ROZPRAWY INŻYNIERSKIE • ENGINEERING TRANSACTIONS • 23, 4, 609—624, 1975
Polska Akademia Nauk • Instytut Podstawowych Problemów Techniki

# FINITE ELEMENT ANALYSIS OF ELASTIC-PLASTIC PLANE STRESS PROBLEM

L. WINNICKI, M. KLEIBER, M. KWIECIŃSKI (WARSZAWA)

The finite element method is applied to the statical analysis of an elastic-perfectly plastic material in plane stress. The in-plane system is split up into triangular elements for which linear shape function is prescribed. Basic relationships are formulated for the elastic-plastic material in the matrix form. The procedure to build up an elastic-plastic stiffness matrix is shown for the Huber-Mises-Hencky material in the three-dimensional situation as well as in both plane stress and plane strain. In the numerical calculations an ideally plastic material is assumed, however, the derived relationships are valid for strain-hardening material.

Basic algorithms are described for the solution of the non-linear system of equations obtained. The method of tangential stiffness is presented together with the method of initial nodal forces and its modification — the initial stress method. In the course of numerical analysis was investigated the possibility of the unloading process to occur in particular elements while the whole system undergoes loading program. The algorithm is given and a numerical example is solved.

## 1. Introduction

In the present-day mechanics of structures much effort is devoted to the establishing of computational procedures to deal with the behaviour of elastic-plastic bodies. Most powerful tool for that purpose appears to be the finite element method. It is especially useful when applied to both physically and geometrically non-linear problems. The elastic-plastic analysis falls into the first category by the very nature of the constitutive equations. The second type of non-linearity occurs when large deformations are allowed for.

Two basic methods are nowadays recognized to study the behaviour of plastically deformable structures: the tangential stiffness method, e.g. [1], and the initial load method. The latter has two variants known as the initial strain method, [2, 3], and the initial stress method [4]. Both variants are closely related iterative procedures. The tangential stiffness method is the most direct and physically understandable although a substantial computational effort is necessary to reach effective solutions. Moreover, the method proves to be expedient when large displacements and plastic deformations have to be accounted for simultaneously.

The tangential stiffness method is based on the concept that a non-linear problem is approached via a number of linearized steps. Thus an elastic-plastic material stiffness matrix is introduced in place of a purely elastic stiffness. At each step of the analysis a single discrete element is brought to yielding and thus the element stiffness matrix of this plastic element must be built up, the global elastic-plastic stiffness matrix must be assembled and a resulting system of equations solved. The clear physical meaning of the above procedure is retained at the cost of having to calculate the stiffness matrix at each computational step anew.

The concept of the method at the elemental stage is shown in Fig. 1.

The initial strain method is based on the principle that the stress increment of a typical step in the elastic-plastic analysis is known beforehand. Then the corresponding increment of plastic strain can be found from the slope of the stress-strain diagram of a uniaxial test. This plastic strain can then be used as initial strain and the elastic analysis of the given structure can be made with initial loads which are defined for each finite element in such a way that they supress the imposed initial strains in a purely elastic manner. The sequence of calculations is as follows:

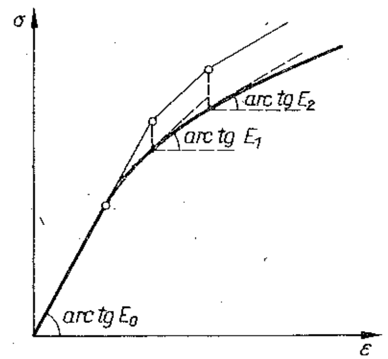


Fig. 1.

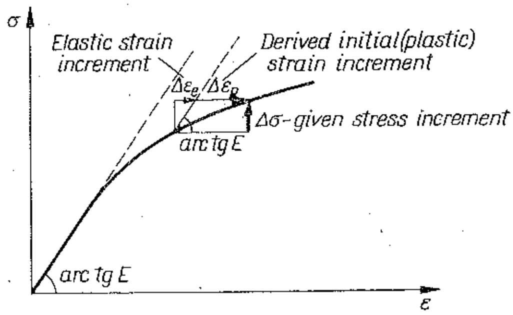


Fig. 2.

Having the incremental initial load, follow the global displacements of the structure accompanied by the element displacements. Then the total strains and their elastic parts are computed. Finally, the stresses which must be equal to those which have been adopted at the beginning of the step are calculated. Clearly iterative procedure must be developed to be repeated until the preset convergence is achieved. The advantage of the initial strain method against the tangential stiffness method consists in the fact that it is only necessary to make an elastic analysis of the structure; thus the stiffness matrices do not change (provided the displacements and strains are kept small). The concept of the method is schematically depicted in Fig. 2.

In the case of ideal plasticity the above philosophy becomes meaningless. The initial strain solution of perfectly plastic structure is impossible, since infinitely large plastic strains occur in this case.

The initial stress method is based on a similar principle as the latter one. Instead of iterative determination of the initial strains, the iteration is employed to find initial stresses. The total strain increment must be preassigned to calculate the initial stress increment which is to be equal to the difference between the actual elastic-plastic stress increment and the ideally elastic stress increment. Since the given total strain increment is in fact not known in advance, an iterative technique must be employed leading to the initial stress increment. This philosophy is illustrated in Fig. 3.

The applicability of the two variants briefly described above depends on how rapid is the convergence of the iterative procedures [5, 6]. Both variants have been here demonstrated in their original, simple version. They have been, and are still being, refined which will not be discussed here.

It is worth stressing that the tangential stiffness method and the initial load method are related to each other to a greater extent than it can be seen at the first sight. Basically, the first method requires an iteration process at the end of each step. On the other hand, the initial load method can appear to be weakly convergent when the load increments are not small enough. The ideal solution seems to be to work out an effective, mixed method.

