Energy, Mines and Resources Canada
Énergie, Mines et Ressources Canada

CANMET

Canada Centre for Mineral and Energy Technology
Centre canadien de la technologie des minéraux et de l'énergie

Mining Research Laboratories
Laboratoires de recherche minière

Canada

PCEPFE USER'S GUIDE - A 2-D ELASTIC-PLASTIC FINITE ELEMENT STRESS ANALYSIS PACKAGE USING A PERSONAL COMPUTER (PC Version 1.0, 1988)

Y.S. Yu and N.A.Toews

MRL 88-95(TR)

PCEPFE USER'S GUIDE - A 2-D ELASTIC-PLASTIC FINITE ELEMENT STRESS ANALYSIS PACKAGE USING A PERSONAL COMPUTER (PC Version 1.0, 1988)

by

Y.S. Yu and N.A. Toews

PCEPFE USER'S GUIDE - A 2-D ELASTIC-PLASTIC FINITE ELEMENT STRESS ANALYSIS PACKAGE USING A PERSONAL COMPUTER (PC Version 1.0, 1988)

by

Y.S. Yu* and N.A. Toews*

ABSTRACT

PCEPFE and its companion programs, a software package which was developed based on the finite element technique using a personal computer, is described. The program is capable of performing static, plane stain analyses of stresses and deformations. Assuming an elastic-perfectly plastic material following a generalized Mohr-Coulomb yield criterion and incremental theory of plasticity, it can simulate the progressive failure of a mine structure. Sequence of excavation and/or construction, such as backfill in mine stopes, can be easily modelled.

This report provides an overview of the PCEPFE system and describes the functions of it component programs. Data input instructions are described.

Key words: finite element, elastic plastic, stresses, displacements, personal computer.
* Research Scientists, Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

GUIDE D'UTILISATION DE PCEPFE: PROGICIEL D'ANALYSE BIDIMENSIONAL PAR ELEMENTS FINIS DES CONTRAINTES ELASTIQUES/PLASTIQUES AU MOYEN D'UN ORDINATEUR PERSONNEL (PC Version 1.0, 1988)

par

Y.S. Yu* et N.A. Toews*

RÉSUMÉ

PCEPFE et ses programmes associés, progiciel mis au point à partir de la technique des éléments finis appliquée sur ordinateur personnel, sont décrits. Le programme est capable d'effectuer des analyses de constraintes et de déformation à patir de déformations planes statiques. Basé sur l'hypothèse voulant qu'un matériau élastique et parfaitement plastique obéit à une limite d'élasticité de Mohr-Coulomb généralisée et à une théorie de la plasticité progressive, il peut simuler la rupture progressive d'un ouvrage minier. Une succession de travaux d'excavation/construction comme le remblayage dans un chantier de mine peut être facilement modélisée.

Le présent raport donne un aperçu du systèm PCEPFE et décrit les fonctions de ses programmes constitutifs. Les instructions pour l'entrée des données sont décrites.

Mots-clé: élément finis, élastique-plastique, contraintes, déplacements, ordinateur personnel.

* Chercheurs scientifiques, Laboratoires de recherche minière, CANMET, Énergie, Mines et Ressources Canada, Ottawa.

CONTENTS

FIGURES

INTRODUCTION

The program PCEPFE is an Elastic Plastic Finite Element stress analysis program for two-dimensional structures in a Personal Computer environment. This program was substantially based on the previous work of Sandhu, Wu and Hooper [1,2]. It was first modified to run on the departmental main frame computer and then on a VAX-11/750 mini computer [3]. Recently, this nonlinear finite element program was further modified to run in a personal computer environment.

The computer code and its elastic plastic constitutive relationships employed in the analysis have been check thoroughly and verified by comparing the results with available analytical solutions [4]. To run PCEPFE efficiently, the interface program - EPFEC and the mesh-generating system - MSHGEN have also been modified. In addition, a post-processor PCPLOT was also developed using GSS*GKS graphic software.

The mesh-generating system MSHGEN and MSHPLT used in the PCEPFE program were extensively based on previous work [5]. The post-processor was partially based on earlier work which required the use of a Calcomp plotter in carrying out the analysis of two-dimensional finite-element results [6].

This report provides documentation and instruction on procedures associated with preparing the input data, checking the input data, running the programs and interpreting the output data from the finite element analysis.

SYSTEM OVERVIEW

Structural analysis using the finite element technique necessary involves large amounts of input and output data. Therefore, in order to speed up data preparation and analysis pre- and post-processors are required. In addition, to meet analytic requirements, a number of companion programs for use with PCEPFE were developed.

This section provides an overview to the PCEPFE software package, and a flow diagram has been produced to show the functions of each companion program and their relationship within the PCEPFE system.

The main program and its companion programs in the system were originally developed and tested on CDC Cyber 74 and VAX-11/750 computers with a Calcomp plotter [3]. In 1988, they were modified in accordance with Fortran 77 standard to run on a personal computer under MSDOS operating system. The plotting device for the pre- and post-processors associated the PC version will function with a color hard-copy device. All basic plotting routines used in this program are GKS*GKS software. In addition, the pre- and post-processors are now fully interactive and menu driven.

The PCEPFE software system consisting of finite element program PCEPFE and a number of companion programs is described briefly below.

Figure 1 summarizes the follow of the system and indicates the inter-relationship between the companion programs within the system.

Program PCEPFE:

As mentioned, PCEPFE is a static, nonlinear finite element program for analysis of two-dimensional structures (plane strain). Initial stresses, simulation of mining sequences (excavation and or backfill), and arbitrary distributed loading, gravity loading as well as concentrated force loading can be handled by this program.

Program MSHGEN:

MSHGEN is the mesh generator that produces the major portion of the finite input data for PCEPFE. The program MSHGEN closely follows the concepts and terminology introduced by Zienkiewicz and Phillips [5].

Some features of the MSHGEN program are:

(a) Quadrilateral elements are generated;

(b) Linearly varying pressures can be generated on element sides;

(c) Mesh grading can be achieved; and

(d) Extensive error checking of input, including a printer-plot for visual inspection is built into the program.

One limitation of MSHGEN, at present, is its inability to generate one dimensional joint elements.

Program EPFEC:

MSHGEN program produces only part of the input data, such as nodal point coordinates, element data and pressure data, etc., required by the finite element program PCEPFE. Other necessary information such as material properties, initial stresses and concentrated nodal forces are absent. EPFEC merges this information with the output file of MSHGEN, called genout.dat, to produce an input file (epfein.dat) acceptable to PCEPFE.

Program MSHPLT:

Finite element analysis for mine structures or other types of geotechnical structures usually involves large and complicated geometries or configurations. The discretization and

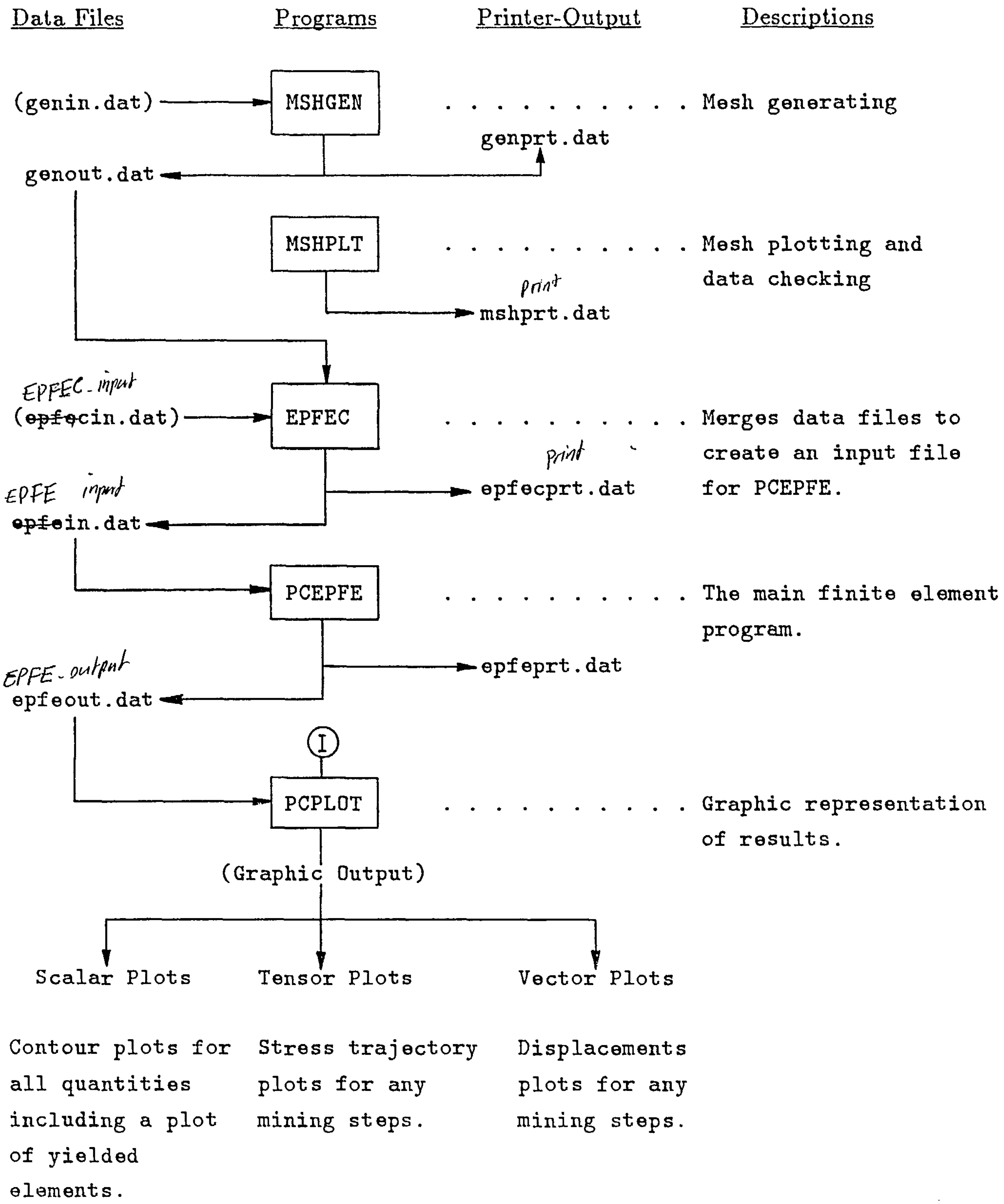


Fig. 1 - PCEPFE software system flow diagram
(data files in brackets are user prepared, I denotes interactive input)

proper grading of a finite element mesh is an integral part of the stress analysis process.

The best way to check whether a generated finite element mesh is adequate or not is to plot the mesh and inspect it visually. MSHPLT is a mesh plotting program especially designed for use in conjunction with the mesh generating program MSHGEN.

Program PCPLOT:

PCPLOT is a post-processor for interpreting PCEPFE output graphically. It converts PCEPFE output data (stresses and displacements) results of any two dimensional finite element analysis. Three types of plots can be generated from PCPLOT, and they are described as follows:

(a) *vectors*;

(b) *tensors*; and

(c) *scalars*.

The *vector* plot was designed to illustrate displacements or other vector data. The *tensor* plot produces a representation of two principal values of second-order tensors, such as the stress trajectories of principal stresses for a plane structure. The *scalar* plot can be used for any quantity dependent on two independent variables, X and Y, for example, a contour plot.

PCEPFE - FINITE ELEMENT PROGRAM

Program Capability:

The program PCEPFE is capable of performing static, plane strain analyses of stresses and deformations. Assuming an elastic-perfectly plastic material following a generalized Mohr-Coulomb yield criterion and incremental plasticity, it can simulate the progressive failure of a mine structure. Sequences of construction (such as back-filling in mine stopes) and or excavation can be easily simulated. Arbitrary initial stresses can be input. In addition, a one dimensional joint element with prescribed stresses could be included in the analysis. However, the companion program MSHGEN - a mesh generating system has no capability to generate one-dimensional elements as yet.

The mathematical formulation incorporated in this program has been discussed in detail in References [1,2]. The elastic-plastic constitutive relations employed for the nonlinear analysis have also been derived independently and given in Reference [4].

Program Structure:

The computer program PCEPFE and its companion programs were written in in accordance with Fortran 77 standard. A number of logical files are used in PCEPFE. Logical files, `'epfein.dat'`, 'epfeprt.dat' and `'epfeout.dat'` are the files, respectively, for input data, print output and output of results stored for post-processing purposes. the calculated stresses and displacements of last iteration at each incremental mining step or subproblem are also stored on the logical file `'epfeout.dat'`.

The program consists of one main program and eight (8) subroutines. An interface program, `EPFEC`, which accepts data from the meshing generating program `MSHGEN` and prepares an input file for `PCEPFE`, is also described.

Main Program EPFE:

The main program `EPFE` allocates the core requirements based on some of the basic control parameters. The control parameters are the number of nodal points (NUMNP), number of elements (NUMEL), number of material types (NUMMAT), number of pressure records (NUMPC) and the maximum number of elements to be removed or added from the model at any incremental mining step (NMR).

