (a) geometry of the problem at the start of analysis; (b) final shape of the specimen; (c)–(f) deformation of the specimen during forging

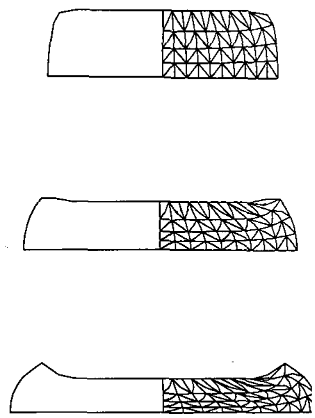

Figure 3. Deformation of specimen and mesh during a quasi-Lagrangian analysis

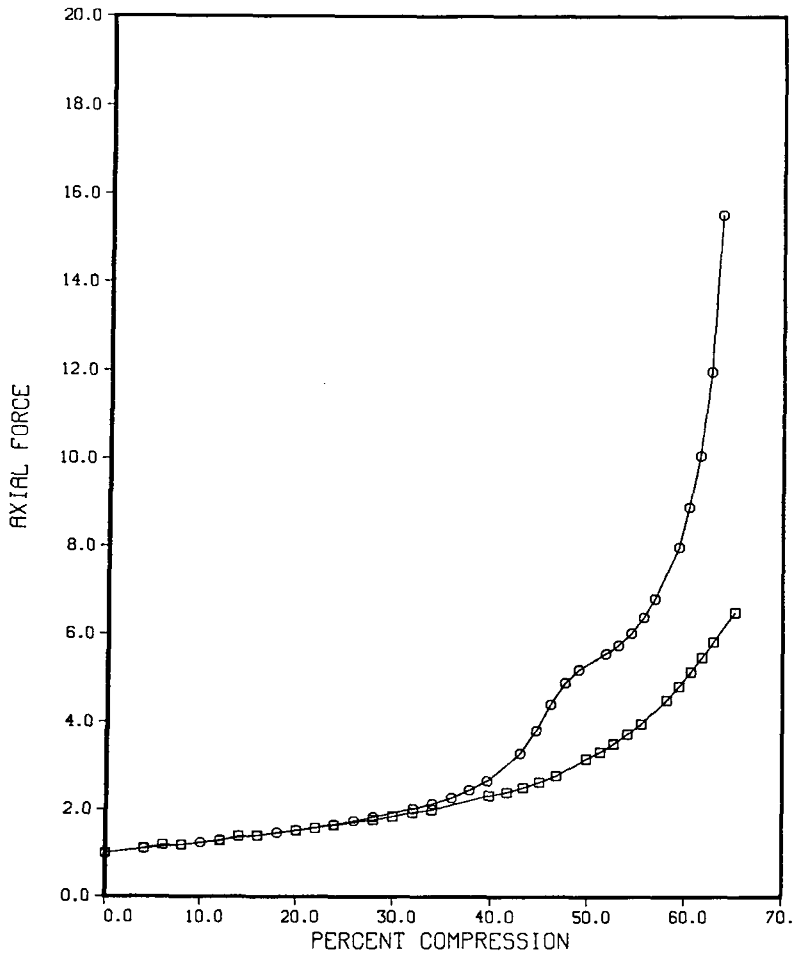

Figure 4. Axial force during compression. Quasi-Lagrangian analysis represented by circles, pseudo-concentration analysis represented by squares

enough to cause significant error, and shortly thereafter the results become meaningless.

Of some concern is the error associated with the integration of the matrix equations when the critical contour passes through an element creating a sharp discontinuity in the viscosity. To assess this error for the current problem, a test case was run where the platens were assumed to be smooth and much larger in diameter than the specimen, thus allowing the specimen to remain in pure compression throughout the deformation. The axial force necessary to maintain a constant velocity of the platens is

$$F = 3{\cdot}0 V \mu u / H^2, \tag{13}$$

Where V is half the volume of the specimen, u is the velocity of the platens relative to the plane of symmetry and H is half the height of the specimen.

The problem was analysed by the pseudo-concentration method and the results are compared with the above solution in Figure 5. The close alignment between the exact solution (the continuous curve) and the finite element solution (represented by the points) demonstrates the accuracy associated with the integration of the stiffness matrix as well as the accuracy with which the pseudo-concentration followed the flow.