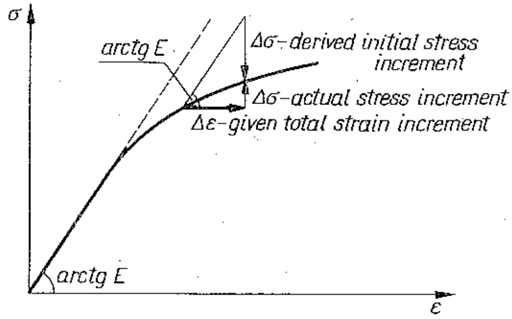

Fig. 3.

The aim of the paper is to show applicable numerical algorithms to deal with both physically and geometrically non-linear situations ([1]). Thus we will confine ourselves to the tangential stiffness method to solve the equilibrium problem described by a set of incremental equations for a discretized structural system. From this point of view, although some other methods exist such as the predictor-corrector schemes, Runge-Kutta procedures, chord stiffness matrix methods and others, it is felt that the presented tangential stiffness method (the Euler forward integration method, in actual fact) is advantageous due to a considerable accuracy achieved and the possibility to study the stability of the solution obtained [6]. It is believed that the method, though consuming a substantial computer time, is very appealing to the structural, stiffness-minded engineer.

## 2. Finite element description of elastic-plastic materials

To study structural problems in discretized mechanics, the matrix notation is usually employed as a convenient means leading to suitable computer programs [5, 7].

A set of all forces acting on a body can be shown as a force column vector $\{F\}_{n\times 1}$, $\{F\}^t=(F_1, F_2, ..., F_n)$ where $n$ is the total number of forces. $\{\ \}$ denotes a column matrix, $(\ )$ denotes a row matrix, the transpose of a matrix $F$ is indicated by $F^t$ and is here used to save space. With such a loading pattern a set of displacements associated is shown similarly as a displacement column vector $\{q\}_{n\times 1}$, $\{q\}^t=(q_1, q_2, ...$

([1]) The only paper which appeared in this country and dealt with similar problems seems to be [9]. However, the study of propagation of plastic zones was based on a simplifying assumption that the material is bilinearly elastic with very small Young's modulus at the second stage. So no constitutive relationship of plasticity was employed, the yielding elements being considered as very soft, elastic ones.

..., $q_n$). General relationship between the forces and displacements can always be shown in the form

$$\{F\}_{n\times 1}=[K]_{n\times n}\{q\}_{n\times 1} \tag{2.1}$$

where $[K]=\int [B]^t [D] [B] dV$ is the so-called stiffness matrix with the dimension $n\times n$. [ ] denotes, in general, a rectangular matrix. The elements of the stiffness matrix are the influence coefficients that give the force at one point of a structure associated with a unit displacement of the same or a different point. The stiffness matrix is clearly a square one. It depends on the mechanical properties of material, geometry of the considered structure and, in general, on the stress and strain states that develop in the structure. It is easy to see that in the case of linearly elastic bodies undergoing small deformations the stiffness matrix is fixed for a given situation. However, this is not so when elastic-plastic behaviour is studied.

Discretization of the continuum can be simply described as the process in which the given body is split up into a number of finite elements constituting an equivalent system. The act of subdivision, although liable to automation, remains very much a process based on the sound judgement on the part of the structural engineer. Thus, within an accuracy, a finite number of nodal points can be selected and (2.1) be understood as related to those nodes. In particular, the element stiffness matrix can first be established consisting of the coefficients of equilibrium equations derived from the material and geometric properties of an element and usually obtained with the help of the minimum potential energy principle. Thus the element stiffness matrix relates the nodal displacements of a single, typical element to the applied forces at the nodal points (nodal forces). The distributed load, if any, is replaced by equivalent concentrated forces at the nodes. As a result of this procedures, (2.1) represents a set of simultaneous linear algebraic equations. The displacements within an element are prescribed by means of a certain displacement model, called also a shape function. Inside the finite elements the equilibrium equations hold well together with displacement compatibility. The global equilibrium of the structure is replaced by the equilibrium requirements related to the nodes only—the equilibrium along the element boundaries can be sacrificed. The interelement continuity of displacements is observed by proper choice of the shape function. A polynomial is the most commonly used displacement model, since it is easy to handle the necessary mathematics such as differentiation and integration. Moreover, a polynomial of infinite degree clearly corresponds to an exact solution, provided it does exist in an analytical form. Thus, by truncating the infinite polynomial we can vary the degree of approximation as needed.

The constitutive equations for linear elasticity of isotropic bodies are in the tensorial notation

$$\sigma_{ij}=\frac{2G}{1-2\nu}[(1-2\nu)\varepsilon^e_{ij}+\nu\varepsilon^e_{kk}\delta_{ij}] \tag{2.2}$$

where $\varepsilon^e_{kk}$ is the trace of the elastic strain tensor and $\delta_{ij}$ denotes the Kronecker delta. In the shorthand matrix notation we have

(2.3) $$\boldsymbol{\sigma}=\frac{2G}{1-2\nu}\left[(1-2\nu)\mathbf{I}_6+\nu ee^t\right]\boldsymbol{\epsilon},$$

where $\mathbf{I}_6$ is a $6\times 6$ unit matrix and $e^t=(1\ 1\ 1\ 0\ 0\ 0)$ or, explicitly:

(2.4) $$\{\sigma\}=\begin{Bmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{zx}\end{Bmatrix}=\frac{2G}{1-2\nu}\begin{bmatrix}1-\nu & \nu & \nu & 0 & 0 & 0\\ & 1-\nu & \nu & 0 & 0 & 0\\ & & 1-\nu & 0 & 0 & 0\\ & & & \frac{1-2\nu}{2} & 0 & 0\\ & \text{(symmetry)} & & & \frac{1-2\nu}{2} & 0\\ & & & & & \frac{1-2\nu}{2}\end{bmatrix}\begin{Bmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_z\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{Bmatrix}.$$

In (2.4), shortened to $\{\sigma\}=[D^e]\{\varepsilon\}$, $[D^e]$ is the symmetrical $6\times 6$ constitutive elastic matrix.

