



Energy, Mines and
Resources Canada

Énergie, Mines et
Ressources Canada

1-13026 c.2

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**

PCSAP2D USER'S GUIDE - A 2-D LINEAR ELASTIC
FINITE ELEMENT STRESS ANALYSIS PACKAGE USING
A PERSONAL COMPUTER (PC Version 1.0, 1988)

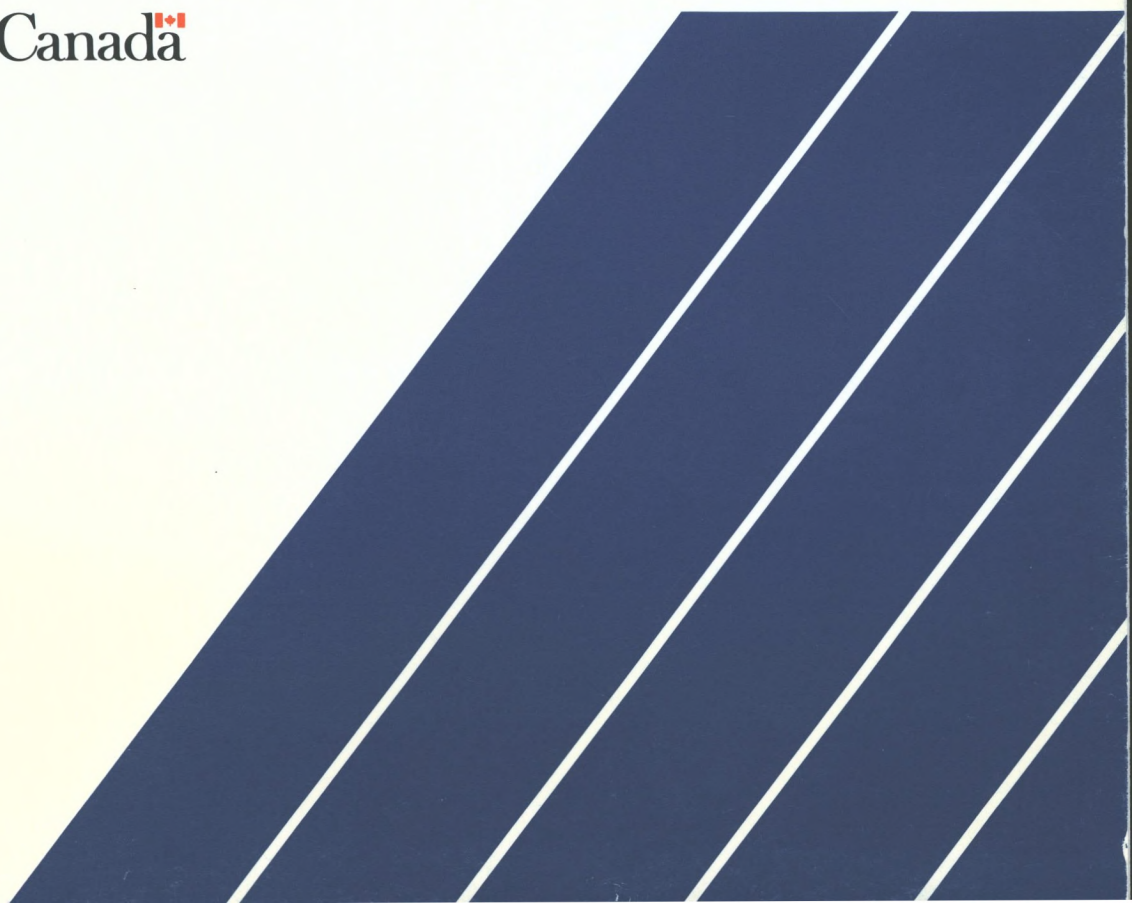
Y.S. Yu, N.A. Toews and A.S. Wong

MRL 88-113(TR)

Canada 

MRL 88-113(TR) c.2

MRL 88-113(TR) c.2



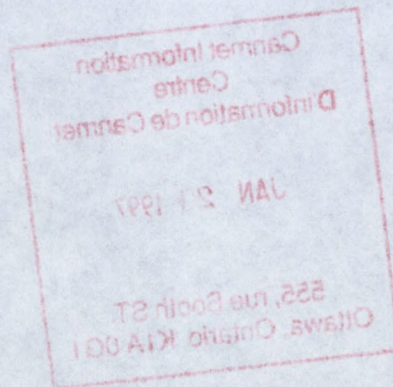
1-13026 c2
CPUB

PCSAP2D USER'S GUIDE - A 2-D LINEAR ELASTIC
FINITE ELEMENT STRESS ANALYSIS PACKAGE USING
A PERSONAL COMPUTER (PC Version 1.0, 1988)

