EURDYN, FINITE ELEMENT CODES FOR DYNAMIC ANALYSIS OF LARGE-DISPLACEMENT, SMALL-STRAIN PROBLEMS WITH MATERIAL NON-LINEARITIES

J. DONEA and S. GIULIANI

Materials Division, Euratom J.R.C., I-21020 Ispra, Italy;
Commission of the European Communities

SUMMARY

A brief description is given of three finite elements programs (EURDYN) designed for the nonlinear transient dynamic analysis of two and three-dimensional structures.

Nonlinearities arise from large displacements as well as from elastic-plastic material behavior. A convected co-ordinate technique is used to deal with the geometric nonlinearity under the assumption that the strains are small. Elastic-plastic relations of the incremental type are used for material description in connection with the von Mises criterion of yielding. Isotropic and kinematic hardening models have been implemented.

A simple central difference time integration scheme together with a diagonal mass matrix is used to compute the transient dynamic response. Such a scheme combined with a direct evaluation of the internal forces in terms of stresses completely eliminates the usual limitations arising from bandwidth or problem size.

Three versions of EURDYN have been developed. The finite element library of EURDYN/1 includes the constant-strain triangle, a rectilinear beam and a conical shell, The convected co-ordinate formulation for these elements is that of Belytschko et al. EURDYN/2 is based on 8-node isoparametric elements with or without curved sides. Plane stress, plane strain and axisymmetric responses can be obtained. EURDYN/3 computes the Kirchhoff-type dynamic response of three-dimensional thin shells. Spatial discretization is achieved by means of triangular, flat-plate bending elements to which a homogeneous membrane deformation field has been added.

Several numerical examples are presented illustrating the transient dynamic responses obtained with the three versions of EURDYN.

1. Introduction

The ability to predict the nonlinear dynamic response of complex structural components subjected to time and space dependent loads is a problem of considerable importance in the field of fast-reactor safety.

The finite element method is ideally suited to the analysis of such problems since it can account for arbitrary geometries, loadings and material property variations.

This method has been chosen for the development of the computer programs EURDYN for the transient dynamic analysis of large-displacement, small-strain problems with material non-linearities.

Since the EURDYN programs were conceived as tools for numerical investigations in the field of fast-reactor safety, their main options were dictated by the type of structural behavior and by the nature of loading encountered in this context.

Dynamic problems with small strains but arbitrarily large linear and angular displacements are very frequent in the analysis of fast reactor explosive accidents. A very efficient technique for dealing with such problems has been suggested by Belytschko et al. [1-2-3] The technique consists in formulating the equations of motion in terms of convected co-ordinates that rotate but do not deform with the elements. Such a procedure linearizes the strain-displacement and simplifies the nodal force-stress relations within the elements' convected co-ordinates. This method has been adopted in EURDYN since it leads to significant improvements in computer time with respect to other treatments of the non-linearity due to large displacements.

Usually the dynamic pressures arising from the simulation of fast-reactor explosive accidents are of relatively short duration.
This indicates that the choice of a lumped-explicit scheme for the time integration of the equations of motion can be justified, even if such schemes do present rather severe stability limitations. Furthermore the use of a lumped-explicit scheme combined with a direct nodal force evaluation in terms of stresses completely eliminates the usual limitations arising from bandwidth or problem size.

Elastic-plastic relations of the incremental type are used for material description in connection with the von Mises criterion of yielding.
The stress-strain curve is input either in a trilinear form or according to the Ramberg-Osgood formula. Isotropic and kinematic hardening models have been implemented. The elasto-plastic matrix relating the stress increments to the total strain increments is evaluated according to the stress state at the beginning of each time step.

Three versions of EURDYN have been developed and are briefly described in the present paper. A more complete description of the programs is given in a report to be presented at the post-conference seminar ELCALAP[4].

2. Basic equations

Although the basic equations for a finite-element solution of nonlinear dynamic problems by means of convected co-ordinates are well known[1-2], they are summarised here for the sake of completeness and also to introduce the notations which will be used later.