In the case of plane stress, $\sigma_z=\tau_{yz}=\tau_{zx}=0$, the matrix $[D^e]$ takes the form

(2.5) $$[D^e]=\frac{2G}{1-\nu}\begin{bmatrix}1 & \nu & 0\\ & 1 & 0\\ \text{(sym.)} & & \frac{1-\nu}{2}\end{bmatrix}.$$

$[D^e]$ for plane strain can be shown similarly to be

(2.5′) $$[D^e]=\frac{2G}{1-2\nu}\begin{bmatrix}1-\nu & \nu & 0\\ & 1-\nu & 0\\ \text{(sym.)} & & \frac{1-2\nu}{2}\end{bmatrix}.$$

In elasto-plasticity we shall start form the Prandtl-Reuss relationships and assume that the total strain rate can be decomposed into an elastic part $\dot{\varepsilon}^e_{ij}$ and a plastic part $\dot{\varepsilon}^p_{ij}$,

(2.6) $$\dot{\varepsilon}_{ij}=\dot{\varepsilon}^e_{ij}+\dot{\varepsilon}^p_{ij}.$$

Now we can write

(2.7) $$\dot{\varepsilon}_{ij}=\frac{\dot{s}_{ij}}{G}+\frac{1-2\nu}{E}\dot{\sigma}\delta_{ij}+\lambda\frac{\partial F}{\partial\sigma_{ij}},$$

where $\dot{s}_{ij}=\dot{\sigma}_{ij}-\dot{\sigma}\delta_{ij}$ is the stress rate deviator, $\dot{\sigma}=\frac{1}{3}\dot{\sigma}_{kk}$ is the mean normal stress rate, $F$ is the left-hand side of a suitably adopted yield condition and $\lambda$ is an unknown parameter. The last term in (2.7) means that the plastic strain rate vector $\dot{\varepsilon}^p_{ij}$ is orthogonal to the yield hypersurface in the nine-dimensional stress space. On selecting the Huber-Mises yield condition in the form

(2.8) $$F=\frac{1}{2}s_{ij}s_{ij}-\frac{1}{3}\bar{\sigma}_0^2=0$$

or, explicitly,

(2.9) $$(\sigma_x-\sigma_y)^2+(\sigma_y+\sigma_z)^2+(\sigma_z-\sigma_x)^2+ +6(\tau_{xy}^2+\tau_{yz}^2+\tau_{zx}^2)-2\bar{\sigma}_0^2=0,$$

the scalar factor $\lambda$ can be determined. The quantity $\bar{\sigma}_0$ is initially equal to the yield point determined in a uniaxial tension test. The equations (2.7) can also be shown in a finite incremental manner to give

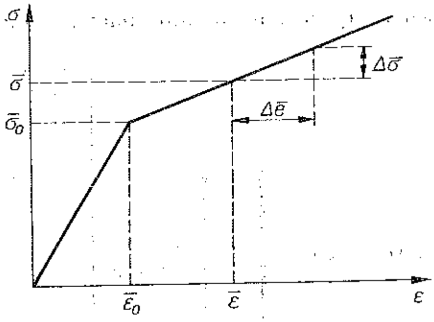

Fig. 4.

(2.10) $$\Delta\varepsilon_{ij}=\frac{\Delta s_{ij}}{G}+\frac{1-2\nu}{E}\Delta\sigma\delta_{ij}+\Delta\lambda\frac{\partial F}{\partial\sigma_{ij}},$$

where $\Delta e_{ij}=\Delta\lambda\frac{\partial F}{\partial\sigma_{ij}}$ is an increment of plastic strains.

For elasto-plastic strain-hardening materials, $\bar{\sigma}_0$ is a current yield point stress depending on the load history as depicted in Fig. 4. The $\bar{\sigma}-\varepsilon$ diagram of a uniaxial test yields the slope

(2.11) $$\zeta=\frac{\Delta\bar{\sigma}}{\Delta\bar{\varepsilon}}.$$

Plastic strain increment is

(2.12) $$\Delta\bar{e}_i=\Delta\lambda\frac{\partial F}{\partial\bar{\sigma}_i},\qquad \text{hence}\ \ \Delta\bar{e}=\frac{2}{3}\Delta\lambda\bar{\sigma}.$$

On differentiating the yield condition with respect to time and multiplying by the time increment $\Delta t$, we arrive at

(2.13) $$s_{ij}\Delta s_{ij}-\frac{2}{3}\bar{\sigma}\Delta\bar{\sigma}=0.$$

On observing that $s_{ij}\,\Delta s_{ij}=s_{ij}\,\Delta\sigma_{ij}$, (2.13) can be rewritten with the help of (2.11) and (2.12) to give

(2.14) $$s_{ij}\Delta\sigma_{ij}=\frac{4}{9}\zeta\bar{\sigma}^2\Delta\lambda.$$

Writing (2.2) incrementally and using (2.14) we obtain

(2.15) $$\Delta\sigma_{ij}s_{ij}=\frac{2G}{1-2\nu}[(1-2\nu)\Delta\varepsilon^e_{kk}s_{ij}+\nu\Delta\varepsilon^e_{kk}\delta_{ij}s_{ij}]=\frac{4}{9}\zeta\bar{\sigma}^2\Delta\lambda.$$