Y.S. Yu, N.A. Toews and A.S. Wong

MRL 88-113(TR)

MRL 88-113 (TR) c.2



Canmet Information
Centre
D'information de Canmet

JAN 29 1997

555, rue Booth ST.
Ottawa, Ontario K1A 0G1

1-13026 c.2
CPUB

MRL 88-113 (TR) c.2

PCSAP2D USER'S GUIDE - A 2-D LINEAR ELASTIC FINITE
ELEMENT STRESS ANALYSIS PACKAGE USING A
PERSONAL COMPUTER (PC Version 1.0, 1988)

by

Y.S. Yu, N.A. Toews and A.S. Wong

Disclaimer

Neither the authors nor the Mining Research Laboratories, Canada Centre for Mineral and Energy Technology, can accept responsibility for the correctness of the results obtained from the use of this software package.

PCSAP2D USER'S GUIDE - A 2-D LINEAR ELASTIC FINITE
ELEMENT STRESS ANALYSIS PACKAGE USING A
PERSONAL COMPUTER (PC Version 1.0, 1988)

by

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

ABSTRACT

PCSAP2D and its companion programs, a software package which was developed based on the finite element technique using a computer, is described. PCSAP2D is the modified version of the SAP2D program which is a static, linear elastic finite element program for analysis of two-dimensional plane or axisymmetric structure.

All the capabilities of SAP2D are maintained in PCSAP2D. Initial stresses, temperature effects, simulation of excavation sequences, arbitrary distributed loading, gravity loading and concentrated loading can be handled by this program.

To run PCSAP2D program efficiently, the interface program - SAP2DC and the mesh generating system - MSHGEN have also been modified. To interpret finite element results graphically a post-processor PCPLOT was developed using GSS*GKS graphic software.

This report provides documentation and instructions 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 using a personal computer.

Key Words: finite element, axisymmetric, stresses, displacements, personal computer

* Research Scientists, ** Physical Scientist, Mining Research Laboratories, CANMET, Energy, Mines and Resources Canada, Ottawa.

GUIDE D'USAGER POUR LE LOGICIEL PCSAP2D – UN PROGICIEL POUR
MICRO-ORDINATEUR SERVANT À L'ANALYSE DE CONTRAINTE
BIDIMENSIONNELLE LINÉAIRE ÉLASTIQUE PAR LA MÉTHODE
DES ÉLÉMENTS FINIS (PC Version 1.0, 1988)

par

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

RÉSUMÉ

Le rapport décrit le progiciel pour micro-ordinateur PCSAP2D. Ce progiciel de modélisation utilise la méthode des éléments finis. PCSAP2D est une version modifiée du logiciel SAP2D, un programme de modélisation linéaire élastique utilisant la méthode des éléments finis pour l'analyse d'un plan bidimensionnel et d'une structure symétrique axiale.

Toutes les options du logiciel SAP2D sont incluses dans le progiciel PCSAP2D. Le progiciel peut accommoder les contraintes initiales, les effets de la température, la simulation des séquences d'excavation, une charge arbitrairement distribuée, une charge gravitationnelle et une charge concentrée.

Le logiciel d'interface SAP2CD ainsi que le logiciel MSHGEN pour générer un grillage ont été modifiés pour optimiser l'exécution du progiciel PCSAP2D. Un post-processeur PCPLOT utilisant le logiciel graphique GSS-GKS a été développé pour l'analyse des résultats graphiques.

Ce rapport contient la documentation et les commandes pour la préparation et validation des données d'entrée ainsi que pour l'exécution du progiciel et l'interprétation des résultats obtenus par la méthode des éléments finis.

Mots-clés: méthode des éléments finis, symétrie axiale, contraintes, déplacements, micro-ordinateur

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

CONTENTS

| | <u>Page</u> |
|--|-------------|
| Disclaimer | i |
| ABSTRACT | ii |
| RÉSUMÉ | iii |
| INTRODUCTION | 1 |
| SYSTEM OVERVIEW | 1 |
| Program PCSAP2D | 2 |
| Program MSHGEN | 2 |
| Program SAP2DC | 4 |
| Program MSHPLT | 4 |
| Program PCPLOT | 4 |
| PCSAP2D - FINITE ELEMENT PROGRAM | 4 |
| Type of Elements | 4 |
| Initial Stresses | 5 |
| Temperature Effects | 6 |
| Structural Loading | 6 |
| Cuts | 7 |
| Stress Calculation | 7 |
| Computer Memory Requirements | 7 |
| Sign Convention for Stresses and System of Units | 9 |
| Input Data Instructions | 9 |
| Output | 21 |
| MSHGEN - MESH GENERATING SYSTEM | 26 |
| Excavation Sequence | 26 |
| MSHGEN Input Data Instructions | 26 |
| SAP2DC - AN INTERFACE PROGRAM | 31 |
| SAP2DC Input Data Instructions | 31 |
| EXAMPLE PROBLEMS | 38 |
| Open Stope Mining | 38 |
| Cantilever Beam Example | 47 |
| Example With Boundary Elements | 50 |