Convective flow

For this illustration the slow convective flow of two fluids in a closed cylindrical container is analysed (see Figure 6(a)). The two fluids have the same viscosity, but the upper fluid has a density twice the density of the lower fluid; hence a convective flow results. Similar flows occur in the earth's crust and mantle and the present example could be likened to the rise of a salt dome. The mesh used for the analysis is shown in Figure 6(b). The pressure was taken as an independent function within each element, hence enforcing the constraint of incompressibility at each point. The initial positions of the two fluids are shown in Figure 7(a), and subsequent positions are shown in the remaining Figures 7(b)–(f). The unit of time indicated in the Figures is defined as $t = \mu/\rho g h$.

The analysis was stopped at the point shown because the fluid was essentially in a state of static equilibrium, with only an insignificant volume of flow occurring in the narrow cylindrical shaft at the axis of symmetry.

Owing to the stagnation points at the corners of the mesh, the M-surface became distorted as the flow progressed, making it necessary to smooth the surface several times during the analysis. Figure 7(g) shows the contours of the M-surface before one such smoothing and Figure 7(h) shows contours after smoothing. The location of the interface contour is indicated by the arrows and is the same for both surfaces.

Folding

The final example is another one applicable to geology and illustrates the possible use of the method to study the folding of stratified rocks in the earth's crust. Figures 8(a) and 8(d) show the initial mesh with the critical contours, and positions of the rock strata. The relative viscosities of the strata are shown in Figure 8(b). The upper, low-viscosity layer represents the absence of material, thus providing a means to study the surface deformation. The initial offset of the strata represents a fault which was assumed to exist at the start of motion. The left hand boundary was assumed smooth, simulating a plane of symmetry. The right hand boundary was given a constant velocity to the left. The lower boundary was also assumed smooth. Figures 8(d)–(h) show the resulting deformation of the strata due to the horizontal compression. Figure 8(c)