The memory requirements can be easily altered by changing the dimension array of AA and the the value of MTOT. The length of array AA must equal to MTOT. This is accomplished by changing the follow two lines in the main program to:

```
COMMON AA (n)
MTOT = n
```

The value of MTOT is determined by the following formula:

$$
\begin{aligned}
MTOT \geq\ & 7 \times NUMNP + 16 \times NUMEL + 6 \times NUMMAT + 4 \times NPC \\
& + 3 \times NMR \\
& + MAX(2 \times NB + 2 \times NB \times MBAND, 2 \times NUMNP \\
& + 5 \times NUMEL + 100)
\end{aligned}
$$

where

$$
\begin{aligned}
NPC &= MAX(1, NUMPC) \\
MBAND &= half \;\; bandwidth \\
&= 2 \times (max. \;\; nodal \;\; point \;\; difference + 1) \\
NB &= MAX(MBAND, NUMBLK) \\
NUMBLK &= (2 \times NUMNP - 1)/MBAND + 1 \\
MAX &= the \;\; maximum \;\; value \;\; of \;\; the \;\; two \;\; quantities
\end{aligned}
$$

Subroutine INPT

The subroutine INPT reads in most of the input data required by the system. The data includes:

(a) Material properties,

(b) Co-ordinates of the nodal points and their associated constraints, and specified loads or displacements,

(c) Elements parameters and the associated initial state of stresses; the initial stresses may be input or computed by the program.

The maximum half bandwidth (MBAND) for the model is evaluated and dimensions of blocks for generation and storage of the system stiffness matrix are defined. After defining these controls, the incremental structure (subproblem) is analyzed in steps. Information required for each incremental step, such as number of nodes and elements to be removed or added, etc., are read.

All this information can be prepared by the interface program EPFEC providing the mesh-generating system is used. The details on the use of the software system will be discussed later.

Subroutine SOLVE

This routine is concerned with obtaining the stresses and displacements at a given stage of the incremental structure allowing for progressive failure. To handle the progressive failure, the solution process traces a sequence of elements reaching the yield point under the load applied. This sequence of yielding is associated with a portion of load application and is described in the print output as successive approximation with increasing 'stress ratio'. The procedure consists of applying the total load and scaling it according to the minimum rate of load increment needed to ensure an excursion to yield by one element at a time.

Subroutine ONED

Subroutine ONED generate the element stiffness for one-dimensional elements as well as the forces corresponding to the unbalanced stress defined by the difference in the total load application and the load taken by the system in the current approximation in progressive failure analysis.

Subroutine QUAD

Subroutine QUAD generates the stiffness matrix for the two-dimensional elements. For the current load increment, an element is either in a state of elastic or in a state of plastic which has reached yielding. QUAD calls subroutine STRSTR to obtain the stress-strain relationship either for the elastic case or the plastic domain depending on the current state of stress in the element.

Subroutine STRSTR

Subroutine STRSTR defines the stress-strain relationship for the elastic or plastic material as the case may be.

The stress strain relationship for the elastic case under a plane strain condition is given by:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_x \\ \gamma_{xy} \end{Bmatrix}$$

and

$$\sigma_z = \nu(\sigma_x + \sigma_y)$$

where E and ν are the Young's modulus and Poisson's ratio, respectively, for the isotropic elastic material.

For the plastic domain the stress-strain relationship is given by:

$$\begin{Bmatrix} \dot{\sigma}_x \\ \dot{\sigma}_y \\ \dot{\tau}_{xy} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{12} & D_{22} & D_{23} \\ D_{31} & D_{32} & D_{33} \\ D_{41} & D_{42} & D_{43} \end{bmatrix} \begin{Bmatrix} \dot{\varepsilon}_x \\ \dot{\varepsilon}_x \\ \dot{\gamma}_{xy} \end{Bmatrix}$$

where

$$
\begin{aligned}
D_{11} &= 2G(1 - h_2 - 2h_1\sigma_x - h_3\sigma_x^2) \\
D_{12} &= -2G[h_2 + h_1(\sigma_x + \sigma_y) + h_3\sigma_x\sigma_y] \\
D_{21} &= D_{12} \\
D_{13} &= -2G(h_1\tau_{xy} + h_3\sigma_x\tau_{xy}) \\
D_{31} &= D_{13} \\
D_{22} &= 2G(1 - h_2 - 2h_2\sigma_y - h_3\sigma_y^2) \\
D_{23} &= -2G(h_1\tau_{xy} + h_3\sigma_y\tau_{xy}) \\
D_{32} &= D_{23} \\
D_{33} &= 2G(0.5 - h_3\tau_{xy}^2) \\
D_{41} &= -2G(h_2 + h_1(\sigma_x + \sigma_z) + h_3\sigma_x\sigma_y] \\
D_{42} &= -2G(h_2 + h_1(\sigma_y + \sigma_z) + h_3\sigma_y\sigma_z] \\
D_{43} &= -2G(h_1\tau_{xy} + h_3\tau_{xy}\sigma_z]
\end{aligned}
$$

and

$$
\begin{aligned}
2G &= \frac{E}{(1+\nu)} \\
h_1 &= \frac{0.5h_4}{h_5 J_2^{0.5}} \\
h_2 &= \frac{h_4 h_6}{h_5} - \frac{\nu}{(1-\nu)}\frac{K}{h_5 J_2^{0.5}} \\
h_3 &= \frac{0.5}{h_5 J_2} \\
h_4 &= 3\alpha\frac{K}{G} - \frac{J_1}{3J_2^{0.5}} \\
h_5 &= 1 + 9\alpha^2\frac{K}{G} \\
h_6 &= \alpha - \frac{J_1}{6J_2^{0.5}} \\
K &= \frac{E}{3(1-2\nu)} \\
&= bulk \quad modulus \\
J_1 &= (\sigma_x + \sigma_y + \sigma_z) \\
&= first \quad invariant \quad of \quad the \quad stress \quad tensor \\
J_2 &= \frac{1}{6}\{(\sigma_x - \sigma_y)^2(\sigma_y - \sigma_z)^2(\sigma_z - \sigma_x)^2\} + \tau_{xy}^2 \\
&= second \quad invariant \quad of \quad the \quad stress \quad tensor
\end{aligned}
$$

Subroutine MODIFY

Subroutine MODIFY modifies the stiffness matrix for the prescribed boundary conditions. The modified matrix is returned to SOLVE.

Subroutine STRESS

The basic theory and technique employed for analyzing progressive failure in the elastic-plastic continuum have been described in detail in references [1,2]. However, a flow diagram for the subroutine STRESS is included for better understanding the logic of this subroutine (Appendix A)

Sign Convention for Stresses and System of Units:

The continuum mechanics sign convention (tensile stress positive and compressive stress negative) is used in the finite element program PCEPFE. However, it is desirable for mining applications that the convention where compressive stresses are positive be adopted. To achieve this all stresses including shear stresses are reversed in sign during the post-processing stage, i.e., the compressive stresses become positive and tensive stresses negative.

The program will accept any consistent system of units. However, SI units are suggested.

Modulus of deformation	MPa
Unit weight of rock	MN/M**3
Length	m
Poisson's ratio	dimensionless
Stresses	MPa
Displacements	m
Cohesive strength	MPa
Angle of internal friction	degrees

Input Data Instructions:

The input data required by the program PCEPFE can be divided into six groups, namely:

(a) Problem identification and control parameters.

(b) Material properties.

(c) Nodal point coordinates.

(d) element parameter data.

(e) Optional input for initial stress evaluation.

(f) Information for incremental mining steps.

The detail of input data is described in the following tables 1-6:

Table 1

Group (a) - Problem Identification and Control Data:		
Variable(s)	Variable Definition or Description	Format
Line No. 1 - Problem title information:		
HEAD	72 character (18 words) problem title.	18A4
Line No. 2 - Problem control information:		
NUMNP	Total number of nodal points.	4I5,2F10.2,4I5
NUMEL	Total number of elements.	
NUMMAT	Total number of different materials.	
NUMPC	Total number of pressure records.	
ACELR	Acceleration in x-direction (horizontal).	
ACELZ	Acceleration in y-direction (vertical).	
NP	Maximum number of iterations for each incremental step.	
NSTEP	Total number of mining steps or subproblems.	
MCASE	Initial stress indicator: If MCASE = 0, the initial stress will be evaluated before the first mining step or subproblem; If MCASE = 1, the initial stress is read from data file.	
NMR	Maximum number of elements/nodal points to be removed or added in any mining step or subproblem.	
Remarks:		

Table 2

Group (b) - Material Properties: A total of NUMMAT sets of material property will be provided. Each set will consist of two (2) lines of data:		
Variable(s)	Variable Definition or Description	Format
Line No. 1 - Material Identification:		
MTYPE RO	Material identification number. Mass density of material. Note that elements with RO = 0.0 will be ignored. The program uses this to indicate an excavated element. Use unit weight of material if ACELZ = -1.0	I5,F10.0
Either line No. 2a - Physical properties for two-dimensional elements:		
E(1,MTYPE) E(2,MTYPE) E(3,MTYPE) E(4,MTYPE)	Modulus of Deformation. Poisson's ratio. Cohesion. Angle of internal friction in degrees.	4F10.0
Or line No. 2b - Physical properties for one-dimensional elements:		
E(1,MTYPE) E(2,MTYPE) E(3,MTYPE) E(4,MTYPE) E(5,MTYPE)	Modulus of Deformation. Poisson's ratio. An indicator: If E(3,MTYPE) = 1, the element is pre-stressed. If E(3,MTYPE) = 0, the element is not pre-stressed. Allowable compressive strength of the material if it is pre-stressed. Cross-sectional area of one-dimensional element.	5F10.0
Remarks: only Line 1 and Line 2(a) or Line 1 and Line 2(b) are considered as one set.		

Table 3

Group (c) - Nodal Point Data:

Each non-generated nodal point has one record associated with it as follows:

Variable(s)	Variable Definition or Description	Format
N	Nodal point number.	I5,F5.0,4F10.0
CODE	Type of nodal point constraint (see Note 1).	
R(N)	x-coordinate (horizontal).	
Z(N)	y-coordinate (vertical).	
UR(N)	x-load or x-displacement.	
UZ(N)	y-load or y-displacement.	

Note 1: CODE = 0, UR is the specified x-load (horizontal).
UZ is the specified y-load (vertical).
CODE = 1, UR is the specified x-displacement.
UZ is the specified y-load.
CODE = 2, UR is the specified x-load.
UZ is the specified y-displacement.
CODE = 3, UR is the specified x-displacement.
UZ is the specified y-displacement.

Note 2: Nodal point data must be in numerical sequence. Nodal points for which no data are input will be generated by interpolation between specified nodal points.

Table 4

Group (d) - Element Parameter Data:

Variable(s)	Variable Definition or Description	Format
M	Element number.	6I5,5F10.0
IX(M,1)	Nodal point I.	
IX(M,2)	Nodal point J.	
IX(M,3)	Nodal point K.	
IX(M,4)	Nodal point L.	
IX(M,5)	Material type number.	
SIGI(M,1)	Initial stress component in x-direction (horizontal)	
SIGI(M,2)	Initial stress component in y-direction (vertical).	
SIGI(M,3)	Initial shearing stress in x-y plane.	
SIGI(M,4)	Initial stress component in z-direction (transverse)	
TH(M)	Thickness of element, the default is 1.	

Remarks: Elements omitted from the sequence will be generated. The material type for each generated element will be the same as that of the preceeding element.

Table 5

Group (e) - Optional Input for Initial Stress Evaluation:

Variable(s)	Variable Definition or Description	Format
Line No. 1 - Title description (a descriptive title of the step)		
HEAD	72 character (18 words) heading or title	18A4
Line No. 2 - Control Information:		
NUMNP	Total number of nodal points in this mining step.	I5
NUMEL	Total number of elements in this mining step.	I5
NUMPC	Total number of pressure records.	I5

Remarks: This information is needed only when MCASE = 0, as specified in group (a). Otherwise, proceed to group (f).

Table 6a

Group (f) - Information for Incremental Mining Step: One set of data will be required for each incremental step of excavation and or construction (backfill). It consists of the following:		
Variable(s)	Variable Definition or Description	Format
Line No. 1 - Title description (a descriptive title of the current step)		
HEAD	72 character (18 words) heading or title	18A4
Line No. 2 - Incremental step Control Information:		
NPMAX	Maximum number of nodal points in this mining step.	
NELMAX	Maximum number of elements in this mining step.	
NUMER	Number of elements to be removed or added.	
NUMPC	Number of pressure records.	
MTYPE	Material number of new element. If this is an excavation step, specify a material number associated with a zero density (RO = 0.). If this is a construction step, specify the material number associated with the properties of construction material (backfill).	
NCODE	An indicator. NCODE = 0, for excavation. NCODE = 1, for construction (backfilling).	
NPMIS	Number of nodal points to be removed or added.	
NUMER1	Number of existing elements for which their material type will be altered in this step.	
MTYPE1	New material number for the altered elements.	
KMORE	An indicator. KMORE = 1, another material is being added or removed. KMORE = 0, no other material is being added or removed.	
THICK	Thickness of element, the default is 1.0	
Remarks: Continued over.		

Table 6b

Group (f) - Information for Incremental Mining Step: (Continued)		
Variable(s)	Variable Definition or Description	Format
Line No. 3 - Data defining elements to be removed or added		
NUMR(N)	N = 1, NUMER, i.e., a total of 'NUMER' elements to be input.	16I5
Line No. 4 - Data defining nodal points to be removed or added.		
NPP(M)	M = 1, NPMIS, i.e., a total of 'NPMIS' nodes to be input.	16I5
Line No. 5 - Boundary Pressure Data. A total of NUMPC records (lines) need to be input.		
IBC(L)	Nodal point I.	I5
JBC(L)	Nodal point J.	I5
PR(L,1)	Normal pressure at node I.	F10.0
PR(L,2)	Normal pressure at node J.	F10.0

Remarks: Note that the boundary of the element must be on the left hand side as one progresses from I to J. Surface tensile distributed loads are input as negative pressures.