| | <u>Page</u> |
|---|-------------|
| GETTING STARTED | 54 |
| Hardware Requirements | 54 |
| Software Requirements | 54 |
| Loading PCSAP2D Software System Onto Your Personal Computer | 56 |
| Running PCSAP2D Software Package | 56 |
| REFERENCES | 58 |
| Appendix A - Nodal Point Generation | 59 |
| Appendix B - Angle of Anisotropy | 60 |
| Appendix C - Quadrilateral Element Generation | 61 |
| Appendix D - Boundary Element Generation | 62 |

FIGURES

No.

| | |
|---|----|
| 1. PCSAP2D software system flow diagram | 3 |
| 2. The definition of axis | 6 |
| 3a. Stress calculation points for a quadrilateral | 17 |
| 3b. Stress calculation points for a triangle | 17 |
| 4. Table of NS value vs stress points | 17 |
| 5. Specification of zone N | 30 |
| 6. A hypothetical open stope mining layout - a sectional view | 39 |
| 7. A zone diagram | 40 |
| 8. Finite element mesh | 41 |
| 9. MSHGEN input data | 42 |
| 10. SAP2DC input data | 43 |
| 11. A Cantilever beam subjected to a concentrated load | 46 |
| 12. MSHGEN input data for cantilever beam example | 46 |
| 13. SAP2DC input data for cantilever beam example | 46 |
| 14. PCSAP2D input data for cantilever beam example - <code>sapin.dat</code> | 47 |
| 15. PCSAP2D example with boundary elements | 48 |
| 16. Input data for PCSAP2D example with boundary elements | 49 |
| 17. Principal material axis N-S | 58 |

INTRODUCTION

The program PCSAP2D is the personal computer version of SAP2D which is a static, linear elastic finite element program for the analysis of 2-dimensional (2-D), plane or axisymmetric structures [1]. Most of the program was extracted from Wilson's SAP program for a CDC 6000 series computer [2]. The program was modified, in 1984, to run on a VAX-11/750 Digital computer [3]. Most recently, this program was modified to run in a personal computer environment and renamed as PCSAP2D.

Initial stresses, simulation of excavation sequences, and arbitrary distributed loading, gravity loading, and concentrated force loading, can be handled by this program.

PCSAP2D is an efficiently programmed set of routines that can economically handle reasonable large sized 2-D problems. Structural analysis using the finite element technique necessarily involves large amounts of input and output data. Therefore, in order to speed up data preparation and analysis pre- and post-processors are required.

To run PCSAP2D program efficiently, the interface program - SAP2DC and the mesh-generating and plotting system - MSHGEN and MSHPLT have also been modified; these programs were extensively based on the previous work [4,5,6]. In addition, a post-processor PCPLOT was developed using GSS*GKS graphic software.

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

SYSTEM OVERVIEW

This section provides an overview to the PCSAP2D software package, and a flow diagram has been produced to show the functions of each companion program and their relationship within the PCSAP2D 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 to be Fortran 77 standard for running on a personal computer, an IBM PC/AT compatible, under the MS-DOS operating system. The plotting device for the pre- and post-processors associated with the PC version will function with a color hard-copy device. All basic plotting routines used in this program are GSS*GKS software. In addition, the pre- and post-processors are fully interactive, user-friendly and menu driven.

The PCSAP2D software system consisting of finite element program PCSAP2D 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 PCSAP2D:

PCSAP2D is a static, linear elastic finite element program for the analysis of 2-D (2-dimensional) structures (plane or axisymmetric).

Initial stresses, simulation of excavation sequences, and arbitrary distributed loading, gravity loading, as well as concentrated force loading, can be handled by this program.

The global axis is considered to be a right-handed y-z system in the plane case or a right-handed r-z system in the axisymmetric case.

The basic element used is an isoparametric quadrilateral. A triangle is treated as a degenerate quadrilateral.

Program MSHGEN:

MSHGEN is the mesh generator that produces the major portion of the finite input data for PCSAP2D. 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 boundary or spring elements.