Since

$$\Delta\varepsilon^{e}_{ij}=\Delta\varepsilon_{ij}-\Delta\varepsilon^{p}_{ij}=\Delta\varepsilon_{ij}-\Delta e^{p}_{ij}=\Delta\varepsilon_{ij}-\Delta\lambda\frac{\partial F}{\partial\sigma_{ij}},$$

we arrive at the equation in $\Delta\lambda$

$$\frac{2G}{1-2\nu}\left[(1-2\nu)\left(\Delta\varepsilon_{ij}-\Delta\lambda\frac{\partial F}{\partial\sigma_{ij}}\right)s_{ij}\right]=\frac{4}{9}\zeta\bar{\sigma}^2\,\Delta\lambda .$$

Its solution provides

$$\Delta\lambda=\frac{2G\Delta\varepsilon_{ij}s_{ij}}{\frac{4}{9}\zeta\bar{\sigma}^2+2G\frac{\partial F}{\partial\sigma_{ij}}s_{ij}}=\frac{2G\Delta\varepsilon_{ij}s_{ij}}{\frac{4}{9}\bar{\sigma}^2(\zeta+3G)}. \tag{2.16}$$

In the matrix form we have

$$\Delta\lambda=\frac{1}{1-2\nu}\left(\frac{s_x}{s}\ \frac{s_y}{s}\ \frac{s_z}{s}\ \frac{s_{xy}}{s}\ \frac{s_{yz}}{s}\ \frac{s_{zx}}{s}\right)\begin{Bmatrix}\Delta\varepsilon_x\\ \Delta\varepsilon_y\\ \Delta\varepsilon_z\\ \Delta\gamma_{xy}\\ \Delta\gamma_{yz}\\ \Delta\gamma_{zx}\end{Bmatrix} \tag{2.17}$$

where

$$s=\frac{1}{2G(1-2\nu)}\left(\frac{4}{9}\zeta\bar{\sigma}^2+2G\frac{\partial F}{\partial\sigma_{ij}}s_{ij}\right)$$

and $s_x, \dots, s_{zx}$ are the components of the stress deviator.

Now we can determine the increments of the stress tensor components for elasto-plastic problems

$$\Delta\sigma_{ij}=\frac{2G}{1-2\nu}[(1-2\nu)\Delta\varepsilon_{ij}+\nu\Delta\varepsilon_{kk}\delta_{ij}]-2G\Delta\lambda\frac{\partial F}{\partial\sigma_{ij}}. \tag{2.18}$$

Remembering (2.4), (2.17) and representing $\partial F/\partial\sigma_{ij}$ in the form

$$\left\{\frac{\partial F}{\partial\sigma_{ij}}\right\}^t=(s_x\ s_y\ s_z\ 2s_{xy}\ 2s_{yz}\ 2s_{zx}),$$

we finally have

$$\Delta\sigma_{ij}=[D^e]\begin{Bmatrix}\Delta\varepsilon_x\\ \Delta\varepsilon_y\\ \Delta\varepsilon_z\\ \Delta\gamma_{xy}\\ \Delta\gamma_{yz}\\ \Delta\gamma_{zx}\end{Bmatrix}-\frac{2G}{1-2\nu}\begin{Bmatrix}s_x\\ s_y\\ s_z\\ 2s_{xy}\\ 2s_{yz}\\ 2s_{zx}\end{Bmatrix}\left(\frac{s_x}{s}\ \frac{s_y}{s}\ \frac{s_z}{s}\ \frac{s_{xy}}{s}\ \frac{s_{yz}}{s}\ \frac{s_{zx}}{s}\right)\begin{Bmatrix}\Delta\varepsilon_x\\ \Delta\varepsilon_y\\ \Delta\varepsilon_z\\ \Delta\gamma_{xy}\\ \Delta\gamma_{yz}\\ \Delta\gamma_{zx}\end{Bmatrix} \tag{2.19}$$

or, symbolically,

$$\Delta\sigma_{ij}=[D^e]\Delta\varepsilon_{ij}-[D^p]\Delta\varepsilon_{ij}=[D^{ep}]\Delta\varepsilon_{ij}. \tag{2.20}$$

The matrix $[D^{ep}]$ is called the constitutive elastoplastic matrix and takes the form

$$[D^{ep}]= \tag{2.21}$$

$$=\frac{2G}{1-2\nu}\begin{bmatrix} 1-\nu-\frac{s_x^2}{s} & \nu-\frac{s_x s_y}{s} & \nu-\frac{s_x s_z}{s} & -\frac{s_x s_{xy}}{s} & -\frac{s_x s_{yz}}{s} & -\frac{s_x s_{zx}}{s} \\ & 1-\nu-\frac{s_y^2}{s} & \nu-\frac{s_y s_z}{s} & -\frac{s_y s_{xy}}{s} & -\frac{s_y s_{yz}}{s} & -\frac{s_y s_{zx}}{s} \\ & & 1-\nu-\frac{s_z^2}{s} & -\frac{s_z s_{xy}}{s} & -\frac{s_z s_{yz}}{s} & -\frac{s_z s_{zx}}{s} \\ & & & \frac{1-2\nu}{2}-\frac{s_{xy}^2}{s} & -\frac{s_{xy} s_{yz}}{s} & -\frac{s_{xy} s_{zx}}{s} \\ & \text{(symmetry)} & & & \frac{1-2\nu}{2}-\frac{s_{yz}^2}{s} & -\frac{s_{yz} s_{zx}}{s} \\ & & & & & \frac{1-2\nu}{2}-\frac{s_{zx}^2}{s} \end{bmatrix},$$

where, specifically, $s_x=(2\sigma_x-\sigma_y-\sigma_z)/3,\ \ldots,\ s_{xy}=\tau_{xy},\ \ldots,$

$$s=\frac{1}{2G(1-2\nu)}\left[\frac{4}{3}\bar{\sigma}^2\left(\frac{1}{3}\zeta-G\right)\right].$$