The displacements within an element are subdivided into rigid body displacements $\{f^{rig}\}$ and displacements that result only in deformation $\{f^{def}\}$. The nodal components can be similarly decomposed so that

$$\{\delta\}^e = \{\delta^{rig}\}^e + \{\delta^{def}\}^e \qquad (1)$$

If it is assumed that the shape functions $\lfloor N \rfloor$ can represent rigid body motion, the deformation displacements can be represented by

$$\{f^{def}\} = [N] \; \{\delta^{def}\}^e \qquad (2)$$

The displacement components in eq. (2) are refered to a fixed co-ordinate system $\{x\}$. Now imagine that each element is associated with a convected

co-ordinate system $\{\hat{x}\}$ which is rotated relative to the fixed co-ordinates $\{x\}$ by the rigid body rotation of that element.
If the rigid body rotation is not constant within the element, an angle which approximates the rigid body rotation is used.

Displacements in convected and fixed co-ordinates are related by an orthogonal transformation

$$\{\hat{\delta}\}^e = [T]\{\delta\}^e \tag{3}$$

Furthermore, it can be shown[1] that for large-displacement, small-strain problems, the strain $\{\hat{\varepsilon}\}$ measured in the convected co-ordinates are linearly related to the deformation displacements $\{f^{def}\}$. Therefore, a matrix $[B]$ can be derived from the linear strain-displacement relationships so that

$$\{\hat{\varepsilon}\} = [B]\{\hat{\delta}^{def}\}^e \tag{4}$$

The vector $\{\hat{\varepsilon}\}$ contains all the relevant direct and shear strains as components.

Using the principle of virtual work with the inertial forces included in a d'Alembert sence and evaluating the internal resisting forces in terms of stresses, the equations of motion are obtained in the form

$$\{\delta^{oo}\} = [M]^{-1}\left(\{F_{ext}\} - \{F_{int}\}\right) \tag{5}$$

where $\{\delta^{oo}\}$ lists all nodal acceleration components, $[M]$ represents the global mass matrix, $\{F_{ext}\}$ are the external nodal loads and $\{F_{int}\}$ are the internal nodal forces.

The local internal forces are calculated in the elements' convected co-ordinates as

$$\{\hat{f}_{int}\}^e = \int_{V_e} [B]^T \{\hat{\sigma}\}\, dV \tag{6}$$

where $\{\hat{\sigma}\}$ are the convected stresses.

These local forces are then transformed to the global co-ordinate system by

$$\{f_{int}\}^e = [T]^T \{\hat{f}_{int}\}^e \tag{7}$$

to permit assembly into the global vector $\{F_{int}\}$ of equation (5).

The EURDYN programs use a lumped mass matrix. The lumped masses are obtained by equally apportioning the mass of each element among its nodes. When an element has rotational degrees of freedom, its mass moment is also equally apportioned among its nodes.

The equations of motion (5) are integrated explicitly in time by a difference method in which the velocities and displacements are computed by

$$\{\delta^{o}(t+\Delta t)\} = \{\delta^{o}(t)\} + 1/2\,\Delta t\left[\{\delta^{oo}(t)\} + \{\delta^{oo}(t+\Delta t)\}\right] \tag{8a}$$

$$\{\delta(t+\Delta t)\} = \{\delta(t)\} + \Delta t\{\delta^{o}(t)\} + 1/2\,\Delta t^2\{\delta^{oo}(t)\} \tag{8b}$$

The time increment Δt is calculated from[5]

$$\Delta t = \beta\,\Delta l / C \tag{9}$$

where C is the dilational speed of sound of the material and ΔL is taken as the smallest element side in the mesh. The coefficient β is chosen between 0.1 and 0.5, depending on the rise time of the loading functions.

3. Material description

Elastic-plastic relations of the incremental type are used for material description. The objective is to obtain the stress state σ_{ij} at time $t+\Delta t$, given the stress state σ_{ij} at time t and the total strain increments $d\varepsilon_{ij}$.