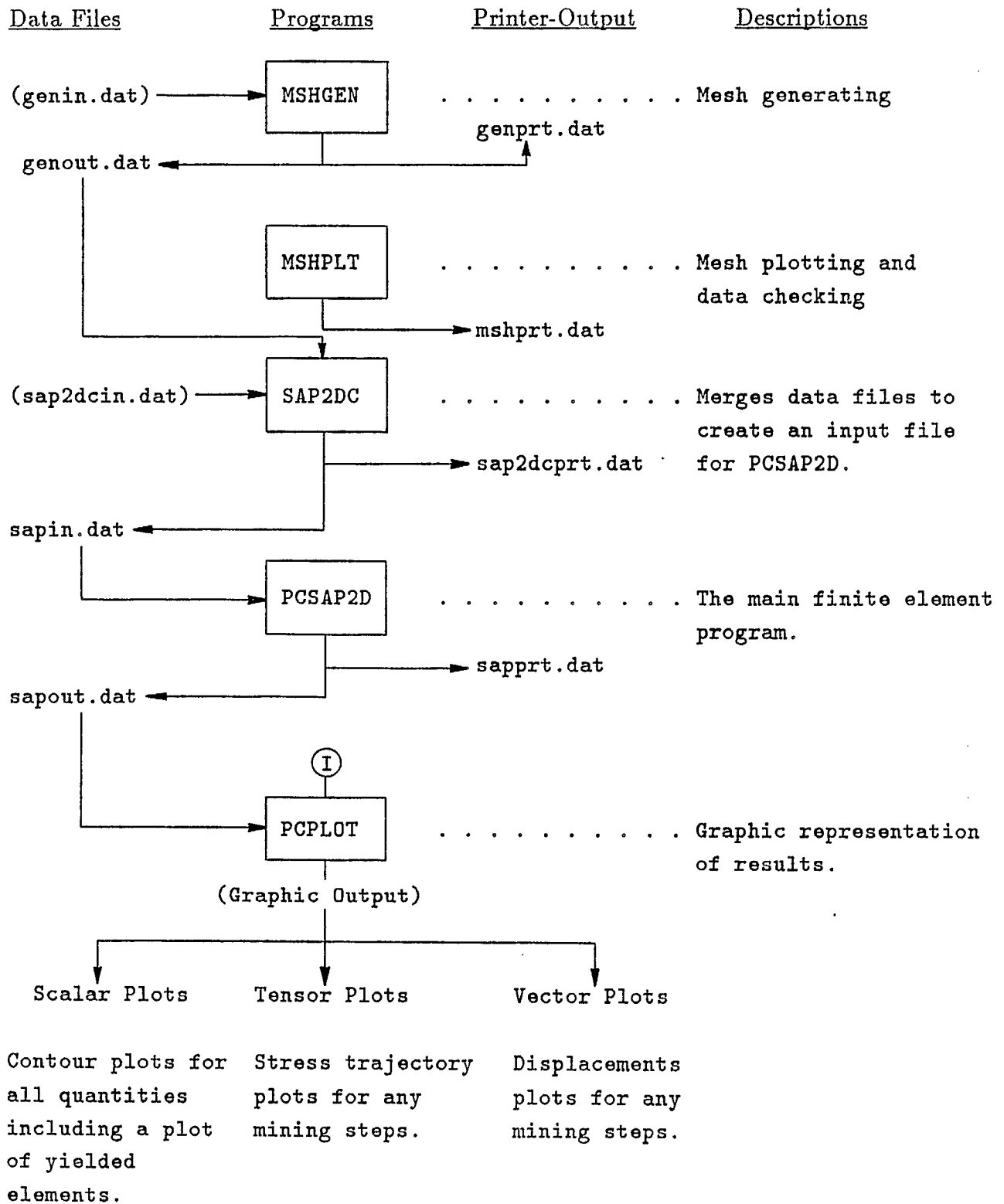


Figure 1 - PCSAP2D software system flow diagram
 (data files in brackets are user prepared, Ⓢ denotes interactive input)

Program SAP2DC:

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 PCSAP2D. Other necessary information such as material properties, initial stresses and concentrated nodal forces are absent. SAP2DC merges this information with the output file of MSHGEN, called 'genout.dat', to produce an input file ('sapin.dat') acceptable to PCSAP2D.

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 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 PCSAP2D output graphically. It converts PCSAP2D output data (stresses and displacements), and presents them in a graphic form. 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 of major or minor principal stresses.