In the case of plane stress, $\sigma_z=\tau_{yz}=\tau_{zx}=0$, similar derivation leads to

$$[D^{ep}]=\frac{E}{1-\nu^2}\begin{bmatrix} 1-\frac{(s_x+\nu s_y)^2}{s} & \nu-\frac{(s_x+\nu s_y)(s_y+\nu s_x)}{s} & -\frac{(1-\nu)(s_x+\nu s_y)s_{xy}}{s} \\ & 1-\frac{(s_y+\nu s_x)^2}{s} & -\frac{(1-\nu)(s_y+\nu s_x)s_{xy}}{s} \\ \text{(symmetry)} & & \frac{1-\nu}{2}-\frac{(1-\nu)^2 s_{xy}^2}{s} \end{bmatrix}, \tag{2.22}$$

where

$$s_x=(2\sigma_x-\sigma_y)/3, \quad s_y=(2\sigma_y-\sigma_x)/3, \quad s_{xy}=\tau_{xy},$$

$$s=s_x^2+2\nu s_x s_y+s_y^2+2(1-\nu)s_{xy}^2+\frac{4}{9}\,\frac{1-\nu^2}{E}\,\bar{\sigma}^2\zeta.$$

For plane strain, the constitutive elastic-plastic matrix can readily be shown to take the following form:

$$(2.23)\qquad [D^{ep}]=2G\begin{bmatrix} \frac{1-\nu}{1-2\nu}-\frac{s_x^2}{s} & \frac{\nu}{1-2\nu}-\frac{s_x s_y}{s} & -\frac{s_x s_{xy}}{s} \\ & \frac{1-\nu}{1-2\nu}-\frac{s_y^2}{s} & -\frac{s_y s_{xy}}{s} \\ \text{(symmetry)} & & \frac{1}{2}-\frac{s_{xy}^2}{s} \end{bmatrix}.$$

## 3. Description of program and computational methods

As can be seen from the flow chart, Fig. 5, the program is divided into a number of main parts.

Steps 1-5 constitute a standard, purely elastic solution of the problem and will not be described here in detail. Some particulars can be found in [8] (2).

Step 6 — In this part of the program stresses are first computed resulting from step 3 and added to the already existing stresses, if any. Then the yield condition is checked in each element: the stress intensity is calculated according to the Huber-Mises yield criterion and compared with the prescribed yield point of the material. If the yield condition is satisfied the element is at yield and the calculation of constitutive elastic-plastic matrix $[D^{ep}]$ follows for this element the formula (2.21) in the three-dimensional case, (2.22) for plane stress, (2.23) for plane strain. Then the test load factor is computed from the condition that the point $A$ of the stress vector in the stress space (Fig. 6) must not be too far from the yield surface $F=0$. For this purpose a suitable second degree algebraic equation is to be solved.

If the yield condition is not reached and elements remain elastic, in each element a ratio of the yield point stress to the stress intensity is calculated. Then an element is selected in which this coefficient attains the least value. From this a suitable test load factor is deduced bringing the next element to yield. Now the smaller load increment (from the two resulting from the elastic and the plastic paths) is selected and fed into the next step.

Step 7 — Displacement increments corresponding to the test load factor determined in step 6 are calculated.

Step 8 — Corresponding stress increments followed by the total stresses are computed. The total stress is represented as a vector $OA$, Fig. 6. Now, this total stress is reduced to occupy the new position $OB$. This reduced total stress is stored to start the next computational stage.

Step 9 — The problem of uniqueness is here studied. The question must be answered whether or not the conditions for loading process are satisfied. To this end in each yielding element the scalar parameter $\Delta\lambda$ which is proportional to the plastic work done must be calculated according to (2.16). This parameter should

(2) An improved program in ALGOL-1204, used here, has recently been written by A. Radwański and M. Witkowski.