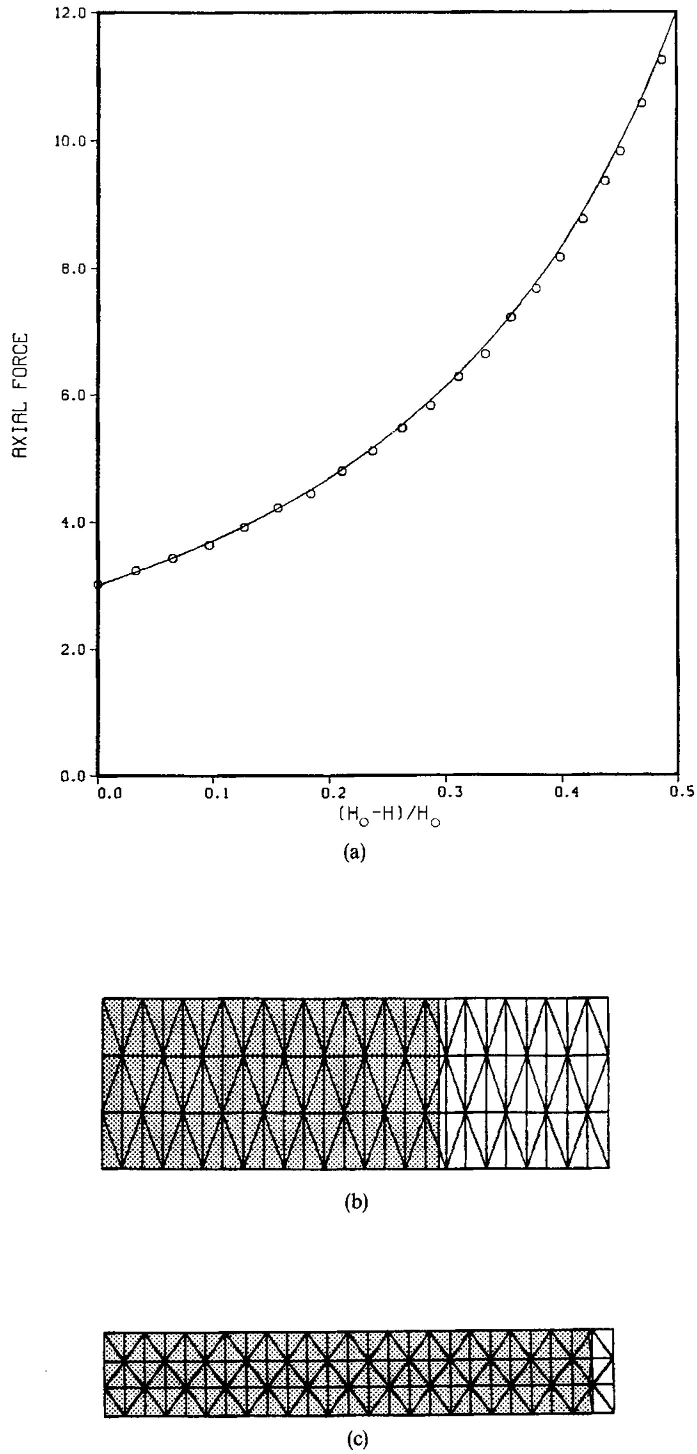

Figure 5. Pure compression test: (a) force-deformation results compared to exact solution; (b) initial mesh and material location; (c) final mesh and material location

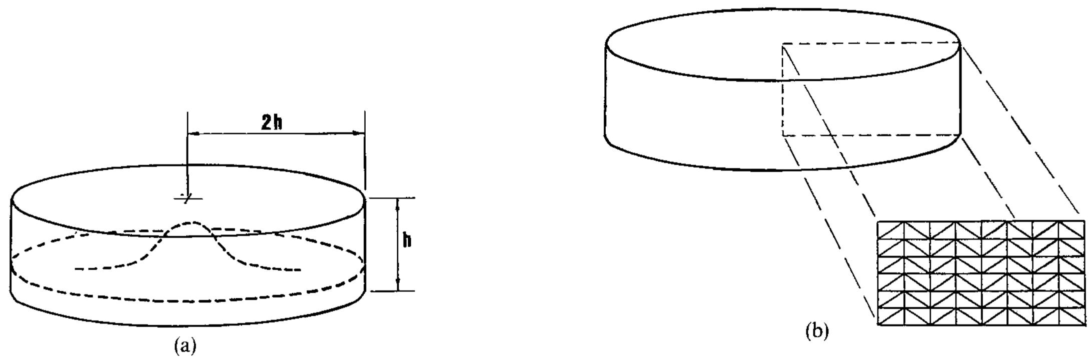

Figure 6. Convective flow of two fluids in a closed container: (a) physical layout showing lighter fluid rising at the centre of the container; (b) finite element mesh used for analysis