PCSAP2D - FINITE ELEMENT PROGRAM

Type of Elements:

The basic element used is an isoparametric quadrilateral. A triangle is treated as a degenerate quadrilateral. To increase the accuracy of the element, incompatible displacement modes can be added, i.e., displacements need no longer be continuous across element boundaries, although to guarantee convergence displacement discontinuities over element sides must

tend to zero as the element size tends to zero. If not desired, these incompatible modes can be suppressed by correctly changing an input parameter.

If SAP2D is used without one-dimensional boundary elements, it is possible only to specify zero displacements in the y (or r) or z global directions. It is not possible to specify non-zero displacement or even zero displacements in other than the global system.

A boundary element is a point (has a 2×2 stiffness matrix) element with only axial stiffness (i.e., acts like a spring).

To define a boundary element it is necessary to specify two nodal points, one node gives the location of the boundary element and the second is only used to define the axial direction of the element. If no convenient direction node already exists then the user will have to create an artificial node. These artificial nodes should be fully constrained so that no equations are assembled for them.

For each boundary element a displacement and an axial stiffness is specified on the input record. The boundary element routine then applies an axial force equal to the product of the specified displacement and the axial stiffness. Now, if the stiffness specified is large compared to the stiffness of the structure at the point then, the calculated displacement at the point will be approximately equal to the specified displacement.

The effect of a spring support can be simulated by letting the specified displacement equal zero and setting the specified axial stiffness equal to the spring stiffness.

Initial Stresses:

The initial stresses are assumed to vary linearly in the direction of z and they are defined as follows:

$$\begin{aligned}\sigma_{yy} \text{ (or } \sigma_{rr}) &= AYY + BYY \times z \\ \sigma_{zz} &= AZZ + BZZ \times z \\ \sigma_{xx} \text{ (or } \sigma_{tt}) &= AXX + BXX \times z \\ \sigma_{yz} \text{ (or } \sigma_{rz}) &= AYZ + BYZ \times z\end{aligned}$$

Where σ_{yy} (or σ_{rr}), σ_{zz} and σ_{xx} (or σ_{tt}) are the normal stresses, respectively in y (or r), z and x (or t) directions. σ_{yz} (or σ_{rz}) is the shear stress in the yz (or rz) plane. AYY, BYY, AZZ, BZZ, . . . etc. are the coefficients. The definition of axis is shown in Figure 2.

The use of initial stresses is further explained in the Example Problems section.

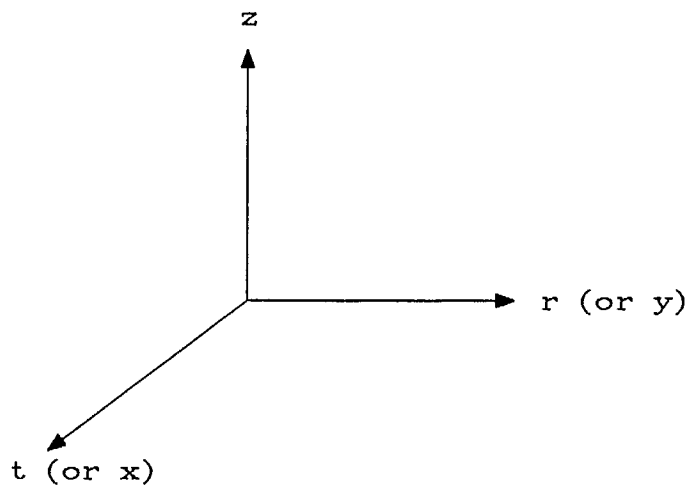


Figure 2. The definition of axis.

Temperature Effects:

Given nodal temperatures and thermal expansion coefficients SAP2D can be used to calculate thermal stresses. Reference temperatures for all elements can be input with the elements. This reference state defines zero thermal stress.

By inputting material properties versus temperature tables the variation of material properties with temperature can be taken into account.

The current version of PCSAP2D has no facility for inputting nodal temperatures. Normally these would reside on some storage device and would have to be input in the subroutine INPUTJ by suitably modifying the coding from INPUTJ from lines 66 to 68.

Structural Loading:

As well as initial stress as described above it is possible using PCSAP2D to load the structure by distributed loads (pressures and shears), gravity and concentrated nodal loads.

The major expense in running large problems is solving the relevant equations. The major component of this expense is in the gaussian decomposition of the stiffness matrix. The additional cost for solving for multiple load vectors is relatively small. To take advantage of this and so reduce the cost PCSAP2D provides the following procedure for construction of multiple load vectors.