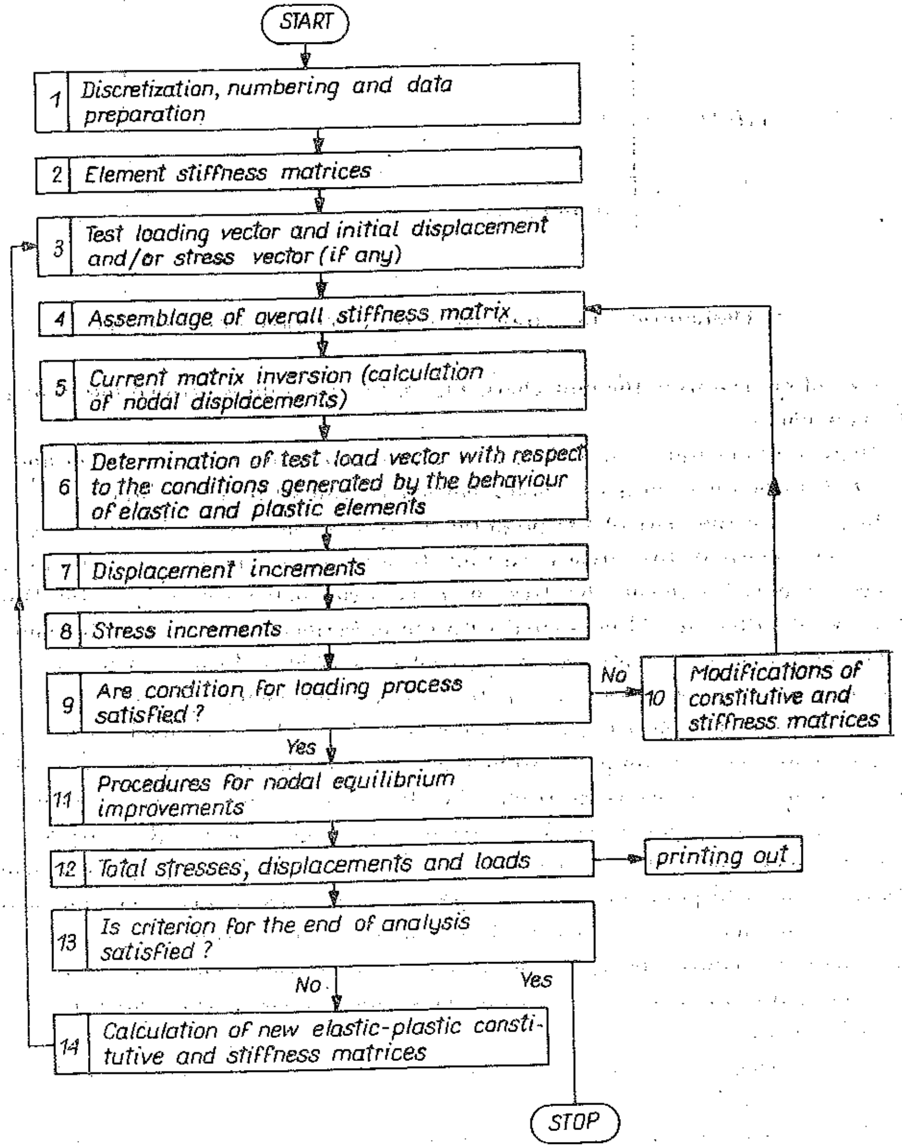


Fig. 5.

appear to be non-negative in order to justify the assumption made at the beginning of a given step that the element considered is undergoing an active plastic process. If $\Delta\lambda$ turns out to be negative the step 10 follows.

Step 10 — The calculations must be repeated, starting from step 4. In all the elements in which $\Delta\lambda<0$ the elastic constitutive and stiffness matrices must be introduced.

Step 11 — Procedures for nodal equilibrium improvement are incorporated here. In the tangential stiffness method the load-displacement curve can happen to depart considerably from the true one, especially when the structure is split up into a large number of finite elements. There exists a number of methods to improve the accuracy of the calculations from which those seem to be preferable which are based on the initial load concept. In particular, a suitable iterative process can ideally be carried out at the end of each step. To save computing time, this process can be performed at every few steps. Similar corrections are introduced in the incremental analysis of geometrically non-linear problems. Another method to improve the current matrix consists in the calculation of an average matrix resulting from the matrices at the beginning and at the end of the step.

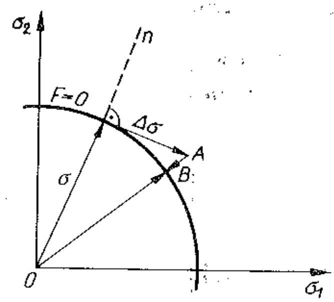

Fig. 6.

Step 12 — Total stresses, displacements and loads are calculated and printed out for the current situation.

Step 13 — A suitable criterion must be formulated at what stage the computational process is to be terminated. For instance, a criterion can be imposed that the displacement of a certain reference point becomes large enough to indicate the ultimate behaviour of an elastic-plastic structure or that the slope of the load-displacement curve becomes small enough. If the adopted criterion is not satisfied the computational process is continued via step 14, starting again from the step 3. If it is, the analysis is terminated.

Step 14 — New elastic-plastic constitutive and stiffness matrices are calculated.

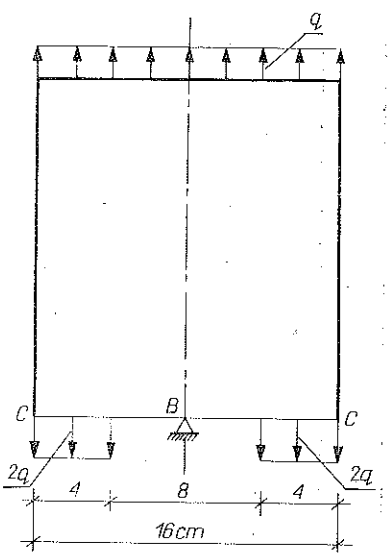

Fig. 7.

## 4. Example

As a test example a rectangular plane stress region is considered made of elastic-perfectly plastic material and subjected to the self-equilibrating load as shown in Fig. 7. The structure is divided into triangular elements within which a linear shape function is adopted. Due to the symmetry a half of the structure is dealt with. The nodes lying on the symmetry axis are constrained against the horizontal motion, point $B$ (Fig. 7) being completely fixed. Displacements of all nodes are calculated, but the behaviour of the structure is represented diagrammatically by means of the reference point $C$ as undergoing the largest vertical displacement. Four divisions are used, numbered I, II, III, IV (starting from the coarsest) in order to compare the accuracy of numerical results. Moreover, the coarsest mesh was used for the purpose of manual calculations as a check on the correctness of the computer program.

### 4.1. *Mesh I*

The considered half of the structure is divided into 16 elements having 30 degrees of freedom, the mesh being made finer towards the bottom edge where a certain stress concentration is expected to take place. The numerical calculations were executed twice — for large load increments (resulting from the yielding of consec-