The total strain increments are assumed to consist of elastic and plastic components, so that the elastic strain increments may be expressed as

$$d\varepsilon_{ij}^{E} = d\varepsilon_{ij} - d\varepsilon_{ij}^{P} \qquad (10)$$

The generalized Hooke's law relates the stress increments to the elastic strain increments.

$$d\sigma_{ij} = D_{ijkl}\,(d\varepsilon_{kl} - d\varepsilon_{kl}^{P}) \qquad (11)$$

The plastic strain increments are taken proportional to the gradient of the loading function

$$d\varepsilon_{ij}^{P} = d\lambda \frac{\partial F}{\partial \sigma_{ij}} \qquad (12)$$

3.1. Isotropic hardening

Using Hooke's law (11) and the von Mises criterion of yielding

$$\left(\frac{3}{2} S_{ij} S_{ij}\right)^{1/2} - \bar{\sigma}_e = 0 \qquad (13)$$

where $S_{ij} = \sigma_{ij} - 1/3\ \delta_{ij}\ \sigma_{kk}$ and $\bar{\sigma}_e$ = current yield in tension, one can express[6] the plastic strain increments (12) in terms of the total strain increments :

$$d\varepsilon_{ij}^{P} = \frac{9G}{2\,\bar{\sigma}_e^2\,(E' + 3G)}\ S_{ij}\ S_{kl}\ d\varepsilon_{kl} \qquad (14)$$

where $G = \dfrac{E}{2(1+\nu)}$ and $E' = \dfrac{E\ E_p}{E - E_p}$

E_p represents the slope of the $\sigma - \varepsilon$ curve in the plastic range. This curve can be given either in a trilinear form or in the Ramberg-Osgood analytical form.

Combining equations (11) and (14) the stress increments are found as functions of the given total strain increments.

3.2. Kinematic hardening

Using the von Mises criterion of yielding and denoting the translation of the center of the yield surface by α_{ij} , the loading function can be written as

$$\left(\frac{3}{2}(S_{ij} - \alpha_{ij})(S_{ij} - \alpha_{ij})\right)^{1/2} - \sigma_0 = 0 \qquad (15)$$

where σ_0 is the yield stress in tension.

According to Prager's kinematic hardening the increment of translation of the yield surface $d\alpha_{ij}$ is in the direction of the plastic strain increment:

$$d\alpha_{ij} = C\ d\varepsilon_{ij}^{P} \qquad (16)$$

For a linearly hardening (bilinear) material one has

$$C = \frac{2}{3}\,\frac{E E_p}{E - E_p} \qquad (17)$$

Combining equations (15)(16) with the flow rule (12), one can express the plastic strain increments in terms of the total strain increments by[4]

$$d\varepsilon_{ij}^{p} = \frac{3G}{\sigma_0^2 (C+2G)} (S_{ij} - \alpha_{ij})(S_{kl} - \alpha_{kl})\, d\varepsilon_{kl} \qquad (18)$$

The introduction of relation (18) into equation (11) gives the stress increments corresponding to the total strain increments.

4. EURDYN/1

The finite-element library of the Fortran program EURDYN/1 includes the constant-strain triangle, plane or axisymmetric, a rectilinear beam element and a conical shell element.

The convected co-ordinate formulation for these elements is that of Belytschko et al.[1-2-3] The Bernoulli-Euler hypothesis is invoked in the derivation of the flexural elements. The transverse displacement is assumed to have a cubic variation along the elements, the axial displacement is linear.

A restart option has been included in the program in order to make it possible to continue a solution at the point it was terminated in an earlier analysis.

5. EURDYN/2

This program is based on 8-node isoparametric elements with or without curved sides. Plane stress, plane strain and axisymmetric situations can be analysed.

The convected co-ordinate formulation of the isoparametric elements is fully described in reference 4. Only a brief description will be given here.