The number of load cases "LL" is entered on the control record of the input file. For each load case the non-zero concentrated nodal loads are entered directly. The element forces

due to temperature effects, pressure and shear loading, initial stress loads and gravity loads are stored in separate arrays for each element when the element is assembled. For each load case a set of 6 load multipliers is defined. A load vector is then found by adding to the concentrated loads for that load case a linear multiple of the load multipliers and the stored element arrays for each element. Thus, by using various load multipliers, the thermal stress loads, the second the y (or z) component of the distributed loads, the third the z component of the distributed loads, the fourth the initial stress loads, the fifth the y component of the gravity and the sixth the z component of the gravity.

Cuts:

'Cuts' provide a convenient means for simulating excavation sequences. Suppose an excavation sequence is to be simulated by solving "NPROB" distinct subproblems. By associating with each element a number called a 'cut' number corresponding to the subproblem in which the element is removed (the element remains removed for higher subproblem numbers) the simulation is readily carried out. Note that an element with 'cut' number equal to one would not be used whereas an element with 'cut' number NPROB+1 would not be removed.

Stress Calculation:

By means of a parameter on the element input data the user can control where stresses are calculated for this particular element. Thus, it is possible to calculate no stresses or to calculate centroid stresses at midpoints of element sides.

The centroid stresses are calculated with respect to the global y-z coordinate system. At a midpoint node the stresses are calculated with respect to a local coordinate system defined by the outward normal of the element side containing the midpoint and the side itself with direction counterclockwise about the element.

As well as calculating the stresses as described above, the major and minor principal stresses and their angle are calculated at each stress point. The angle for the centroid stress is with respect to the y-axis and for mid-point stresses is with respect to the outward normal.

Computer Memory Requirements:

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

```
COMMON A (n)
MTOT = n
```

The value of MTOT must be greater than or equal to the following quantities:

- (a) $5 \times NUMNP + 3 \times NUMMAT + NUMMAT \times NUMTC$
- (b) $2 \times NUMNP + (NEQB + 6) \times LL$
- (c) $2 \times NEQB \times (MBAND + LL) + NEQB$
- (d) $NEQB + (NEQB + LL) + NEQB + NEQB \times LL \times (2 + (MBAND - 1)/NEQB)$
- (e) $3 \times NUMNP + 2 \times LL + NEQB \times LL$
- (f) $NEQB \times +NEQB \times +12$

If temperature effects are considered, 'NUMNP' must be added to (c) and (d).

where

NUMNP = the total number of nodal points.

NUMMAT = the number of materials.

NUMTC = The maximum number temperature records associated
with any material.

NEQB = the number of equations in a block (see below).

LL = the number of load cases (number of righthand sides).

MBAND = the half bandwidth of the global stiffness matrix.
 $= 2 \times (\text{max. nodal point difference} + 1).$

NEQ = the total number of equations.

Note that $NEQ \leq 2 \times NUMNP$ and $MBAND \leq 2 \times NPDIF + 2$, where NPDIF is the maximum nodal point difference in any element. NEQB must be at least one. For efficient running of the program make $NEQB = MBAND$ if possible.

Assuming no temperature effects, number of load cases $LL = 1$, and we let $MBAND = 120$, then the required MTOT is 29160.

The value of MTOT is set as 30000 in the program PCSAP2D. Now, suppose we have a structure which can be divided equally (number of subdivisions) along both r (or y) and z directions, say 57 elements along r (or y) direction and 57 elements along z direction. The maximum nodal point difference in this structure will be 59 and the calculated half bandwidth (MBAND) is 120. The total number of elements in the mesh will be $57 \times 57 = 3249$. Therefore, a structure with approximately 3000 elements, which is a practical sized problem, can be handled by PCSAP2D. However, even to analyze a structure of approximately 2000 elements, requires large disk storage space, approximately 10 MB.

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 PCSAP2D. 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 tensile stresses negative.

The program will accept any consistent system of units. For example, if co-ordinates are in meters, Young's moduli in kPa (kilopascal), stresses in kPa, weight density in kPa/m, then PCSAP2D will output displacements in meters and stresses in kPa. The direction of principal stress is measured from the horizontal axis (y or r-axis), and counter-clockwise positive.

If co-ordinates are in inches, Young's moduli in psi (pound per square inch), stresses in psi, weight density in pci (pound per cubic inch), then PCSAP2D will output displacements in inches and stresses in psi.