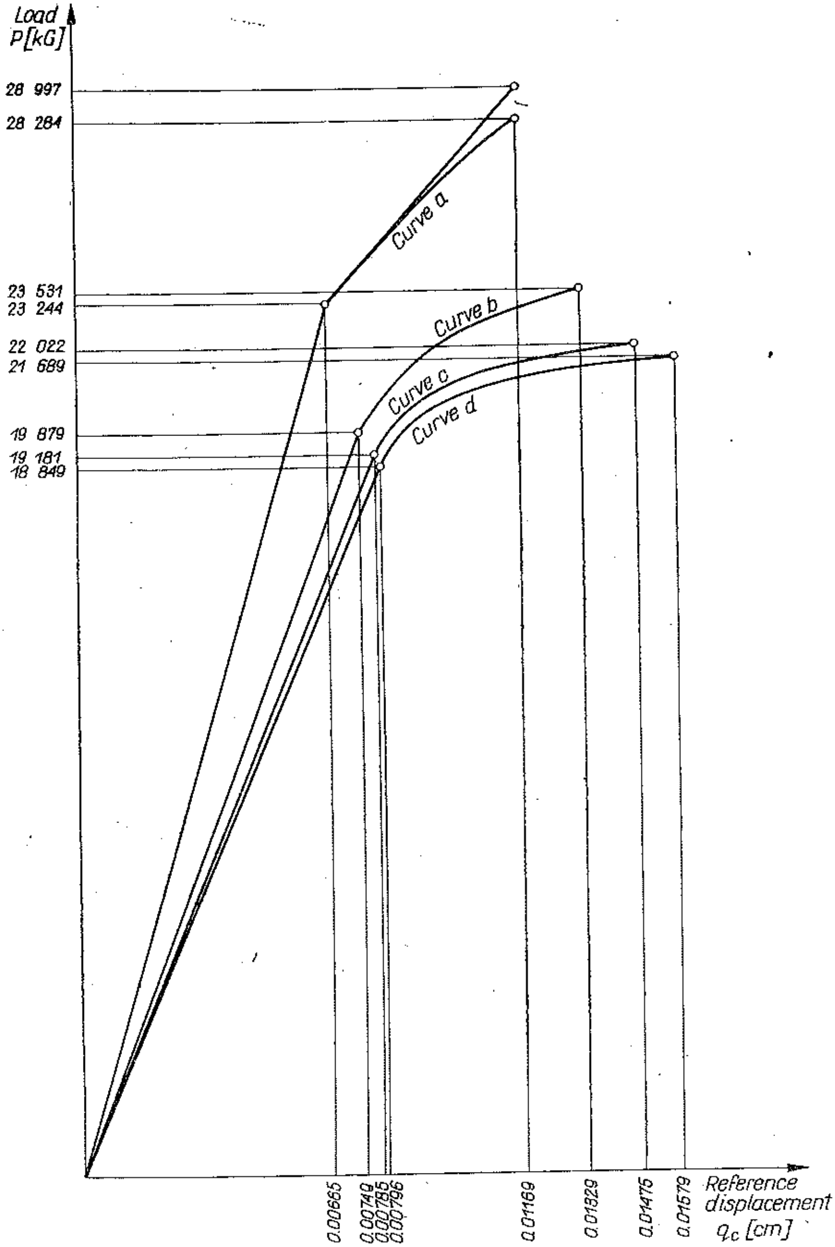


Fig. 8.

utive elements) and for small load increments — fractions of the former increments — in order to exceed less the yield surface at each step and obtain a smoother load-displacement diagram in its elastic-plastic part. Fig. 8, curve $a$ shows the obtained

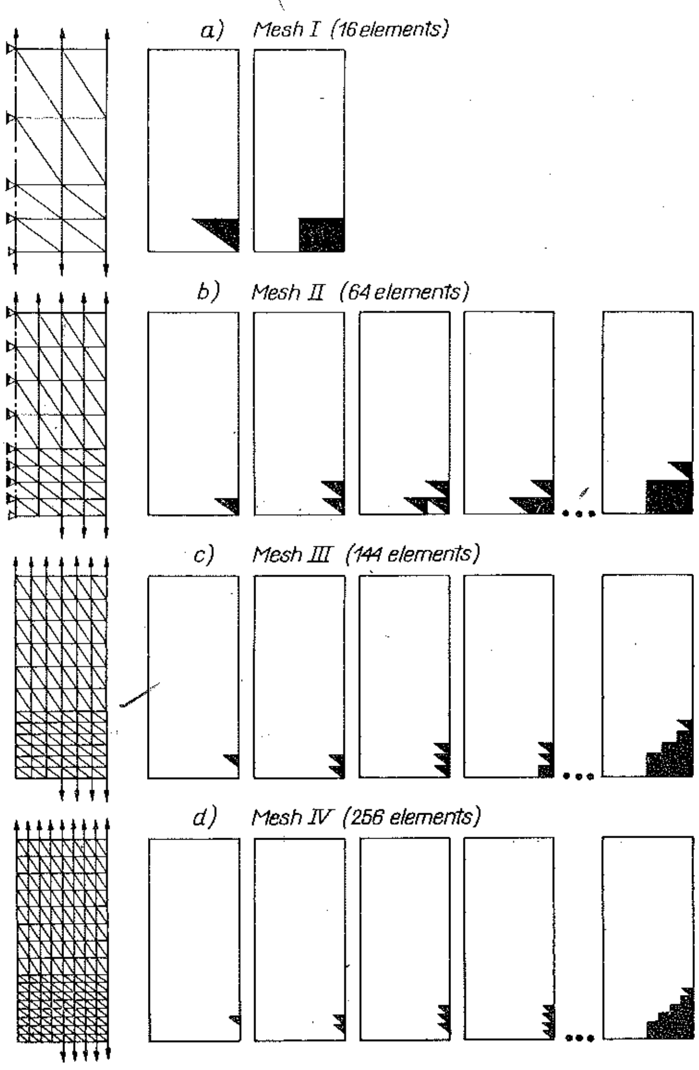


Fig. 9.

results in the load-displacement of the point $C$-plane. The sequence of yielding elements for large load increments is shown in Fig. 9a.

The upper branch of curve $a$ corresponds to large increments, the lower one to small increments.

### 4.2. Mesh II

The half of the structure is split up into 64 elements having 90 degrees of freedom. Only such load increments were applied that were bringing the consecutive elements to yield. There was 11 load increments to cause a substantial increase in the displacement of the reference point $C$. The results are shown in Fig. 8, curve $b$. The sequence of yielding elements is shown in Fig. 9b.

### 4.3. Mesh III

The half of the structure is divided into 144 elements and has 182 degrees of freedom. There was 19 load increments, each causing the next element to yield. The result is shown in Fig. 8, curve $c$. The propagation of plastic region can be seen in Fig. 9c.