Output:

Printer Output:

The input data such as problem identification, control parameters, material properties, nodal point coordinates and element connectivity, etc., will be printed out. Displacements and stresses can be output either for all iterations of each incremental mining step or for the last iteration of each incremental mining step. The yielded or failed element(s), if any, will be printed for each iteration even if the stresses and displacements are not printed out.

Save File - `'epfeout.dat'`:

In order to facilitate graphical representation of stresses and displacements for each incremental mining step, certain data is written onto a disk file and saved for latter processing. There are a total of (5 + 2× NSTEP) files to be written on this save file. The logical name is call `'epfeout.dat'` and its contents are described below in Tables 7a, 7b and 7c.

Table 7a

Contents of Save File - `epfeout.dat`	
(a) Mining Step Information:	
	Consists of one record header - Format (1H ,4I10,18A4,2X) Contents of header are NC, NC1, NREC, NDUM, TITLE. Here NC = 0, NC1 = 0, NREC = 1, NDUM = 0, and TITLE is an eighteen word vector containing the problem title. The rest of the file contains an additional record, formatted as (1H ,2I10). Its contents are: NSTEP - total number of incremental mining steps, and NRES - an initial stress indicator and set NRES = 0. (not used in post-processing).
Co-ordinates of Nodal Points:	
	Consists of one recorder header - Format (1H , 4I10, 18A4, 2X). Contents of header are NC, NC1, NUMNP, NDUM, TITLE. Here NC = 0, NC1 = 1, NUMNP is the total number of nodal points. NDUM = 0, and TITLE is an eighteen word vector containing the problem title. The rest of the file contains the X and Y co-ordinates of all (NUMNP) nodal points: (X(I), Y(I), I = 1, NUMNP) in Format (1X, 2E15.7).
(c) Material Properties (a dummy record):	
	Consists of one record header - Format (1H ,4I10, 18A4, 2X) Contents of header are NC, NC1, NREC, NDUM, TITLE. Here NC = 0, NC1 = 2, NREC = 0, NDUM = 0, and TITLE is an eighteen word vector containing the problem title.
(d) Initial Stresses (a dummy record):	
	Consists of one recorder header - Format (1H ,4I10, 18A4, 2X). Contents of header are NC, NC1, NREC, NDUM, TITLE. Here NC = 0, NC1 = 3, NREC = 0, NDUM = 0, and TITLE is an eighteen word vector containing the problem title.
Remarks: continued over.	

Contents of Save File - `epfeout.dat` (continued)	
(e) Element Data:	
	Consists of one record header - Format (1H ,4I10,18A4,2X) Contents of header are NC, NC1, NUMEL, NDUM, TITLE. Here NC = 0, NC1 = 4, NUMEL is the total number of elements, NDUM = 0, and TITLE is an eighteen word vector containing the problem title. The rest of the file contains 'NUMEL' records with Format (1H ,6I10, 10X); and contents of each record are: N, (IX(I), I = 1, 4), MAT. Where N is the element number, IX(1), ... IX(4) are the nodal points defining the element, MAT is the material number.
	The above data is followed by 2×NSTEP files, i.e., for each incremental mining step or subproblem (1 to NSTEP) there is associated one file containing the nodal displacements and a second containing element stresses.
(f) Nodal Displacements:	
	The displacement output associated with the incremental step NC is stored as follows: One record header - Format (1H ,4I10, 18A4, 2X). Contents are NC, NC1, NONNP, LL, TITLE. Where NC is the incremental mining step or subproblem number, NC1 = 1, NONNP is the number of non-deleted nodal points, LL is the load case number (=1), TITLE is an eighteen word vector containing the problem title. The header id followed by 'NONNP' records containing the displacements: Contents are N, LL, DX(N), DY(N) in Format (1H ,I10, I5, 2E15.7). Where N is the nodal point number, LL is the load case number (=1), DX(N) and DY(N) are the horizontal and vertical displacements respectively.
Remarks: continued over.	

Table 7c

Contents of Save File - `epfeout.dat` (continued)	
(f) Element Stresses:	
	The stress output associated with the incremental mining step or subproblem NC is stored as follows: One record header - Format (1H ,4I10,18A4,2X) Contents of header are NC, NC1, NONEL, LL, TITLE. Here NC is the incremental mining step or subproblem number, NC1 = 2, NONEL is the number of non-deleted elements and TITLE is an eighteen word vector containing the problem title. The header is followed by 'NONEL' records giving the stresses in each element. Contents are N, (SIG(N,I), I=1, 4), SIG1, SIG2, SANG, XC, YC. N is the element number, (SIG(N,I), I=1, 4) are $\sigma_{xx}, \sigma_{yy}, \sigma_{zz}$ and σ_{xy}, respectively. SIG1 and SIG2 are the principal stresses and SANG is the angle in degrees that the major principal stress makes with the X axis. XC and YC are the x and y co-ordinates of the centroid of each element. The format is: (1H ,2I5, A3, 9E13.7)
Remarks:	

Excavation and/or Construction Sequence:

Excavation and/or construction (such as backfill in mines) can be conveniently simulated by associating with each element a number call cut number corresponding to the subproblem on the incremental mining step number, in which the element is removed or added. The removed element can be added later in a higher subproblem. However, it is a necessary restriction that any one element cannot be altered (removed/added) in the same incremental mining step or subproblem.

The cut numbers are generated by the mesh-generating system MSHGEN. However, it is important that when MSHGEN is used a four-digit integer must be assigned for the the cut number if a zone of the structure is involved with both excavation and construction sequences. The left-most two two integers (0-99) are reserved for the construction sequence and the right-most two integers (0-99) are used for excavation. For example, elements with a cut number 0301 indicates that these elements are to be removed in the first incremental mining step or the first subproblem (the integer 01); these elements removed will be added back in the third incremental mining step or third subproblem to simulate construction or backfill (the integer 03). This rule must be followed in the preparation of MSHGEN input.

MSHGEN Input Data Instructions:

The concepts involved and procedures used in MSHGEN have been discussed in detail in reference [4]. This input data required for MSHGEN is simple and extracted below for easy reference. For users who are not familar with the mesh generating system MSHGEN it is recommended to refer to the above-mentioned reference.

The input for MSHGEN is subdivided into the following groups:

(a) Title and problem control information,

(b) Block of data defining specified nodal points, and

(c) Block of data defining zones.

The detail of input is described in the following tables 8-9. All the formats for data entry is list-directed, i.e., free format. Note that for free format, the character string must be quoted, i.e., it begins with a quote (') and ends with a quote (').

Table 8

Variable(s)	Variable Definition or Description	Format
Group (a) - Problem Identification and Control Data:		
Line No. 1 - Problem title information:		
HEAD	72 character (18 words) problem title.	18A4
Line No. 2 - Problem control information:		
NSPNP	Total number of specified nodal points.	I5
NVZONE	Total number of non-void zones.	I5
NSPAN1	Total number of Spans in ϵ direction.	I5
NSPAN2	Total number of spans in η direction.	I5
NPROB	The total number of subproblems or incremental mining steps making up the excavation and or construction sequence. Default value is 1.	I5
NSIDNT	Identification indicator, a value of zero or blank means no identification; NSIDNT = 1, indicates identification. Default = 0 for most of the meshes generated.	I5
Line No. 3 - data defining zone subdivisions:		
NSBDV1(I)	Array defining number of subdivisions in each zone in ξ direction.	I5
BSBDV2(I)	Array defining number of subdivisions in each zone in η direction.	I5

Remarks:

Table 9a

Group (b) - Block of Data defining Specified Nodal Point.
A total of 'NSPNP' records will follow.

Variable(s)	Variable Definition or Description	Format
	Record No. 1 - Coordinates of Specified Nodes:	
N	Specified nodal point number.	
NCODSP(N)	Constraint code of specified nodal point N. The table below defines the possibilities:	

NCODSP(N)	Constraint at Node N
0	No constraint on displacements.
1	x-displacement = 0.
2	y-displacement = 0.
3	x-displacement = 0, y-displacement = 0.

Variable(s)	Variable Definition or Description	Format
XSP(N)	x coordinate of node N.	
YSP(N)	y coordinate of node N.	

Group (c) - Block of Data Defining Non-void Zones.
With each non-void zone the following records must follow:

Variable(s)	Variable Definition or Description	Format
N	Non-void zone number.	
IZ(,N)	Nodal number defining non-void zone N. (see Note 1)	
MATZ(N)	The material number associated with the non-void zone N. the default value is 1. (see Note 2)	
NCUTZ(N)	The cut number associated with non-void zone N. If blank or zero is assigned the value 'NPROB + 1', i.e., the element in this zone will never be removed or excavated. (see Note 3)	4 dig. No
NUMPC No. of pressure card	The number of sides (0-4) of non-void zone with pressure applied, i.e., a total of 'NUMPC' records defining pressure loading will follow immediately.	

Remarks: Continued over.
The definition of axes is shown in Figs. 3 and 4.

Table 9b

Group (c) - Block of Data defining Non-void Zones (continued).

Variable(s)	Variable Definition or Description	Format
Record No. 2 - Required only if NUMPC is greater than zero the the above data is followed by 'NUMPC' records defining pressure sides on zone N.		
NSIDE	The side number of the zone N. See Fig. 2 for the number associated with sides.	I5
P1	The pressure at node 1 of zone side 'NSIDE'.	F10.2
P2	The pressure at node 2 of zone side 'NSIDE'.	F10.2

Remarks:

Note 1: Zone Specification:
The corner nodes must always be specified. Intermediate nodes of zero or blank are assumed by `MSHGEN` to be mid-point nodes. Once the global (ξ, η) reference system has been selected then the order in which nodes are specified is complete rigid. Order of de specification is counter-clockwise starting with node at $(\text{MIN}(\xi), \text{MIN}(\eta))$.

Note 2: Material Specification:
Different material should be given numbers 1 to NUMMAT, where NUMMAT is the number of different materials. The reason for this is that `MSHGEN` assumes that maximum material number equals the number of different materials.

Note 3. Cut Number:
Excavation and or construction sequences can be simulated by assigning a cut number to a zone. All elements in a zone will be removed when the subproblem number or incremental mining step number equals the cut number. A cut number of one (1) means that elements in this zone will never be used. Elements with cut number greater than 'NPROB' (the maximum number of subproblem) will never be deleted. The rules for assigning a cut number to a zone in which excavation and construction will be involved has been discussed in both previous sections.

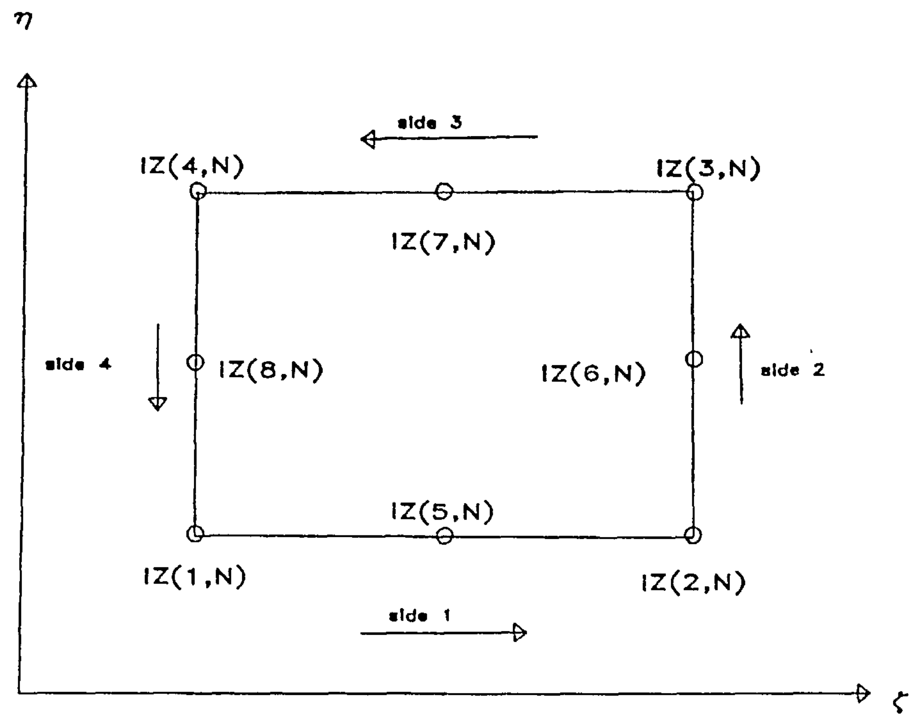


Fig. 2 Specification of zone N

EPFEC - AN INTERFACE PROGRAM

The mesh-generating system MSHGEN produces only part of the input data required by the programPCEPFE such as the nodal points coordinates and elements data. Other information such as material properties, element to be removed or added for simulation of construction/backfill as described previously are absent. EPFEC is an interfacing program which merges this additional information with the output (genout.dat) of MSHGEN to produce an input file (epfein.dat) acceptable to PCEPFE. It should be noted that the following restrictions apply:

(a) No one-dimensional elements, and

(b) No change of material properties for existing elements.