However, SI units are strongly recommended.

| | |
|------------------------|--------------------|
| Modulus of deformation | MPa or kPa |
| Unit weight of rock | MN/M**3 or KN/M**3 |
| Length | m |
| Poisson's ratio | dimensionless |
| Stresses | MPa or kPa |
| Displacements | m |

Input Data Instructions:

The input data required by PCSAP2D are divided into 9 groups, namely:

- (a) Problem Identification and Control Information
- (b) Nodal Point Input Data
- (c) Plane or Axisymmetric Element Control Data
- (d) Material Properties
- (e) Initial Stress Coefficients
- (f) Plane or Axisymmetric Element Input Data
- (g) Boundary Element Input Data

(h) Concentrated Nodal Load Input Data

(i) Load Multipliers

The detailed input data for PCSAP2d is described in the following tables 1-8:

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 - Control Parameter Data | | 9I5, F10.0 |
| NUMNP | Total number of nodal points. | |
| NELTYP | Number of element types (1 or 2). This is equal to 2 only if boundary elements as well as plane of axisymmetric elements are used. The default value is 1. | |
| LL | Number of load cases, i.e., the number of force vectors that are formed. The default value is 1. | |
| NPRNT(1) | A print indicator. If non-zero printing is performed and this applies to all NPRNT(1), NPRNT(2) AND NPRNT(3). NPRNT(1) controls printing of input data. | |
| NPRNT(2) | NPRNT(2) controls printing of displacement output. | |
| NPRNT(3) | NPRNT(3) controls printing of stress output. | |
| NPROB | The total number of subproblems. Default is 1. | |
| NRES | Initial stress indicator. If non-zero initial stresses are required to input. | |
| INDTMP | Temperature effect indicator. If this parameter is non-zero temperature effects are to be considered by program. | |
| SCALE | Scaling factor for y (or r) and z coordinates. The default value is 1. | |
| Remarks: | Material properties are not scaled by this factor, i.e., material properties must be adjusted appropriately for the dimension units. The input of initial stresses is further explained in the Example Problems Section. | |

Table 2

| Group (b) - Nodal Point Input Data: Each non-generated nodal point has a record associated with it as follows: | | |
|---|---|-----------------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1 - Nodal point data: a total of 'NUMNP' records to follow. | | 3I5, 2E15.7, I5 |
| N | Nodal number | |
| ID(N,1) | y or r constraint indicator : a zero displacement in y or r direction is indicated if this indicator is non-zero. | |
| ID(N,2) | z constraint indicator : a zero displacement in the z direction is indicated if this indicator is non-zero. | |
| Y(N) | y or r co-ordinate of node N. | |
| Z(N) | z co-ordinate of node N. | |
| KN | Node generate parameter. This is explained more fully in Appendix A. | |
| Group (c) - Plane or Axisymmetric Element Control Data: | | |
| Record No. 1 - Element Parameter Control | | 6I5 |
| NPAR(1) | The number 4. | |
| NPAR(2) | The number of plane or axisymmetric elements. | |
| NPAR(3) | The number of different materials. Default is 1. This is also the maximum material number used. | |
| NPAR(4) | Max. number of temperature records associated with any material. Default is 1. This is explained more fully below. | |
| NPAR(5) | Structure type indicator. An axisymmetric, plane strain or plane stress structure is indicated by 0, 1 or 2 respectively. | |
| NPAR(6) | Incompatible mode indicator. If non-zero then incompatible modes will be suppressed when element stiffnesses are generated. If the structure is axisymmetric incompatible modes should be suppressed. | |
| Remarks: Continued over the following page. | | |

Table 3

| Group (d) - Material Properties: With each material there is associated '1 + 2 × NTC(MAT)' records. | | |
|--|---|---------------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1: Data defining material properties. | | 2I5, 2F10.0 |
| MAT NTC(MAT) WT(MAT) WANG(MAT) | Material identification number. The number of temperature records defining this material. The default value is 1. The weight density of this material. The angle of anisotropy with respect to the y or r axis. This is explained more fully on Appendix B. | |
| Record(s) No. 2 - A total of '2 × NTC(MAT)' records defining the mechanical and thermal properties of this material as a function of temperature. Each pair of records is as follows: | | 8F10.0/3F10.0 |
| TEMP EN ES ET RNUNS RNUNT RNUST GNS | Temperature at which material has the following properties. Young's modulus in N-direction (see Figure 17). Young's modulus in S-direction. Young's modulus in T-direction. Poisson's ratio associated with N-S directions. Poisson's ratio associated with N-T directions. Poisson's ratio associated with S-T directions. Shear modulus in N-S directions. | |
| HAN HAS HAT | Coefficient of thermal expansion in N direction. Coefficient of thermal expansion in S direction. Coefficient of thermal expansion in T direction. | |
| Remarks: | Refer to Appendix B for the definition of principal material axis. | |