With reference to Fig.1-2, the parabolic displacement field relative to node 1 of the element can be written as

$$u^{rel} = \sum_{i=2}^{8} N_i\, u_i^{rel}, \qquad v^{rel} = \sum_{i=2}^{8} N_i\, v_i^{rel} \qquad (19)$$

where the shape functions N_i are defined in terms of the normalized co-ordinates (ξ,η) and (u_i^{rel}, v_i^{rel}) are the relative nodal displacements in the global co-ordinate system (x,y) for plane problems and (r,z) for axisymmetric situations.

Since the rigid body rotation is not constant within the element, an angle θ which approximates the rigid body rotation is used. This angle is defined as indicated in Fig. 1-2 by the rotation of the diagonal 1-3. The rigid-body displacements associated with the angle θ are readily evaluated and the deformation nodal displacements in the convected axes $(\hat{u}_i^{def}, \hat{v}_i^{def})$ are found.

Since the shape functions N_i can represent rigid-body motion, the deformation displacements $(\hat{u}^{def}, \hat{v}^{def})$ can be expressed in the form given by equation (19). The convected strains are then found using the linear strain-displacement relationships. In the case of plane situations one has,

$$\hat{\varepsilon}_x = \sum_{i=2}^{8} \frac{\partial N_i}{\partial \hat{x}}\, \hat{u}_i^{def} \quad ; \quad \hat{\varepsilon}_y = \sum_{i=2}^{8} \frac{\partial N_i}{\partial \hat{y}}\, \hat{v}_i^{def}$$

$$\hat{\gamma}_{xy} = \sum_{i=2}^{8} \left(\frac{\partial N_i}{\partial \hat{y}}\, \hat{u}_i^{def} + \frac{\partial N_i}{\partial \hat{x}}\, \hat{v}_i^{def} \right) \qquad (20)$$

The internal nodal forces defined by equation (6) can now be written as

$$\hat{f}_i^x = \int_V \left(\frac{\partial N}{\partial \hat{x}} \hat{\sigma}_x + \frac{\partial N}{\partial \hat{y}} \hat{\tau}_{xy} \right) dV$$

$$\hat{f}_i^y = \int_V \left(\frac{\partial N}{\partial \hat{y}} \hat{\sigma}_y + \frac{\partial N}{\partial \hat{x}} \hat{\tau}_{xy} \right) dV \qquad (i=1,2,\dots,8) \tag{21}$$

where $(\hat{\sigma}_x, \hat{\sigma}_y, \hat{\tau}_{xy})$ is the stress vector measured in the convected co-ordinates. These forces are then transformed to the global co-ordinate system to permit assembly into the global vector $\{F_{int}\}$ of equation (5).

Since the system of nodal forces is self-equilibrated, the nodal forces at node 1 of the element are given by

$$f_1^x = -\sum_{i=2}^{8} f_i^x \qquad f_1^y = -\sum_{i=2}^{8} f_i^y$$

A 2x2 Gaussian quadrature formula is used to evaluate the local internal forces (21).

6. EURDYN/3

This program has been developed in order to compute the Kirchhoff-type dynamic response of three-dimensional thin shells.

In order to achieve reasonable computational time, a simple element has been chosen for spatial discretization. As suggested by Zudans in his DYPLAS [7], a triangular, flat-plate bending element with cubic transverse displacements has been used. For the in plane displacements, the usual Kirchhoff assumptions are invoked in combination with linear displacements of the reference surface of the element.

The convected co-ordinate formulation for the non-conforming triangular element has been obtained as a generalization of the Euler-Bernoulli rectilinear beam element of reference (1). A complete description of the procedure is presented in references (4)(8); only a brief illustration will be given here.

The convected axes associated with the triangular element are defined in such a way that the $(\hat{x},\hat{y})$ plane passes through the element nodes in the deformed configuration. The origin coincides with node 1 of the element and the $\hat{x}$ axis is taken along the bisectrix of the angle at node 1. Generalizing the beam element, the deformation displacements of the element are taken to be the nodal rotations relative to the $(\hat{x},\hat{y})$ axes and, as suggested by Argyris[9], the membrane strains associated with the elongations of the element sides. The convected strains $\{\hat{\varepsilon}\}$ are then found by making use of the usual Kirchhoff hypothesis. The element nodal forces and moments corresponding to the deformation displacements are obtained by application of relation (6). The transverse forces result from equilibrium considerations.