EPFEC Input Data Instructions:

The input data for EPFEC are divided into four groups, namely:

(a) Problem control information,

(b) Material properties of rock mass,

(c) Initial stress coefficients, and

(c) Incremental mining step informations.

The detail of input requirement is described in the following tables 10-13. All the formats for data entry are list-directed, i.e., free format unless it is mentioned otherwise.

The input of initial stresses is further explained in the next section - Example Problems.

Printer: Tension + compression
Screen: Tension x compression

Table 10

Group (a) - Problem Identification and Control Data:		
Variable(s)	**Variable Definition or Description**	**Format**
Line No. 1 - Problem control information:		
MAXELR	Maximum number of elements to be removed or added in any excavation and or construction step.	
NP	Maximum number of iterations for each incremental mining step.	
NRES	An indicator for input initial stresses: = 0, input initial stress coefficients; initial stresses will be calculated by routine CALRES and will be input as data. = 1, no initial stresses will be input.	
NMAT	Total number of different materials. NMAT = actual number of materials + 1. This extra material type which has density equal to zero (0) is included to describe excavated elements.	
ACELR	Acceleration in x-direction (horizontal).	
ACELZ	Acceleration in y-direction (vertical).	
SCALE	A scaling factor. Coordinates generated from MSHGEN will be multiplied by this factor. The default is 1.0	
Line No. 2 - Output Print Control:		
INDPRT	An output print control indicator. = 0, will print out stresses and displacements for last iteration only for each incremental mining step. = 1, will print out stresses and displacements for every iteration of each incremental mining step.	
Remarks:	The use of initial stresses is further explained in Example Problems Section.	

Table 11

Group (b) - Material Properties of Rock Formations or Backfill:		
Variable(s)	**Variable Definition or Description**	**Format**
With each material type there are associated two records:		
Record No. 1		
MTYPE RO	Material identification number. Density of material. To simulate gravitational effects set RO equal to unit weight of the material and set ACELZ = - 1.	
Record No. 2		
E(MTYPE,1) E(MTYPE,1) E(MTYPE,1) E(MTYPE,1) E(MTYPE,1)	Modulus of deformation. Poisson's ratio. Cohesive strength. Angle of internal friction. Area of one-dimensional element. (set to zero)	
Repeat Records No. 1 and 2 'NMAT' times.		
Group (c) - Concentrated Nodal Forces:		
Record No. 1		
NUMCON	Total number of nodes where nodal forces are acting	
Record N0. 2: If NUMCON = 0, skip record No. 2 and proceed to Group (d). Otherwise, repeat Record No. 2 'NUMCON' times.		
NPC XLOAD YLOAD	Nodal point number. x-load or y-displacement. y-load or x- displacement.	
Remarks:		

Table 12

Group (d) - Initial Stress Coefficients:

Variable(s)	Variable Definition or Description	Format
Record No. 1		
AXX BXX	Coefficient Coefficient	
Record No. 2		
AYY BYY	Coefficient Coefficient	
Reocrd No. 3		
AZZ BZZ	Coefficient Coefficient	
Record No. 4		
AXY BXY	Coefficient Coefficient	

The initial stresses are assumed to be varying linearly with depth, Y. The above coefficients are better illustrated by the following equations:

$$\sigma_{xx} = AXX + BXX \times Y$$
$$\sigma_{yy} = AYY + BYY \times Y$$
$$\sigma_{zz} = AZZ + BZZ \times Y$$
$$\sigma_{xy} = AXY + BXY \times Y$$

Where σ_{xx}, σ_{yy} and σ_{zz} , are the initial stresses in the horizontal, vertical and transverse directions, respectively. σ_{xy} is the shearing stress in the xy plane. Y is the depth.
The definition of axes is shown in Figs. 3 and 4.

Table 13

Group (e) - Incremental Mining Step Information:		
Variable(s)	Variable Definition or Description	Format
Record No. 1 - Title or problem identification.		
HEAD	A title or heading describes the current incremental step.	18A4
Record No. 2 - Material type.		
MTYPE	Material identification number. If this is a construction step, i.e., backfilling, then input the material number for the backfill material. If it is an excavation step, then input the material number for which its density should be equal to zero, i.e., RO(MTYPE) = 0.	
Remarks:		

EXAMPLE PROBLEMS

Cut-and-Fill Mining

A hypothetical cut-and-fill mining system was devised to illustrate the use of the computer program. The orebody, approximately 10m thick, dips at 70°. The stope at the lower level was mined first before the upper stope was mined. The stope are 50m high, separated by a sill pillar of 30m. Figure 3 shows the schematic diagram of the mining geometry, and its associated zone diagram is shown in Fig. 4. Figure 5 shows the corresponding finite element mesh. The input data required by `MSHGEN` to generate part of the input required by `PCEPFE` is shown in Fig. 6. Additional input data required by `EPFEC`, as shown in Fig. 7, is then merged with `MSHGEN` output data (`'genout.dat'`) to produce an input file which is acceptable to `PCEPFE`.

The Initial Stresses are assumed to be varying linearly with depth and are in the form of:

$$\sigma_{xx} = a_{xx} + b_{xx} \times Y$$
$$\sigma_{yy} = a_{yy} + b_{yy} \times Y$$
$$\sigma_{zz} = a_{zz} + b_{zz} \times Y$$
$$\sigma_{xy} = a_{xy} + b_{xy} \times Y$$

where σ_{xx}, σ_{yy} are the horizontal and vertical stresses respectively. σ_{zz} is the stress perpendicular to the plane and σ_{xy} is the shearing stress in the XY plane. a_{xx}, b_{xx}, a_{yy} ...... are the coefficients relating the stress components with depth. Y is the depth at which the stresses are evaluated. The definition of axes is shown in Figs. 3 and 4.

Under Gravitational Loading and under plane strain conditions, the loading for evaluating the resultant stresses from the finite element model can either be achieved by applying appropriate tractions along the boundary of a model or by placing appropriate constraints along the sides of the model. When the loading conditions are known, the coefficients relating the stress components, as shown above, can be easily evaluated.

However, if displacements are of no concern, then it is not required to enter the initial stresses. The input of initial stresses will not affect, in any way, the resultant stresses resulting from any excavation, but it will have an effect on the displacements. In other words, a model, consisting of no excavation, is loaded with boundary tractions together with the input of initial stresses which are compatible with the applied tractions, then, the displacements everywhere within the model should be zero. This establishes the reference point for evaluating displacements in the subsequent sub-problems.

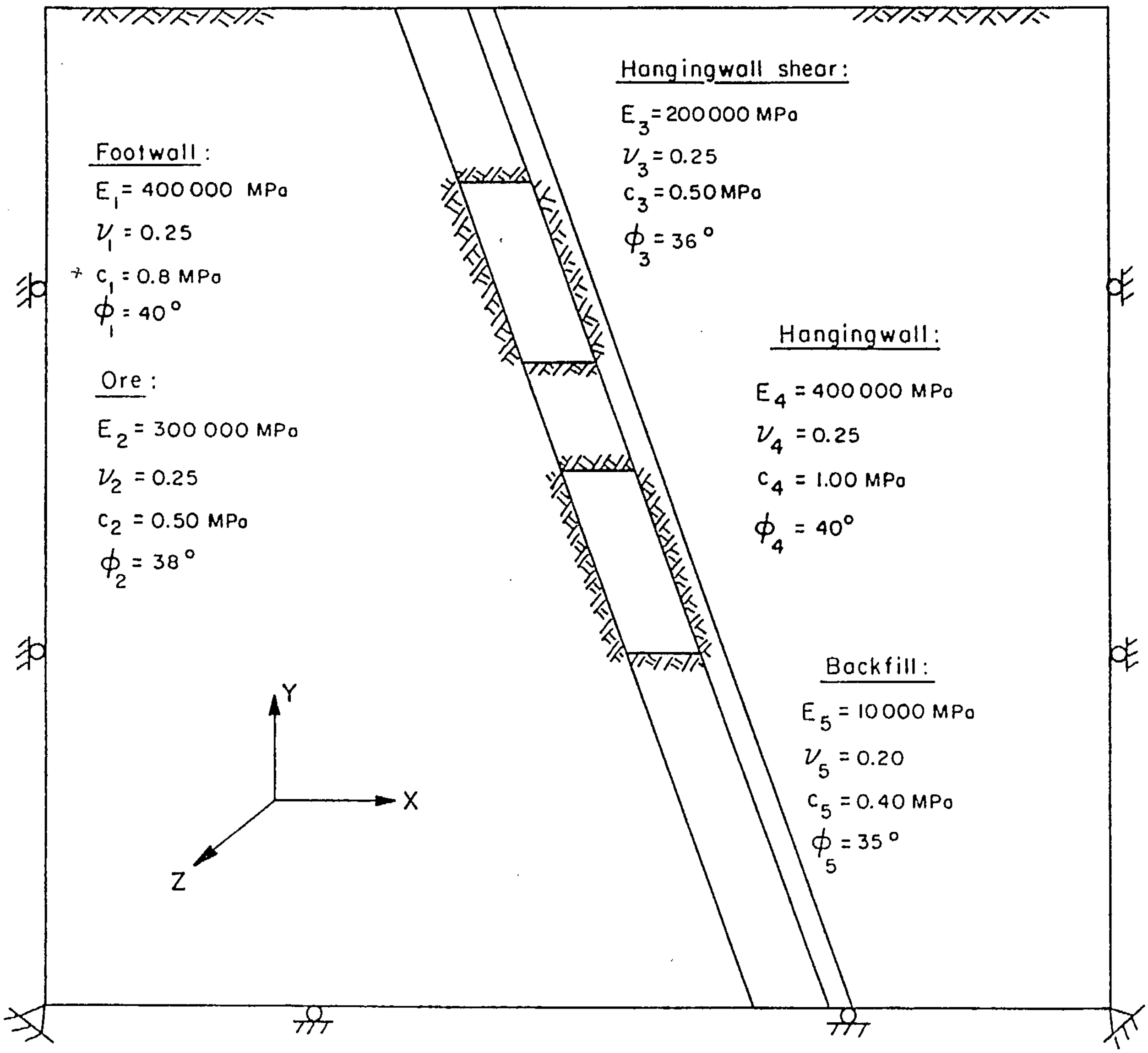


Fig. 3 A hypothetical cut-and-fill mining layouts - a sectional view

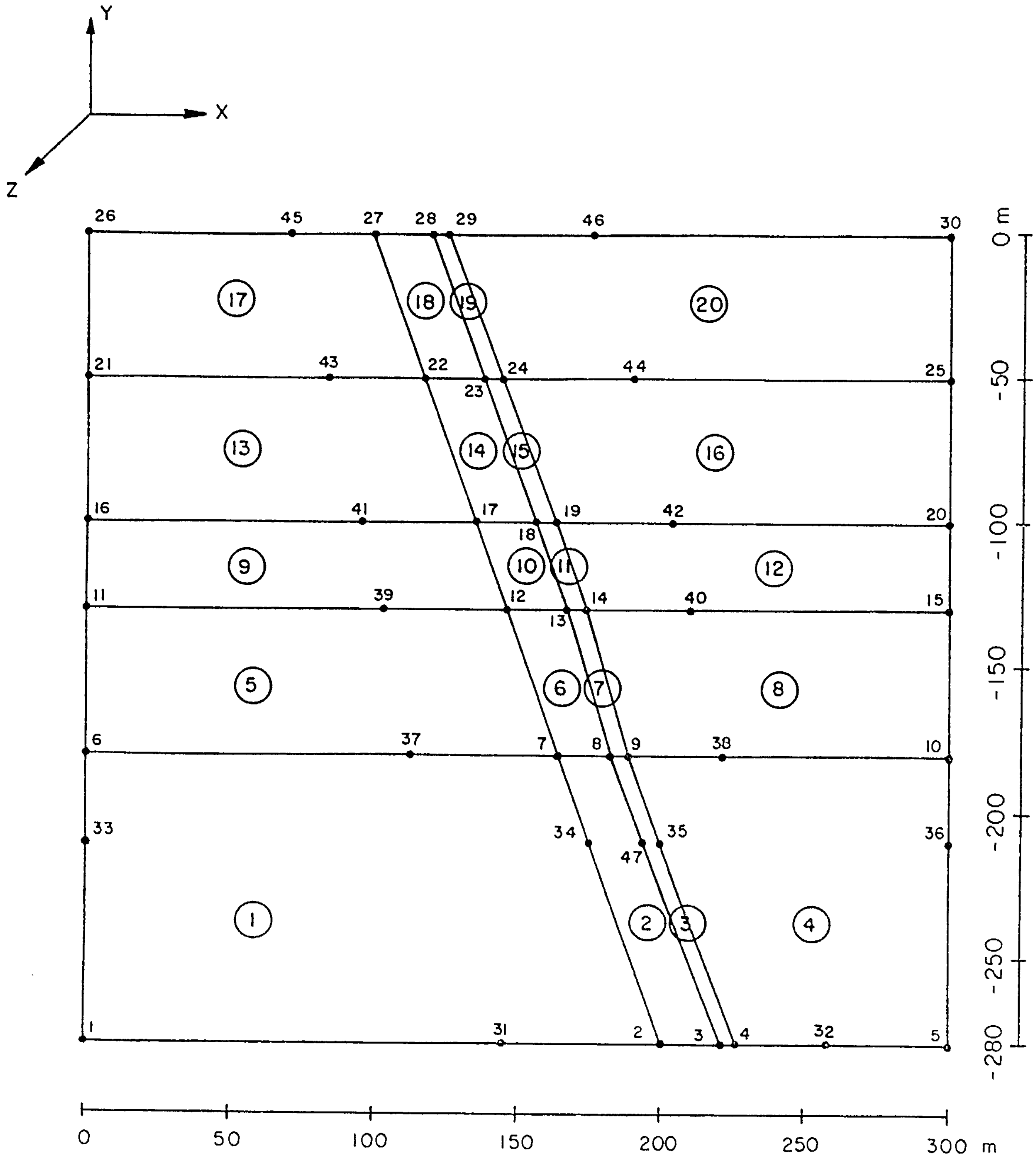


Fig. 4 A zone diagram (numbers are specified nodes and circled numbers are zones)