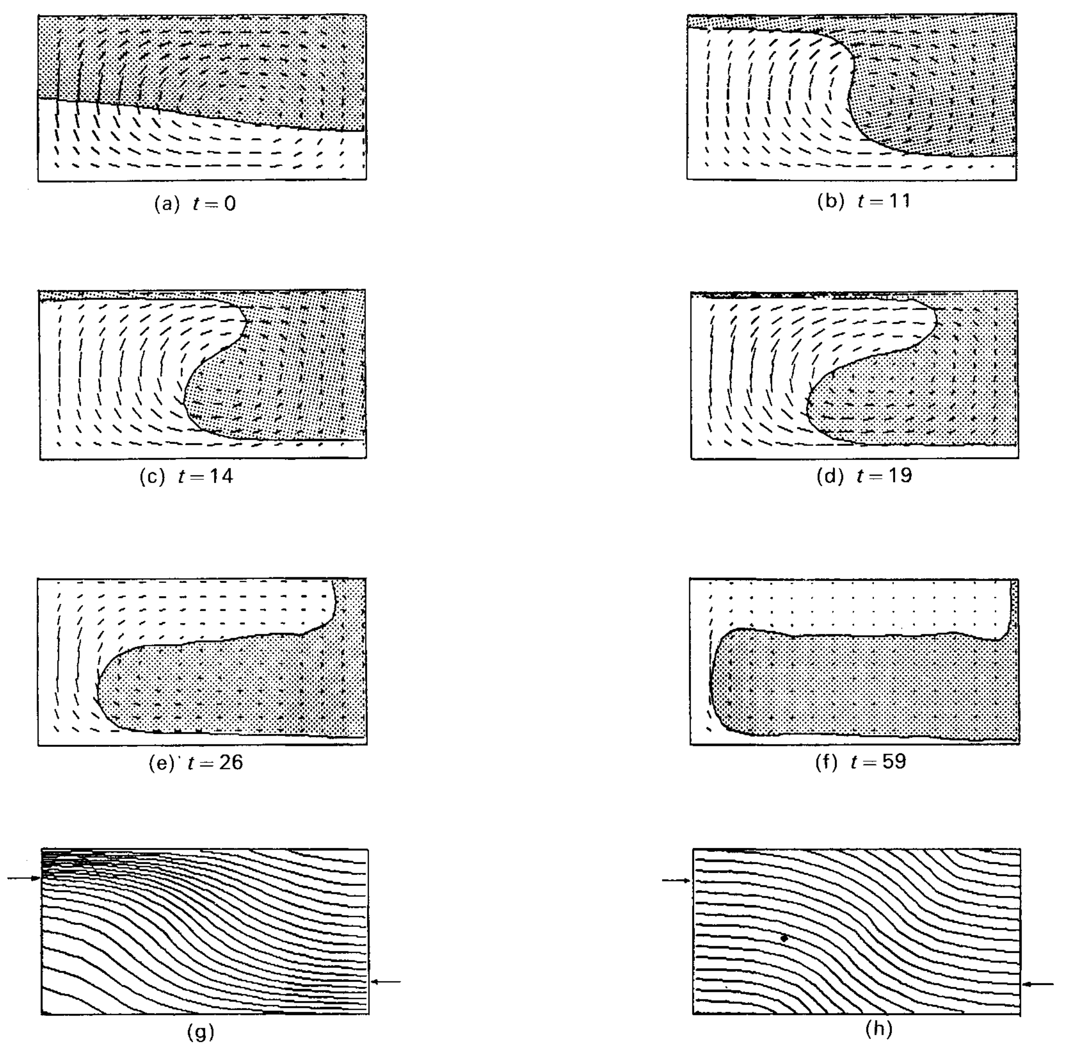

Figure 7. Solution to the convective flow problem: (a)–(f) convective flow patterns showing velocity vectors and the interface contour; (g) M-surface contours before smoothing; (h) M-surface contours after smoothing