### 4.4. Mesh IV

The half of the structure is divided into 256 elements and has 306 degrees of freedom. The obtained load-displacement diagram is shown in Fig. 8, curve $d$. The development of plastic zone is seen in Fig. 9d.

### 4.5. Technical data

The calculations were carried out on the ODRA-1204 computer with 16 K, supplied with 2 drum memories, 36K each. The ALGOL language was used. Due to limited operation memory the program is divided into 4 segments. The computation times for each mesh were, respectively; 12′ 14″, 1h 11′39″, 3 h 17′33″, 8 h 44′44″. The program enables the computational process to be carried out in two ways: either automatically — when the magnitude of the load increment results from the fact that the next element is brought to yielding or the yield surface is exceeded by a prescribed amount — or the load increments can be controlled manually with the use of monitor — when smaller load increments are required to obtain smoother load-displacement curve.

## 5. Concluding remarks

The applied procedure makes it possible to study the propagation of plastic zones up to an instant at which the intensity of limit load is practically reached. Any plane stress configuration can be dealt with by the program. In plane strain situations the matrices must only be replaced by different ones as shown in Sec. 1. In Fig. 8 the convergence of results as depending upon the number of elements is demonstrated.

## References


1. G. G. Pope, *A discrete element method for analysis of plane elasto-plastic strain problems*, Aeronaut. Q., **17**, 83, 1966.
2. J. H. Argyris, *Elasto-plastic matrix displacement analysis of three-dimensional continua*, J. R. Aeronaut. Soc., **69**, 1965.
3. J. H. Argyris, D. W. Scharpf, J. B. Spooner, *Die elastoplastische Berechnung von allgemeiner Tragwerken und Kontinua*, Ing. Arch., **37**, 1969.
4. O. C. Zienkiewicz, S. Valliapan, I. P. King, *Elasto-plastic solutions of engineering problems, initial stress, finite element approach*, Int. J. Num. Meth. Eng., **1**, 1969.
5. J. H. Argyris, D. W. Scharpf, *Methods of elasto-plastic analysis*, J. Appl. Math. Phys., **23**, 1972.
6. J. A. Stricklin, W. E. Haisler, A. A. von Riesemann, *Evaluation of solution procedures for material and for geometrically non-linear structural analysis*, AIAA Journal, **11**, 3, 1973.
7. Y. Yamada, N. Yoshimura, T. Sakurai, *Plastic stress-strain matrix and its application for the solution of elastic-plastic problems by the finite element method*, Int. J. Mech. Sci., 10, 1968.
8. J. Szmelter, M. Dacko, S. Dobrociński, M. Wieczorek, *Programy metody elementów skończonych*, Arkady, Warszawa 1973.
9. J. Kapkowski, *Granice obszarów plastycznych w rozciąganych elementach z karbem lub otworem*, Mechanika Teoretyczna i Stosowana, 3, 1970.



## Streszczenie

### SPRĘŻYSTO-PLASTYCZNA ANALIZA PŁASKIEGO STANU NAPRĘŻENIA METODĄ ELEMENTÓW SKOŃCZONYCH

W pracy zastosowano metodę elementów skończonych do statycznej analizy tarczy z materiału sprężysto-idealnie plastycznego. Tarczę podzielono na elementy trójkątne i zastosowano liniową funkcję kształtu. Sformułowano podstawowe zależności dla materiału sprężysto-plastycznego w zapisie macierzowym. Podano sposób obliczania macierzy sprężysto-plastycznej dla materiału spełniającego warunek plastyczności Hubera-Misesa-Henckiego w przypadku przestrzennego stanu naprężenia, płaskiego stanu naprężenia oraz płaskiego stanu odkształcenia. W obliczeniach numerycznych przyjęto materiał idealnie plastyczny, natomiast wyprowadzone związki są ważne dla materiału ze wzmocnieniem.

Omówiono podstawowe algorytmy rozwiązania otrzymanego nieliniowego układu równań. Przedstawiono metodę zmiennej sztywności oraz początkowych obciążeń węzłowych, w szczególności jej wariant — metodę naprężeń początkowych. W trakcie obliczeń numerycznych badano możliwość wystąpienia procesu odciążenia w poszczególnych elementach tarczy podczas obciążenia zewnętrznego całej konstrukcji. Podano algorytm postępowania, rozwiązano przykład numeryczny.



## Резюме

### АНАЛИЗ ПЛОСКОГО, УПРУГО-ПЛАСТИЧЕСКОГО НАПРЯЖЕННОГО СОСТАЯНИЯ МЕТОДОМ КОНЕЧНЫХ ЭЛЕМЕНТОВ

В работе применен метод конечных элементов к статическому анализу пластинки из упруго-идеально пластического материала. Применяются треугольные элементы и линейная аппроксимирующая функция. Определены, в матричной формулировке, основные зависимости для упруго-пластического тела. Построен метод расчета упруго-пластической матрицы, для тела подчиняющегося условию пластичности Губера-Мизеса-Генке, в случае пространственного напряженного состояния, плоского напряженного сотояния и плоской деформации.

Численные расчеты проведены для идеально-пластического тела, но полученные зависимости применимы для материала с упрочнением. Рассмотрены основные алгоритмы решения полученной нелинейной системы уравнений.

Обсужден метод переменной жесткости и начальных узловых сил с его частным случаем — методом начальных напряжений. Численные расчеты связаны были с исследованием возможности появления разгруженного состояния в отдельных элементах пластинки сопутствующего внешней нагрузке конструкции. Работа содержит соответствующие алгоритмы, как и численные расчеты.

TECHNICAL UNIVERSITY OF WARSAW, FACULTY OF CIVIL ENGINEERING
POLISH ACADEMY OF SCIENCES, INSTITUTE OF FUNDAMENTAL TECHNOLOGICAL RESEARCH

*Received October 4, 1974.*