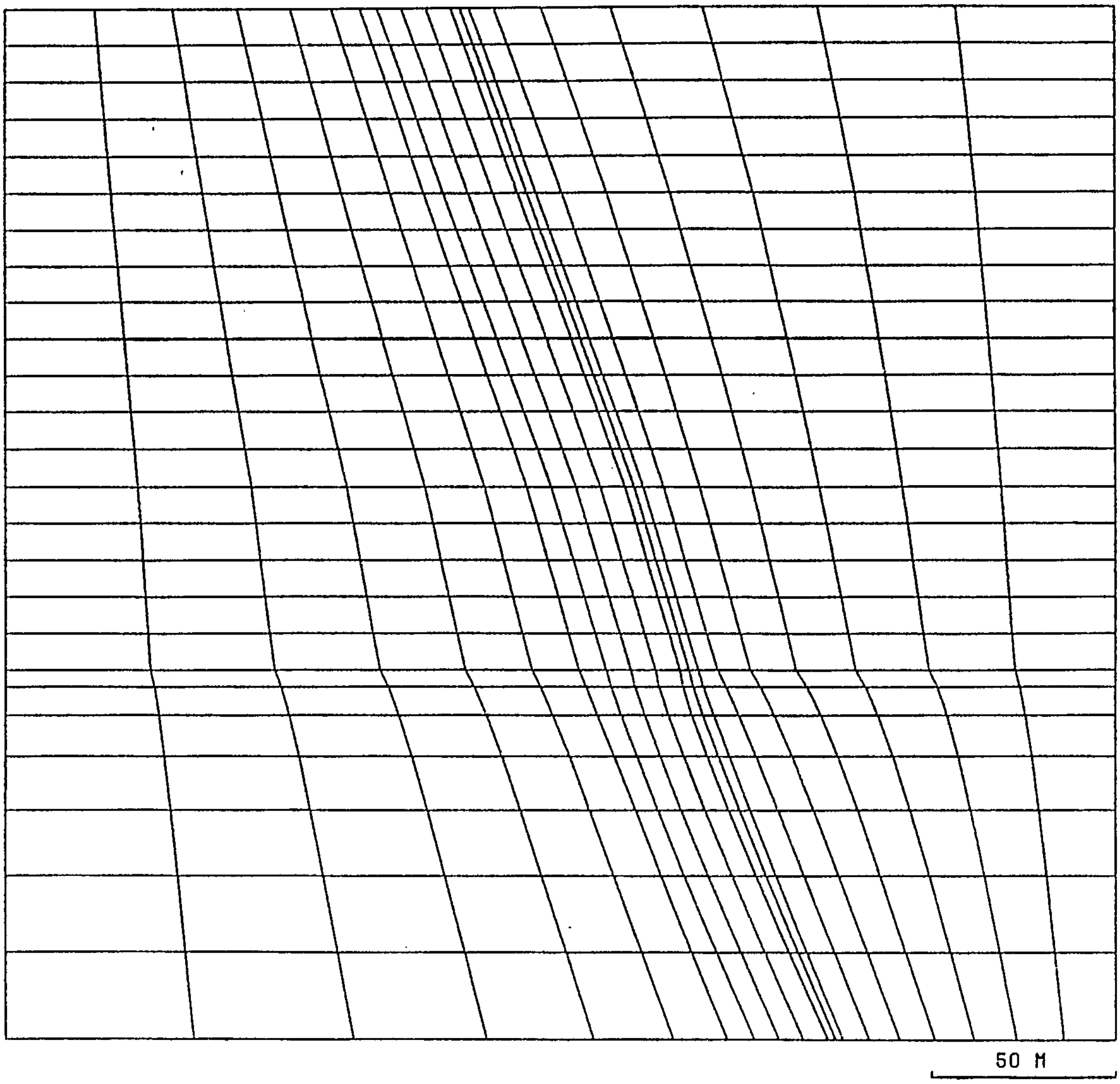


Fig. 5 Finite element mesh

```
'Example for elastic-plastic finite element analysis - gravity only'
  47   20    4    5    3    0
   7    3    2    7
   7    5    3    5    5
   1    3      0.0   -280.0
   2    2    201.9   -280.0
   3    2    221.9   -280.0
   4    2    226.0   -280.0
   5    3    300.0   -280.0
   6    1      0.0   -180.0
   7    0    162.0   -180.0
   8    0    182.0   -180.0
   9    0    187.0   -180.0
  10    1    300.0   -180.0
  11    1      0.0   -130.0
  12    0    147.0   -130.0
  13    0    167.0   -130.0
  14    0    172.0   -130.0
  15    1    300.0   -130.0
  16    1      0.0   -100.0
  17    0    136.0   -100.0
  18    0    156.0   -100.0
  19    0    161.0   -100.0
  20    1    300.0   -100.0
  21    1      0.0    -50.0
  22    0    118.0    -50.0
  23    0    138.0    -50.0
  24    0    143.0    -50.0
  25    1    300.0    -50.0
  26    1      0.0      0.0
  27    0    100.0      0.0
  28    0    120.0      0.0
  29    0    125.0      0.0
  30    1    300.0      0.0
  31    0    145.0   -280.0
  32    0    256.0   -280.0
  33    0      0.0   -210.0
  34    0    172.9   -210.0
  35    0    197.9   -210.0
  36    0    300.0   -210.0
  37    0    113.0   -180.0
  38    0    221.0   -180.0
  39    0    103.0   -130.0
  40    0    210.0   -130.0
  41    0     95.0   -100.0
  42    0    202.7   -100.0
  43    0     82.6    -50.0
  44    0    190.0    -50.0
  45    0     70.0      0.0
  46    0    175.0      0.0
  47    0    192.0   -210.0
```

Fig. 6 MSHGEN input data (continued over)

1	1	2	7	6	31	34	37	33	1	0	0
2	2	3	8	7	0	47	0	34	2	0	0
3	3	4	9	8	0	35	0	47	3	0	0
4	4	5	10	9	32	36	38	35	4	0	0
5	6	7	12	11	37	0	39	0	1	0	0
6	7	8	13	12	0	0	0	0	2	201	0
7	8	9	14	13	0	0	0	0	3	0	0
8	9	10	15	14	38	0	40	0	4	0	0
9	11	12	17	16	39	0	41	0	1	0	0
10	12	13	18	17	0	0	0	0	2	0	0
11	13	14	19	18	0	0	0	0	3	0	0
12	14	15	20	19	40	0	42	0	4	0	0
13	16	17	22	21	41	0	43	0	1	0	0
14	17	18	23	22	0	0	0	0	2	3	0
15	18	19	24	23	0	0	0	0	3	0	0
16	19	20	25	24	42	0	44	0	4	0	0
17	21	22	27	26	43	0	45	0	1	0	0
18	22	23	28	27	0	0	0	0	2	0	0
19	23	24	29	28	0	0	0	0	3	0	0
20	24	25	30	29	44	0	46	0	4	0	0

Fig. 6 MSHGEN input data (continued)

```
   50   15    0    6         0.       -1.0         1.
    0
    1     0.029
    4.E+05         .25       0.8       40.       0.0
    2     0.029
    3.E+05         .25       0.8       38.       0.0
    3     0.029
    2.E+05         .25       0.5       36.       0.0
    4     0.029
    4.E+05         .25       1.0       40.       0.0
    5     0.025
    1.E+04       .20         0.4       35.       0.0
    6       0.0
    0.0            .0         .0        0.       0.0
    0
0.,0.0097
0.,0.0290
0.,0.0097
0.,0.
***** Excavating a stope at lower level *****
    6
***** Backfilling the lower stope *****
    5
***** Excavating a stope at upper level *****
    6
```

Fig. 7 EPFEC input data

Under gravitational loading the initial shearing stress $\sigma_{xy} = 0$, therefore, the coefficients $a_{xy} = b_{xy} = 0$. The vertical stress σ_{yy} is due to gravity only and the horizonal stresses, σ_{xx} and σ_{zz}, are due to Poisson's effect. If we assume that γ, the average unit weight of rock mass, is 0.029 MPa/m, and the Poisson's ratio is 0.25, then, the stresses are given:

$$\begin{aligned}\sigma_{yy} &= \gamma \times h \\ \sigma_{xx} &= \frac{\nu}{1-\nu}\sigma_{yy} \\ \sigma_{zz} &= \nu(\sigma_{xx} + \sigma_{yy})\end{aligned}$$

Where h is the depth below the ground surface. At the top of the model, i.e., at the ground surface, $h = Y = 0$.

We have: $\sigma_{yy} = a_{yy} + b_{yy} \times (0) = 0$, therefore, $a_{yy} = 0.0$.

At the bottom of the model, where $h = Y = -280.0$m, we have:

$$\begin{aligned}\sigma_{yy} &= \gamma \times Y \\ &= 0.029 \times (-280.0)\end{aligned}$$

Also, we have:
$a_{yy} + b_{yy} \times (-280.0) = 0.0 + -0.029 \times 280.0$

Therefore, we have: $b_{yy} = 0.029$

From the following two equations:

$$\begin{aligned}a_{xx} + b_{xx} \times Y &= \frac{0.25}{1-0.25}(0.029 \times Y) \\ a_{zz} + b_{zz} \times Y &= 0.25 \times (0.097 + 0.029) \times Y\end{aligned}$$

we obtain:

$$\begin{aligned}a_{xx} &= 0.0 \\ b_{xx} &= 0.0097 \\ a_{zz} &= 0.0 \\ b_{xx} &= 0.0097\end{aligned}$$

Summarizing, we have:

$$a_{xx} = 0.0$$
$$b_{xx} = 0.0097$$
$$a_{yy} = 0.0$$
$$b_{yy} = 0.029$$
$$a_{zz} = 0.0$$
$$b_{zz} = 0.0097$$
$$a_{xy} = 0.0$$
$$b_{xy} = 0.0$$

These coefficients are shown in Fig. 7.

In the Canadian Shield, it is known that horizontal stresses are greater than vertical stresses. In this case the loading simulating the in-situ stress conditions must be achieved by applying appropriate tractions along the boundary of the mine model. Let's suppose that the vertical stress σ_{yy} is due to gravity only. The horizonal stresses, σ_{xx} and σ_{zz}, are consisting of two components, one of which is the tectonic stress uniformly distributed across the depth, say 3 MPa in x-direction and 2 MPa in z-direction, and the other part is due to the Poisson's effect, i.e., $\frac{\nu}{1-\nu}\sigma_{yy}$. Also we assume that the vertical stress is one of the principal stresses. Then the initial shearing stress $\sigma_{xy} = 0$, and therefore, the coefficients $a_{xy} = b_{xy} = 0$.

Now we assume that γ, the average unit weight of rock mass, is 0.029 MPa/m, and the Poisson's ratio is 0.25, then, the stresses are given:

$$\sigma_{yy} = \gamma \times h$$
$$\sigma_{xx} = -3.0 + \frac{\nu}{1-\nu}\sigma_{yy}$$
$$\sigma_{zz} = -2.0 + \frac{\nu}{1-\nu}\sigma_{yy}$$

Similarily, at the top of the model, i.e., at the ground surface, $h = Y = 0$.

We have: $\sigma_{yy} = a_{yy} + b_{yy} \times Y = 0$, therefore, $a_{yy} = 0.0$.*

---

* If the top of the model is located at some distance below the ground surface, then $a_{yy} = h' \times \gamma$, where h' is the distance below the ground surface. A traction of the same magnitude should be applied along the top boundary simulating the 'overburden load'.

At the bottom of the model, where $h = Y = -280.0\text{m}$, we have:

$$\begin{aligned}\sigma_{yy} &= \gamma \times Y \\ &= 0.029 \times (-280.0)\end{aligned}$$

Therefore, we have: $b_{yy} = 0.029$

From the following two equations:

$$\begin{aligned}a_{xx} + b_{xx} \times Y &= -3.0 + \frac{0.25}{1-0.25}(0.029 \times Y) \\ a_{zz} + b_{zz} \times Y &= -2.0 + \frac{0.25}{1-0.25}(0.029 \times Y)\end{aligned}$$

we obtain:

$$\begin{aligned}a_{xx} &= -3.0 \\ b_{xx} &= \quad 0.3333 \\ a_{zz} &= -2.0 \\ b_{xx} &= \quad 0.3333\end{aligned}$$

Summarizing, we have:

$$\begin{aligned}a_{xx} &= -3.0 \\ b_{xx} &= \quad 0.3333 \\ a_{yy} &= \quad 0.0 \\ b_{yy} &= \quad 0.029 \\ a_{zz} &= -2.0 \\ b_{zz} &= \quad 03333 \\ a_{xy} &= \quad 0.0 \\ b_{xy} &= \quad 0.0\end{aligned}$$

Note that the calculation of these coefficients are dependent on the coordinate system you selected for your model.