The nodal forces are transformed to the global co-ordinate system as indicated in equation (7) to permit assembly into the global vector $\{F_{int}\}$ of equation (5). The translational equations of motion can then be solved.

The treatment of the rotational degrees of freedom is more difficult. As suggested in reference (3), the rotational equations of motion are expressed at each node in the principal axes of inertia. The principal moments of inertia are assumed to remain constant during the deformation process. Only the directions of the principal axes are adjusted at the end of each time step as described in references (4)(8).

7. Numerical examples.

Two numerical examples are presented in this section illustrating the dynamic responses that can be obtained with the three versions of the

EURDYN program.

Elastic-plastic dynamic analysis of a beam under blast loading.

This example has been taken from reference 5. The beam geometry and loading history are shown in Fig.3 together with the material properties. The displacement history of the beam center has been calculated with the three versions of EURDYN and compared with the experimental data given in reference 10.

Plane strain was assumed in the two-dimensional calculations. Twenty rectilinear beam elements were used in EURDYN/1 to model half of the beam. For the EURDYN/2 response twenty isoparametric elements were employed. A 3D calculation was also performed using twenty triangular elements to represent one quart of the beam (the beam depth is 1.2in).

Fig.3 shows that the numerical solutions are in good agreement with the experimental values. The EURDYN/1 response appears to be the best one. Modeling with isoparametric elements when flexural effects are dominant yields a slightly stiff response characterized by a reduced peak amplitude and a shorter period of vibration. Finally, it should be noted that the 3D-solution has been obtained with a coarser mesh (10 elements) along the beam axix. Isotropic hardening was assumed in this example.

Nonlinear dynamic response of a spherical cap.

The spherical cap represented in Fig.4 has been analysed with a uniform pressure of 1000 psi applied as a step load. A high pressure was chosen in order to induce a very pronounced elasto-plastic behavior. The stress-strain curve of the material was input in the Ramberg-Osgood analytical form and nonlinear isotropic hardening was considered.

A first analysis was made with both EURDYN/1 and EURDYN/2 using ten elements to discretize the symmetric portion of the spherical cap. The solution was then repeated with both programs using twenty elements. Fig.5 shows the displacement history of the apex of the cap. It can be seen that the responses obtained with both programs are in reasonable agreement. Numerical integration of the conical shell element was effected using two Gauss points along the length and five points through the thickness. A 2x2 Gaussian scheme was employed for the isoparametric element.

8. Conclusions.

A brief description has been given of three finite-element computer programs for nonlinear dynamic analysis of two and three-dimensional structures.

Although only simple academic examples have been presented, the program capabilities do cover a large range of problems in the field of fast-reactor engineering.

Acknowledgements

The authors wish to express their gratitude to Mr J.P. Halleux of Liège University (Belgium) for his very significant contribution to the development of the EURDYN/3 program.

References

1. T. Belytschko, B.J. Hsieh:"Nonlinear Transient Finite Element Analysis with Convected Co-ordinates", Int. J. for Num. Methods in Engng, Vol 7, 255-271 (1973).

2. T. Belytschko, B.J. Hsieh : "Nonlinear Transient Analysis of Shells and Solids of Revolution by Convected Elements", AIAA Journal, Vol 12, n°8, 1031-1035, (1974).

3. T. Belytschko, B.J. Hsieh, J.M. Kennedy : "Convected Co-ordinates for Transient Nonlinear Finite-Element Analysis", Paper M3/9, 2nd International Conference on Structural Mechanics in Reactor Technology, Berlin, Germany, (1973).

4. J. Donéa, S. Giuliani, J.P. Halleux : "Prediction of the Nonlinear Dynamic Response of Structural Components Using Finite Elements", International Seminar on Extreme Load Conditions and Limit Analysis Procedures for Structural Reactor Safeguards and Containment Structures, BAM, Berlin, 8-11 September 1975.