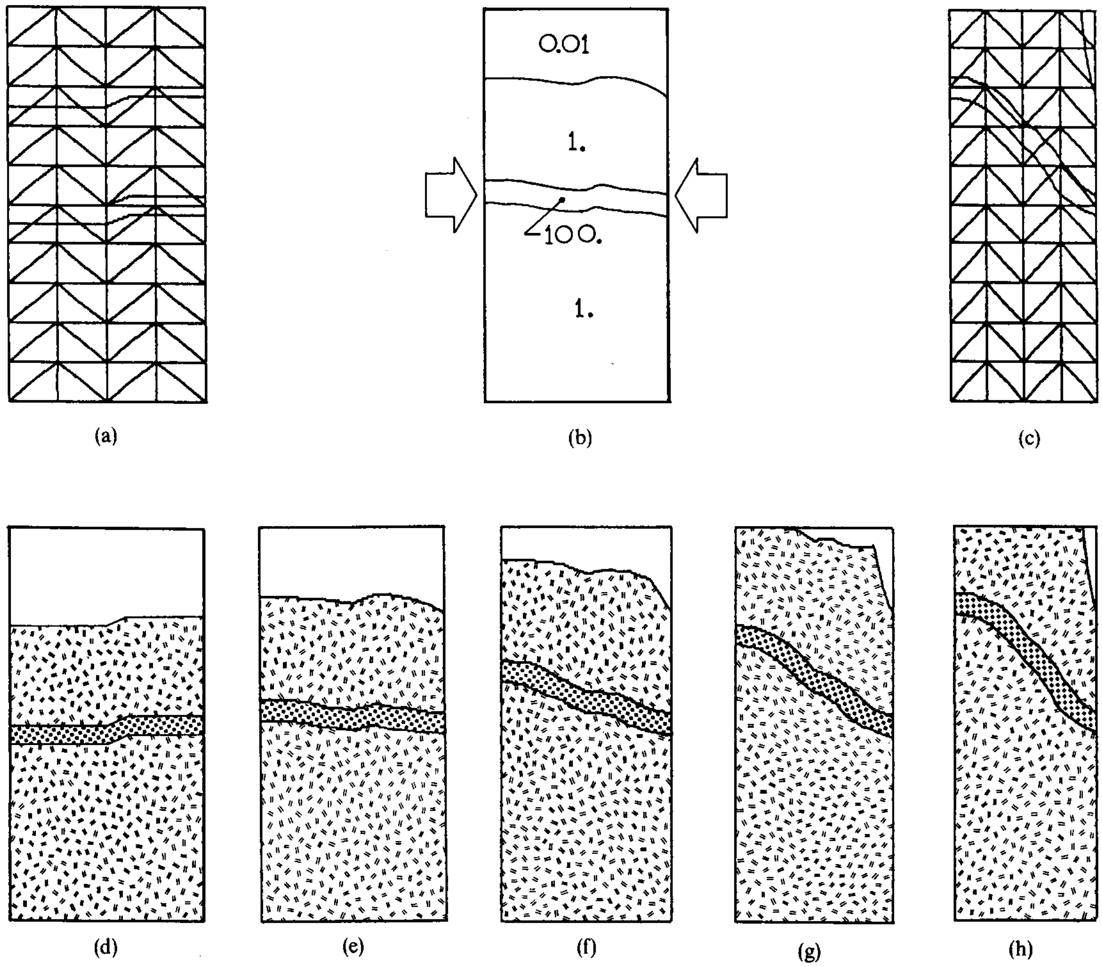

Figure 8. Deformation of rock strata during compression: (a) initial mesh with critical contours; (b) relative viscosities of the rock; (c) final mesh with critical contours; (d)–(h) deformation of the strata during compression

shows the final position of the mesh with the critical contours. For this illustration, the pressure was taken to be constant within each element.

CONCLUDING REMARKS

The use of a pseudo-concentration to designate the location of a creeping viscous material during a transient analysis has been found to be an easy method to follow flows through large deformations. An increase in computer time is needed to increment the pseudo-concentration, which is compensated for by not having to re-mesh during an analysis. In addition, the method allows the design of meshes without the need for matching element boundaries with specified free surfaces or interfaces. Although the surface representing the concentration can become distorted and prevent continuation of the analysis, it is relatively easy to define a smoother surface in such a way that the critical contour remains in place. One such smoothing routine has proved successful and was used in the analyses presented in this paper.

ACKNOWLEDGEMENTS

The work presented in this paper was supported by the Aluminum Company of America. This support is gratefully acknowledged.

REFERENCES

1. E. G. Thompson and M. I. Haque, 'A high order finite element for completely incompressible creeping flows', *Int. j. numer. methods eng.*, **6**, 315–321 (1973).
2. O. C. Zienkiewicz and P. N. Godbole, 'A penalty function approach to problems of plastic flow of metals with large surface deformations', *J. Strain Analysis*, **1**, (3), (1975).
3. Oskar Mahrenholtz, 'Different finite element approaches to large plastic deformations', *Computer Methods in Applied Machanics and Engineering*, **33**, 453–468 (1982).
4. Shiro Kobayashi, 'Thermoviscoplastic analysis of metal forming problems by the finite element method', in J. F. T. Pittman, O. C. Zienkiewicz, R. D. Wood and J. M. Alexander (eds), *Numerical Analysis of Forming Processes*, Wiley, Chichester, 1984, Chapter 2.
5. O. C. Zienkiewicz, 'Flow formulation for numerical solution of forming Processes', in J. F. T. Pittman, O. C. Zienkiewicz, R. D. Wood and J. M. Alexander (eds), *Numerical Analysis of Forming Processes*, Wiley, Chichester, 1984, Chapter 1.
6. J. H Argyris, T. J. R. Hughes, J. T. Hughes and R. E. Ewing, *Computer methods in applied machanics and engineering*', *Special Issue on Oil Reservoir Simulation*, **47**, (1–2), (1984).
7. E. G. Thompson, J. F. T. Pittman and O. C. Zienkiewicz, 'Some integration techniques for the analysis of viscoelastic flows', *Int. j. numer. methods fluids*, **3**, 165–177 (1983).
8. O. C. Zienkiewicz, *The Finite Element Method, 3rd Edition*, McGraw-Hill, London, 1977.
9. C. W. Hirt, A. A. Amsden and J. L. Cook, 'An arbitrary Lagrangian–Eulerian computing method for all flow speeds, *Journal of Computational Physics*, **14**, 227–253 (1974).
10. P. J. G. Schreurs, F. E. Veldpaus and Q. A. M. Brekelmans, 'An arbitrary Eulerian–Lagrangian finite element model for the simulation of geometrical non-linear hyper-elastic and elasto-plastic deformation Processes', in J. F. T. Pittman, R. D. Wood, J. M. Alexander and O. C. Zienkiewicz (eds), *Numerical Methods in Industrial Forming Processes*, Pineridge Press, Swansea, 1982, pp. 491–500.