Cantilever Beam Example:

A cantilever beam subjected to a load acting at the end is a classical test for most numerical methods. The dimension of the beam is shown in Fig. 8. The input for `MSHGEN` and `EPFEC` are shown, respectively, in Figs. 9 and 10.

Three discretizations for the cantilever beam have been used. In order to compare the results, we have taken the displacement of the tip. The displacement at the tip is calculated by:

$$\delta = \frac{P \times L^3}{3EI}$$

where P is the applied load, L is the beam length, E is Young's modulus and I is the moment of inertia.

To prevent any yielding in the cantilever beam, i.e., for an elastic analysis, a large value of cohesive strength, C, was assigned to the material properties as shown in Fig. 10. The finite element results and closed-form solution are given in Fig. 11.

The output file from `EPFEC` or the input file for `PCEPFE`, `'epfein.dat'`, for the mesh No.2 is given in Fig. 12.

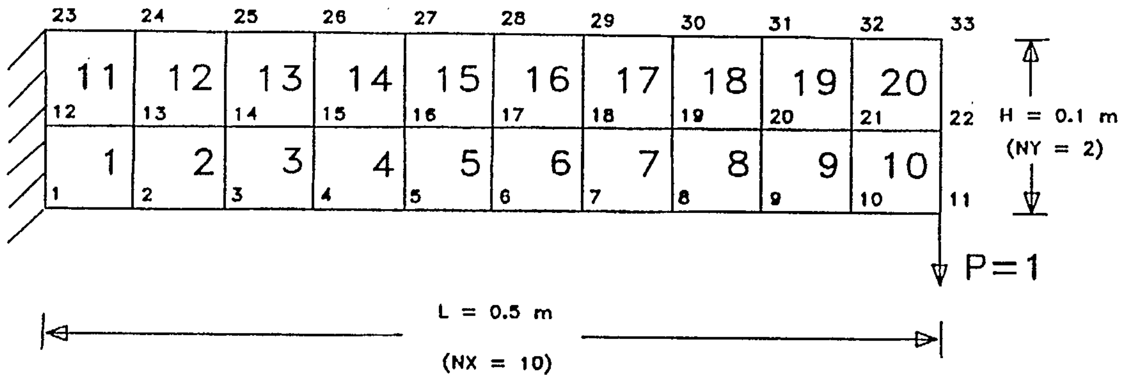


Fig. 8 - A cantilever beam subjected to a concentrated load

```
'Example - a cantilever beam with concentrated force at one end'
   4    1    1    1    1    0
  10
   2
   1    3       0.0       0.0
   2    0       0.5       0.0
   3    3       0.0       0.1
   4    0       0.5       0.1
   1    1    2    4    3    0    0    0    0    1    0    0
```

Fig. 9 MSHGEN input data for cantilever beam example

```
   1    1    1    1         0.         0.0        1.
   0
   1       1.0
   5.E+06       .25      3000.       45.        0.0
   1
  11       0.0      -1.0
***** Evaluating the stresses and displacements *****
   1
```

Fig. 10 EPFEC input data for cantilever beam example

Finite Element Mesh	NX	NY	Displacement (m)
1	5	2	0.000061
2	10	2	0.000082
3	20	4	0.000092
Closed-form Solution			0.000100

Fig. 11 - A comparison of results between finite element and closed-form solutions

```
Example - a cantilever beam with concentrated force at one end
  18   10    1    1      0.00       0.00    1    1    1    1
   0
   10.1000E+01
0.5000E+070.2500E+000.3000E+040.4500E+020.0000E+00
   1   3.  0.0000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
   2   0.  0.1000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
   3   0.  0.2000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
   4   0.  0.3000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
   5   0.  0.4000000E+00  0.0000000E+00  0.0000000E+00  0.0000000E+00
   6   0.  0.5000000E+00  0.0000000E+00  0.0000000E+00 -1.0000000E+00
   7   3.  0.0000000E+00  0.5000000E-01  0.0000000E+00  0.0000000E+00
   8   0.  0.9999999E-01  0.5000000E-01  0.0000000E+00  0.0000000E+00
   9   0.  0.2000000E+00  0.5000000E-01  0.0000000E+00  0.0000000E+00
  10   0.  0.3000000E+00  0.5000000E-01  0.0000000E+00  0.0000000E+00
  11   0.  0.4000000E+00  0.5000000E-01  0.0000000E+00  0.0000000E+00
  12   0.  0.5000000E+00  0.5000000E-01  0.0000000E+00  0.0000000E+00
  13   3.  0.0000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
  14   0.  0.1000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
  15   0.  0.2000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
  16   0.  0.3000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
  17   0.  0.4000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
  18   0.  0.5000000E+00  0.1000000E+00  0.0000000E+00  0.0000000E+00
   1    1    2    8    7    1       0.00       0.00       0.00       0.00     1.0000
   2    2    3    9    8    1       0.00       0.00       0.00       0.00     1.0000
   3    3    4   10    9    1       0.00       0.00       0.00       0.00     1.0000
   4    4    5   11   10    1       0.00       0.00       0.00       0.00     1.0000
   5    5    6   12   11    1       0.00       0.00       0.00       0.00     1.0000
   6    7    8   14   13    1       0.00       0.00       0.00       0.00     1.0000
   7    8    9   15   14    1       0.00       0.00       0.00       0.00     1.0000
   8    9   10   16   15    1       0.00       0.00       0.00       0.00     1.0000
   9   10   11   17   16    1       0.00       0.00       0.00       0.00     1.0000
  10   11   12   18   17    1       0.00       0.00       0.00       0.00     1.0000
***** Evaluating the stresses and displacements *****
  18   10    0    1    1    0    0
   0    0  0.0000000E+00  0.0000000E+00
```

Fig. 12 PCEPFE input file (epfein.dat) for the cantilever beam example

GETTING STARTED

As mentioned earlier, PCEPFE software package consists of the following programs:

(a) PCEPFE

(b) MSHGEN

(c) EPFEC

(c) MSHPLT

(e) PCPLOT

The function of each program has been discussed in previous sections.

A command procedure called EPFE is written for the MS DOS operating system and is used to access these individual modules of the software package. The main menu of the command procedure, EPFE, is shown in Fig. 13. The details concerning the use of MSHPLT and PCPLOT are given in References [6,7].

Hardware Requirements:

To run PCEPFE software package efficiently, an IBM PC/AT compatible is required. The minimum desirable configuration of the system is described below in Table 14.

```
*****************************************************************
*                         P C E P F E                           *
*                                                               *
*     TWO-DIMENSIONAL NONLINEAR ELASTIC-PLASTIC FINITE ELEMENT  *
*           PROGRAM (1988 CANMET - VERSION 1.0 (MAR,1988)       *
*                                                               *
*****************************************************************
*                                                               *
*                       M A I N   M E N U                       *
*                                                               *
*****************************************************************
*                                                               *
*               1. EXECUTE MSHGEN  - GENERATING MESH DATA       *
*               2. EXECUTE MSHPLT  - GRAPHICS                   *
*               3. EXECUTE EPFEC   - MERGING MSHGEN DATA        *
*               4. EXECUTE PCEPFE  - NUMERICAL CODE             *
*               5. EXECUTE PCPLOT  - GRAPHICS ON SCREEN         *
*               6. EXECUTE PCPLOT  - GRAPHICS ON PRINTER        *
*               7. HELP MENU                                    *
*               8. EXIT TO OPERATING SYSTEM (DOS)               *
*                                                               *
*****************************************************************
```

Fig. 13 - The main menu of EPFE command procedure

Table 14

Minimum Configuration of the System		
Component	Description	Comments
CPU	Intel 80286 with math coprocessor	
Memory	640 kilobytes (see computer memory requirements)	
Monitor	color monitor with EGA graphic board	
Mass Storage	one 5.25 inch, double density, dual sideed floppy diskette drive and a twenty (20) or thirty (30) megabyte hard disk drive	
Printer	HP PaintJet printer	
Plotter		
Mouse	Microsoft compatible mouse	
Remarks:	1. At present the pre- and post- processors require a clour printer to produce hard copy. 2. To output finite element results a 132 column printer is desirable.	

Software Requirements:

The main program PCEPFE and the companion programs are compiled and linked with Ryan-McFarland Fortran complier. the graphic programs MSHPLT and PCPLOT are compiled under Ryan-McFarland Fortran and linked with GSS*GKS graphic library and Ryan-McFarland Fortran.

GSS*GKS graphic software is required for graphic display. If executable files are provided, users only have to purchase GSS*CGI Device Drivers from Graphic Software System Inc., 9580 SW Gemini Drive, PO Box 900, Beaveton, Oregan 9005. Their telephone number is (503) 641-2200, Fax: (503) 643-8642 and Telex: 499 4839.

To install GSS*CGI drivers, please refer to installation instructions for installing GSS*CGI device drivers supplied by GSS.

Operating System: Operating system has to be MS-DOS 3.3.

Loading PCEPFE Software Package onto Your Personal Computer:

The PCEPFE software package, which resides on several diskettes, was created by the MS-DOS command BACKUP. To load the software package onto your personal computer you may create a subdirectory named PCEPFE on your hard disk and restore all the files onto this subdirectory. If you did not create the subdirectory the RESTORE command will create one for you automatically. The following commands can be used:

(a) [path]> md PCEPFE

(b) [path]> restore a: [path]:\pcepfe*.*

The path can be either C or D drive depending on whether you have partitioned your hard disk or not. If you have not partitioned it, C is the default drive.

Running PCEPFE Software Package:

Before you execute the finite element program or its companion programs please ensure that:

(a) The GSS*GKS device drivers are properly installed.

(b) The two files, CONFIG.SYS and AUTOEXEC.BAT, are set up properly.

Now, prior to running the main finite element program PCEPFE, two input files, which are relatively simple, have to be created via screen editing sessions. The two input files are 'genin.dat' and 'epfecin.dat', respectively, for the programs MSHGEN and EPFEC. MSHGEN takes 'genin.dat' and generates an output file called 'genout.dat'. The interface program EPFEC takes 'genout.dat' and merges with the input data 'epfecin.dat' and produces an input file called 'epfein.dat' for the finite element program PCEPFE.

Before you execute PCEPFE, it is a good practice to check and verify your MSHGEN input data. The best way to check your MSHGEN input data is to plot the mesh. If MSHGEN runs successfully, it produces an output file called 'genout.dat', then you proceed to execute MSHPLT. The MSHPLT will run interactively via the screen menu. The 'genin.dat' file is modified until you are satisfied with your finite element discretization.

If the discretization of your finite element mesh is satisfied, then you can proceed to run EPFEC. The interface program EPFEC will produce an input file, epfein.dat, for PCEPFE.

After execution of PCEPFE, two additional files are created, namely: the save file - epfeout.dat and the printer-output file - epfeprt.dat.

Now, in your sub-directory, you have the following files being created:

(a) genin.dat

(b) genout.dat

(c) epfecin.dat

(d) epfein.dat

(e) epfeout.dat

In addition, there are three printer-output files, genprt.dat, epfecprt.dat and epfeprt.dat, are also created. They can be printed (if you have a 132 column printer) or deleted from the subdirectory.

In summary, 'genin.dat' is used by MSHGEN which produces an output file named as 'genout.dat'. genout.dat can be used by MSHPLT for plotting and checking the finite element mesh. The interface program EPFEC takes the input files 'epfecin.dat', merges with 'genout.dat' and creates a file called 'epfein.dat'. 'epfein.dat is the input file required by PCEPFE. After the execution of PCEPFE, it produces a 'save' file called epfeout.dat which will be used by PCPLOT for post-processing, i.e., graphical representation of stresses and displacements. No additional input will be required by PCPLOT and it is entirely interactive, user-friendly and menu-driven. More details concerning the use of MSHPLT and PCPLOT are given in References [7, 8].

REFERENCES

1. Sandhu, R.S., Wu, T.H. and Hooper, J.R.; stresses, Deformation and Progresses failure of Non-Homogeneous Fissured Rock; OSURF-3177-73-1F, Vol. 1; The Ohio State University; 1973.

2. Sandhu, R.S., Wu, T.H. and Hooper, J.R.; stresses, Deformation and Progresses failure of Non-Homogeneous Fissured Rock; OSURF-3177-73-1F, Vol. 2; The Ohio State University; 1973.