5. M. Hartzman, J.R. Hutchinson : "Nonlinear Dynamics of Solids by the Finite Element Method", Computers and Structures, Vol. 2, 47-77, (1972).

6. F. Frey : "Le Calcul Elasto-Plastique des Structures par la Méthode des Eléments Finis et son Application à l'état plan de Contrainte", Rapport n°33, Laboratoire de Mécanique des Matériaux et de Statique des Constructions, Université de Liège, Belgique (1973).

7. Z. Zudans, M.M. Reddi, Hsi-Chin Tsai : "DYPLAS, a Finite Element Dynamic Elastic-Plastic Large Deformation Analysis Program", Invited Lecture L6/1, 2nd International Conference on Structural Mechanics in Reactor Technology, Berlin, (1973).

8. J.P. Halleux : "Analyse Dynamique Non-Linéaire de Structures Minces à trois Dimensions à l'aide d'un Elément Fini de Plaque", Mémoire d'Ing. Physicien, Université de Liège, Belgium (1975).

9. J.H. Argyris, S. Kelsey, H. Kamel : "Matrix Methods of Structural Analysis : A Precis of Recent Developments", in Matrix Methods of Structural Analysis, Editor B. Fraeijs de Veubeke, AGARDograph 72, Pergamon Press (1964).

10. H.A. Balmer : "Theoretical-Experimental Correlation of Large Dynamic and Permanent Deformations of Impulsively Loaded Simple Structure", Research and Tech. Div., Wright-Patterson AFB, Rept. FDL-TDR 64-108, (1964).

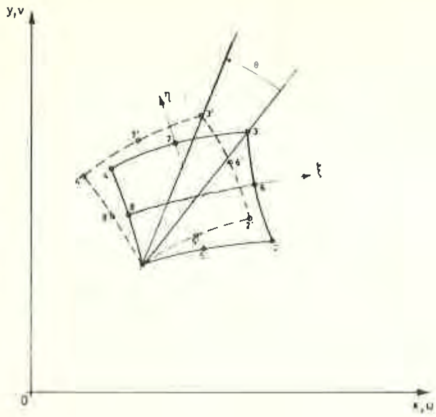

Fig. 1 : Plane, parabolic isoparametric element. The angle θ approximates the rigid-body rotation of the element.

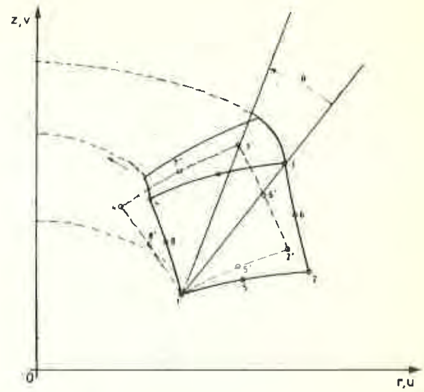

Fig. 2 : Axisymmetric, parabolic isoparametric element. The angle θ approximates the rigid-body rotation of the element.