3. Yu, Y.S. and Toews, N.A.; EPFE Documentation (1977) - A Two-dimensional Elastic-Plastic Finite element Analysis Computer system; Report MRP/MRL 77-144(TR); Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

4. Yu, Y.S. and Toews, N.A.; Stresses in An Elastic-Plastic Wedge - An assessment of a Non-Linear Finite Element Program; Report MRP/MRL 79-41(TR); Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

5. Toews, N.A. and Yu, Y.S.; SAP2D Documentation (1975 Version) - 2-D Linear Elastic Finite Element Computer System; Report MRP/MRL 75-109(TR); Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

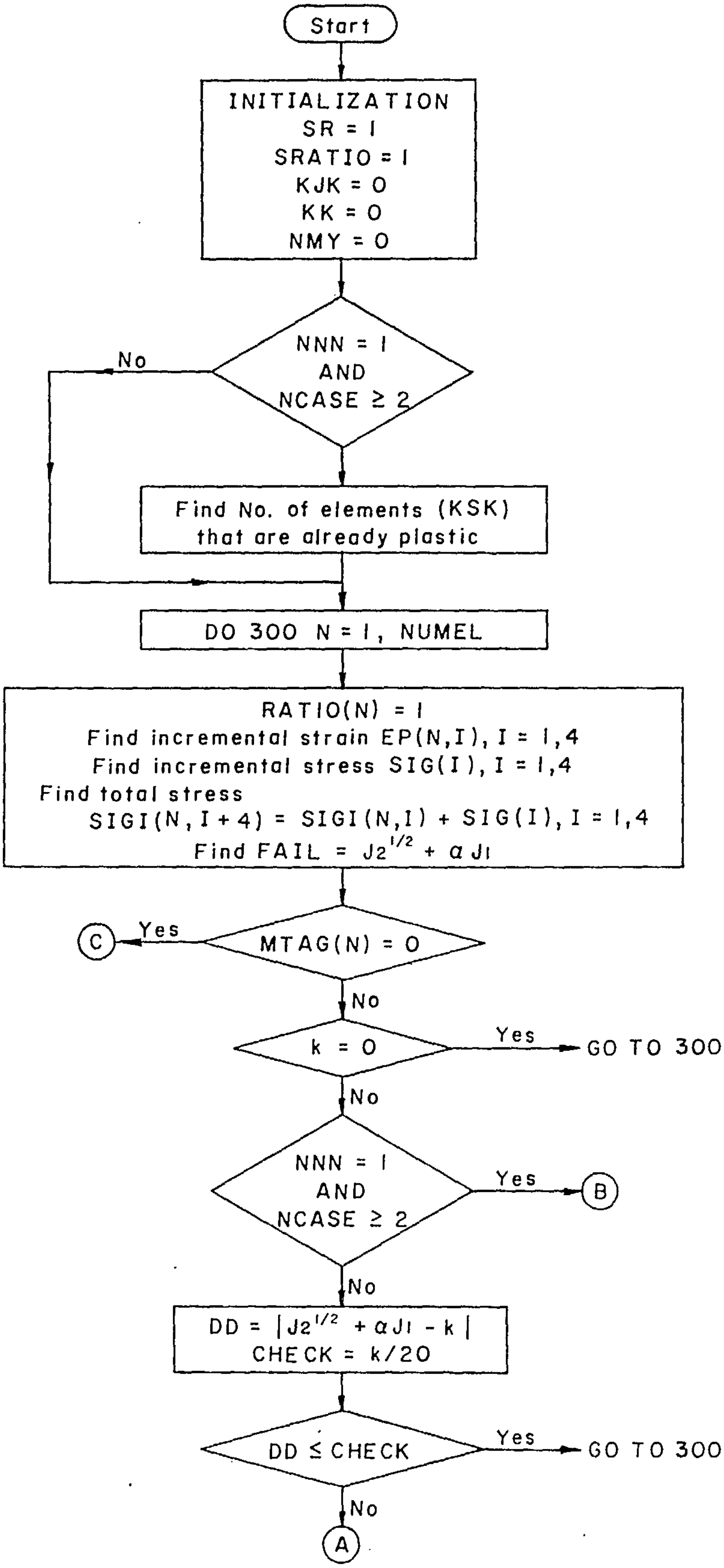
6. Y.S. Yu, N.A. Toews, A.S. Wong and H.D. Morrison; MRLPLT User's Guide - A General Purpose Plotting Program for Graphic Representation of Data Over 2-D Regions (VAX 11/750 Version); Report MRL 88-97(TR); Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

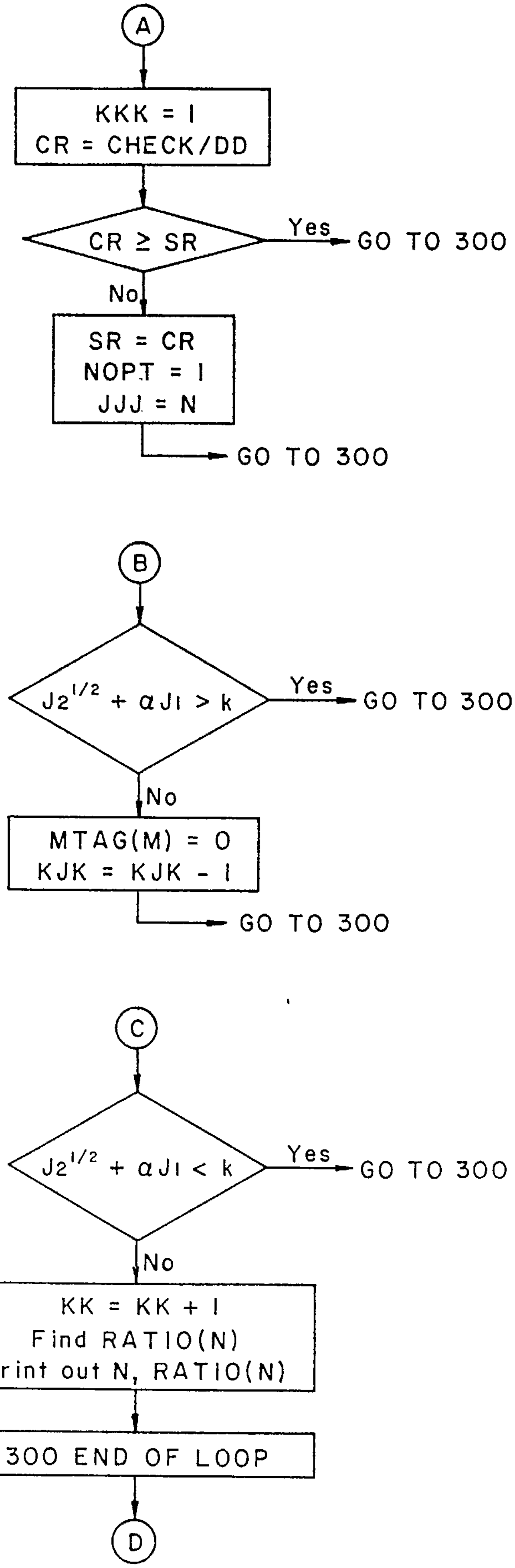
7. PCMSHPLT User's Manual (in preparation).

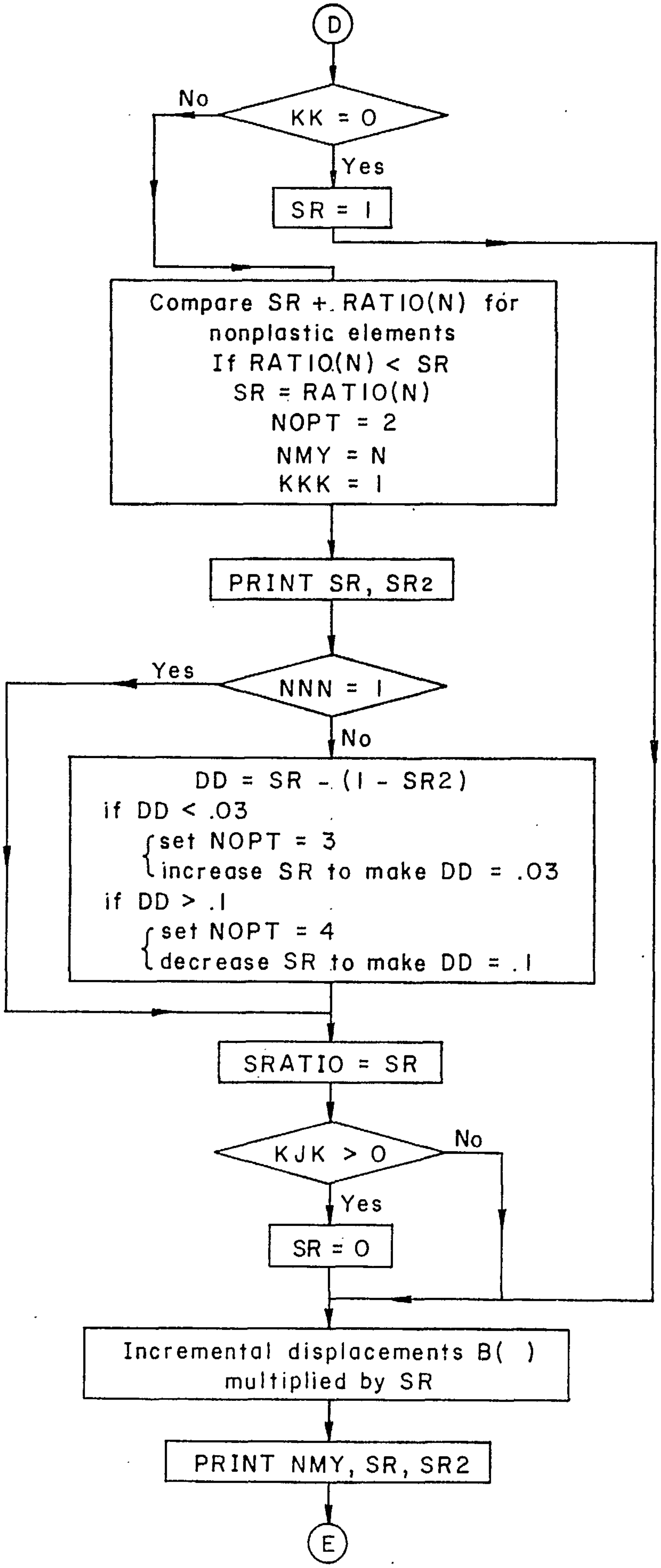
8. PCPLOT User's Manual (in preparation).

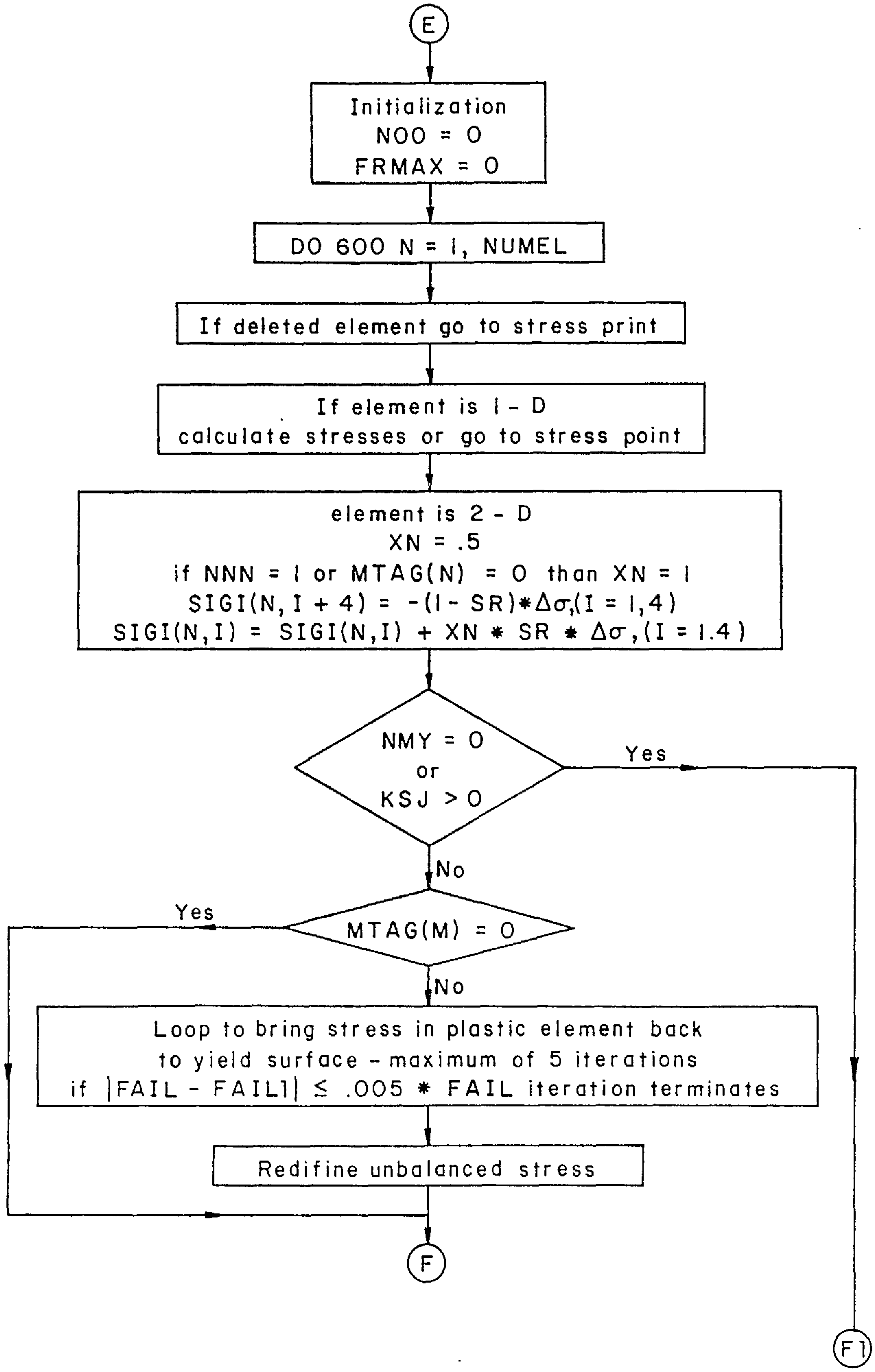
Appendix A - Flow Diagram of the Subroutine STRESS

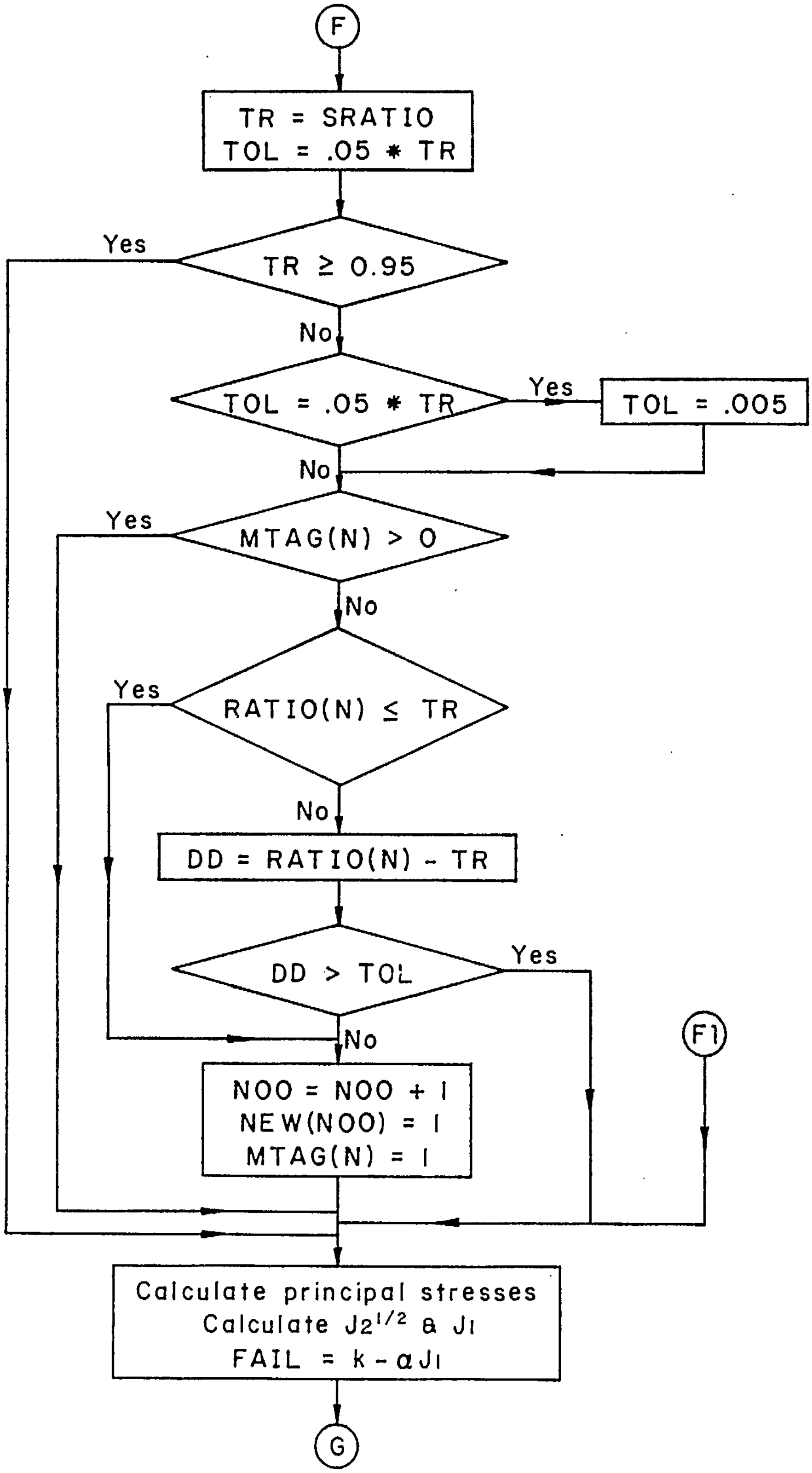
FLOW DIAGRAM FOR STRESS

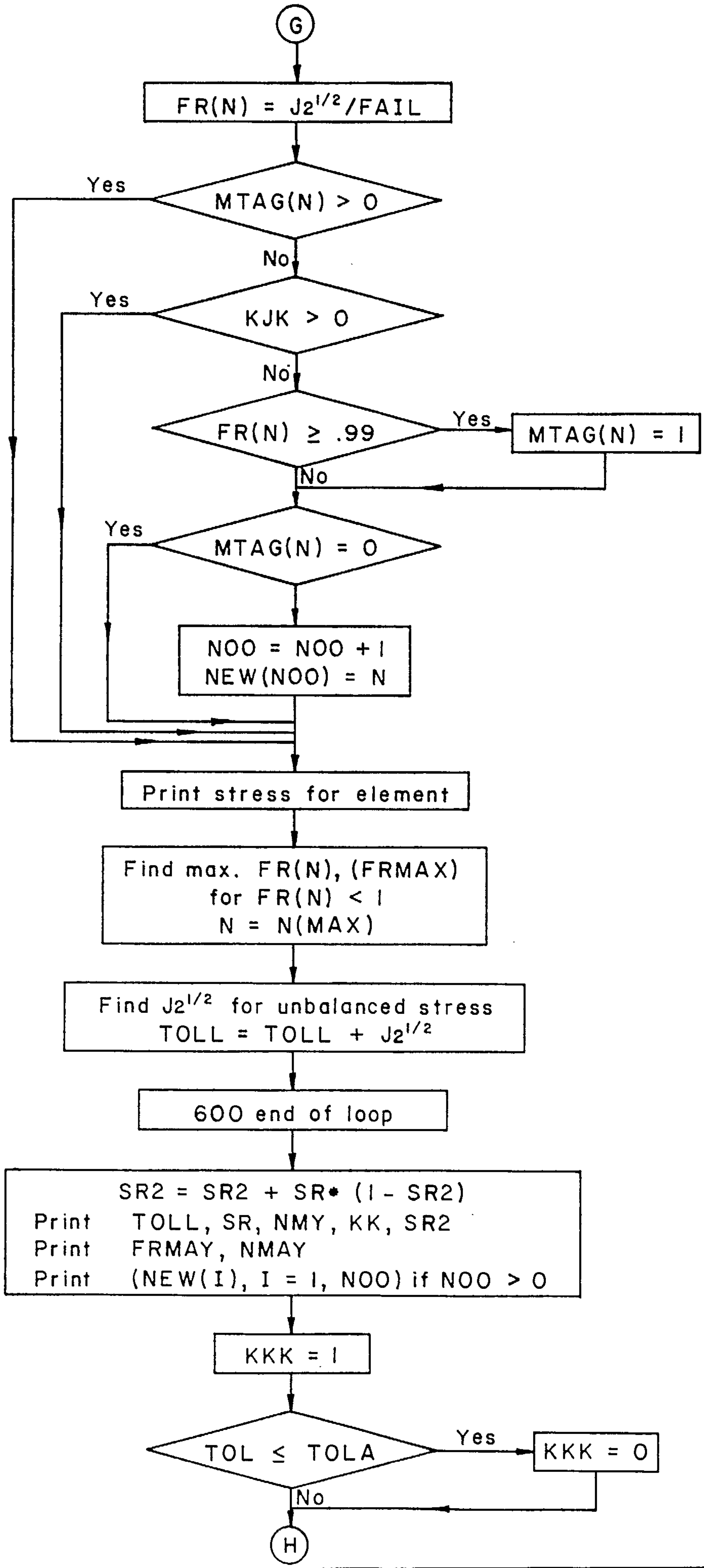


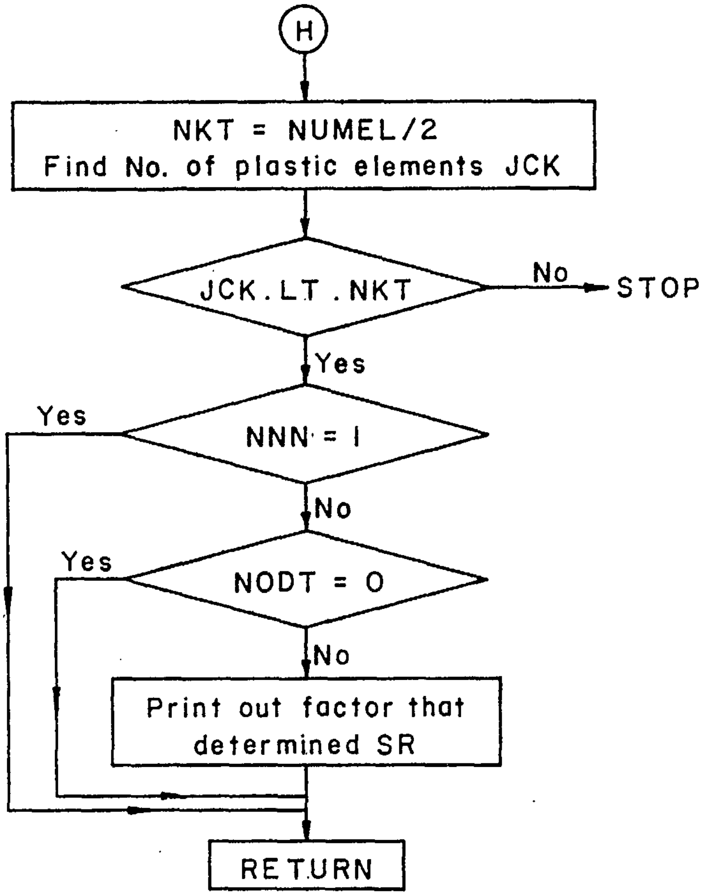

A
KKK = 1
CR = CHECK/DD
CR ≥ SR
Yes
GO TO 300
No
SR = CR
NOPT = 1
JJJ = N
GO TO 300
B
$J_2^{1/2} + \alpha J_1 > k$
Yes
GO TO 300
No
MTAG(M) = 0
KJK = KJK - 1
GO TO 300
C
$J_2^{1/2} + \alpha J_1 < k$
Yes
GO TO 300
No
KK = KK + 1
Find RATIO(N)
Print out N, RATIO(N)
300 END OF LOOP
D


D
KK = 0
No
Yes
SR = 1
Compare SR + RATIO(N) for
nonplastic elements
If RATIO(N) < SR
SR = RATIO(N)
NOPT = 2
NMY = N
KKK = 1
PRINT SR, SR2
NNN = 1
Yes
No
DD = SR - (1 - SR2)
if DD < .03
set NOPT = 3
increase SR to make DD = .03
if DD > .1
set NOPT = 4
decrease SR to make DD = .1
SRATIO = SR
KJK > 0
No
Yes
SR = 0
Incremental displacements B( )
multiplied by SR
PRINT NMY, SR, SR2
E


E
Initialization
NOO = 0
FRMAX = 0
DO 600 N = 1, NUMEL
If deleted element go to stress print
If element is 1 - D
calculate stresses or go to stress point
element is 2 - D
XN = .5
if NNN = 1 or MTAG(N) = 0 than XN = 1
SIGI(N, I + 4) = -(1 - SR)*Δσ,(I = 1,4)
SIGI(N,I) = SIGI(N,I) + XN * SR * Δσ,(I = 1.4)
NMY = 0
or
KSJ > 0
Yes
No
MTAG(M) = 0
Yes
No
Loop to bring stress in plastic element back
to yield surface - maximum of 5 iterations
if |FAIL - FAIL1| ≤ .005 * FAIL iteration terminates
Redifine unbalanced stress
F
F1


F
TR = SRATIO
TOL = .05 * TR
TR ≥ 0.95
Yes
No
TOL = .05 * TR
Yes
TOL = .005
No
MTAG(N) > 0
Yes
No
RATIO(N) ≤ TR
Yes
No
DD = RATIO(N) - TR
DD > TOL
Yes
No
F1
NOO = NOO + 1
NEW(NOO) = 1
MTAG(N) = 1
Calculate principal stresses
Calculate J2^1/2 & J1
FAIL = k - αJ1
G


G
FR(N) = J2^1/2/FAIL
MTAG(N) > 0
Yes
No
KJK > 0
Yes
No
FR(N) ≥ .99
Yes
MTAG(N) = 1
No
MTAG(N) = 0
Yes
NOO = NOO + 1
NEW(NOO) = N
Print stress for element
Find max. FR(N), (FRMAX)
for FR(N) < 1
N = N(MAX)
Find J2^1/2 for unbalanced stress
TOLL = TOLL + J2^1/2
600 end of loop
SR2 = SR2 + SR* (1 - SR2)
Print TOLL, SR, NMY, KK, SR2
Print FRMAY, NMAY
Print (NEW(I), I = 1, NOO) if NOO > 0
KKK = 1
TOL ≤ TOLA
Yes
KKK = 0
No
H


H
NKT = NUMEL/2
Find No. of plastic elements JCK
JCK.LT.NKT
No
STOP
Yes
NNN = 1
Yes
No
NODT = 0
Yes
No
Print out factor that
determined SR
RETURN

Corrections required to "PCEPFE User's Guide"
Division Report MRL 88-95 (TR)

Page No.	Para.	Comments

Please sent to:

Mining Research Laboratories
CANMET
555 Booth Street
Ottawa, Ontario, Canada
K1A 0G1
Att: Y. S. Yu