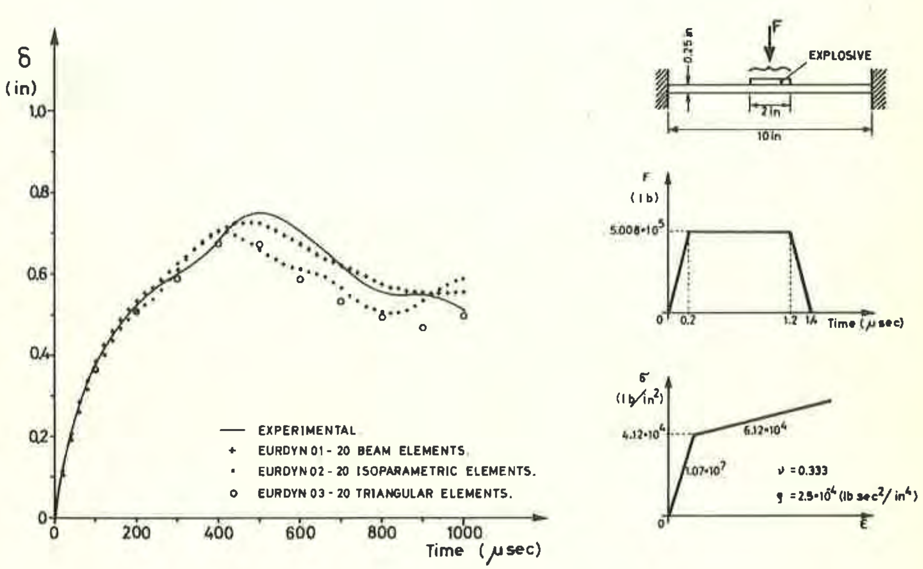

Fig. 3 : Elastic-Plastic dynamic response of a beam under blast loading. δ is the displacement of the beam center.

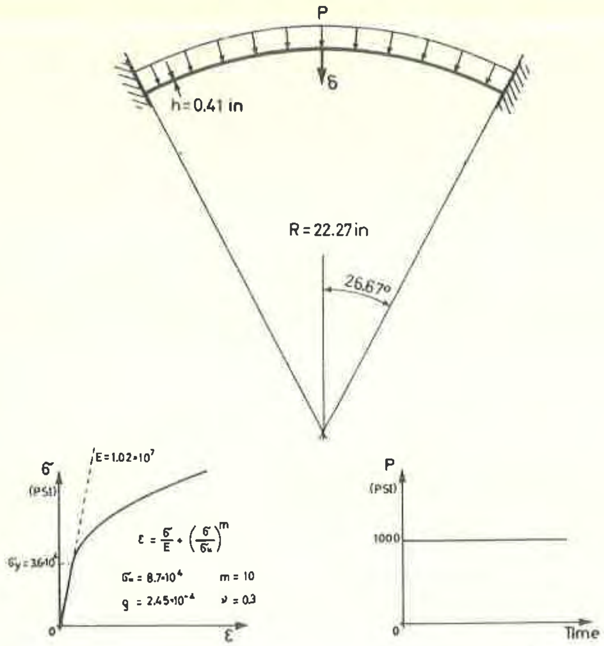

Fig. 4 : Spherical cap with uniform pressure applied as a step load. The stress-strain curve of the material is given in the Ramberg-Osgood analytical form.

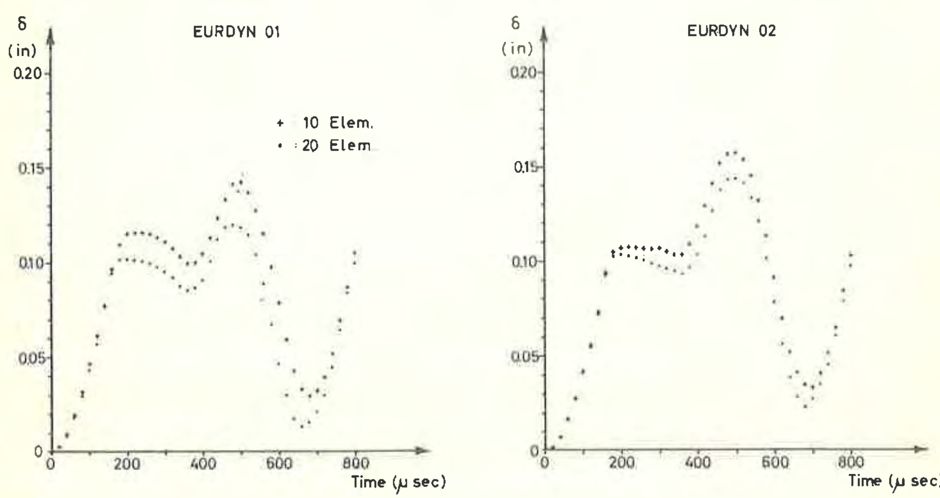

Fig. 5 : Displacement history of the apex of the spherical cap in Fig.4. EURDYN/1 uses conical shell elements, EURDYN/2 isoparametric elements.