Table 4

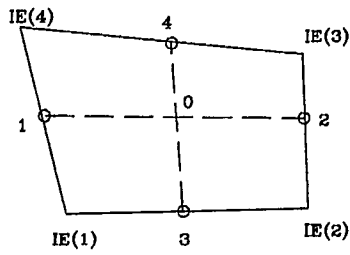
| Group (e) - Initial Stress Coefficients: If NRES is non-zero, the initial stresses need to be specified. Otherwise, proceed to Group (f). | | |
|---|---|--------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1 - Initial Stress Coefficients. | | 8E15.7 |
| AYY BYY AZZ BZZ AXX BXX AYZ BYZ | Coefficient. Coefficient. Coefficient. Coefficient. Coefficient. Coefficient. Coefficient. Coefficient. | |
| Remarks: | Where AYY, BYY, AZZ, BZZ, . . . etc. are the coefficients defining the initial stresses as described in previous section. They are assumed to vary linearly in the direction of Z and defined by the following equations: $\sigma_{yy}(\text{or } \sigma_{rr}) = AYY + BYY * Z$ $\sigma_{zz} = AZZ + BZZ * Z$ $\sigma_{xx}(\text{or } \sigma_{tt}) = AXX + BXX * Z$ $\sigma_{yz}(\text{or } \sigma_{rz}) = AYZ + BYZ * Z$ | |

Table 5

| Group (f) - Plane or Axisymmetric Element Specification: | | |
|---|--|--------------|
| <p>Each non-generated element has associated with it a total number of '1 + NUMPC' input records (an element record and 'NUMPC' pressure records associated with the element). Elements must be entered in ascending order. Omitted element are generated as described below.</p> | | |
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1 - Element Data. | | 10I5, 2F10.0 |
| M (IE(I),I=1,4) | The element number. An array of length 4 which contains the nodal numbers defining the element. These must be in counterclockwise order. If IE(3) = IE(4) or if IE(4) = 0, or blank, then, the element is assumed to be triangular. | |
| MAT | The material number of the element. The default value is 1. | |
| NCUT | The cut number associated with the element. The default value is 'NPROB + 1', i.e., the element will never be removed. | |
| NUMPC | The number of element sides that carry distributed pressure and shear loads. Also the number of records that follow this record and carry distributed load information. The maximum value for NUMPC is 4. (It is assumed that pressure and shear vary linearly over the side of an element. Shear stress is considered positive if it tends to turn an element counter-clockwise.) | |
| NS | Stress point calculation indicator. Stresses can be calculated at the centroid and at mid-points of element sides depending on the value of this parameter. See Figures 3a, 3b and 4 for more information. | |
| KG | Element generation parameter. Default value is 1. This is more fully described in Appendix C. | |
| REFT | The references temperature associated with the element. | |
| THICK | Element thickness. It has meaning only in the plane stress case. | |
| Remarks: | Continued over the following page. | |

Table 5 (continued)

| Group (f) - Plane or Axisymmetric Element Specification: (Continued) | | |
|--|---|-----------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 2 - Pressure records. If 'NUMPC' is greater than zero, then 'NUMPC' records follow: | | I5,4F10.0 |
| J PRES(1,J) PRES(2,J) SHEAR(1,J) SHEAR(2,J) | The side number of the element. Side IE(1) - IE(2) has side number 1, IE(2) - IE(3) side number 2, IE(3) - IE(4) side number 3 and IE(4) - IE(1) side number 4 assuming element is a quadrilateral. Pressure at first node of side J. Pressure at second node of side J. Shear at first node of side J. Shear at second node of side J. | |
| Remarks: | | |



Stress calculation points 1, 2, 3 and 4 are mid-points of sides.

The "centroid" point is defined by joining the mid-points.

Figure 3a. Stress calculation points for a quadrilateral.

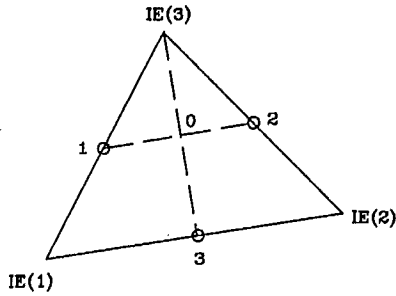


Figure 3b. Stress calculation points for a triangle.

| NS | STRESS POINTS |
|----|---------------|
| 1 | none |
| 0 | 0 |
| 8 | 0, 1 |
| 16 | 0, 1, 2, 3 |
| 20 | 0, 1, 2, 3, 4 |

Figure 4. Table of NS value vs stress points.

Table 6

| <p>Group (g) - Boundary Element Data: If no boundary elements are specified (i.e., NELTYP on the problem control record equals 1), then, no data will be required and proceed to Group (h).</p> | | |
|--|--|-------------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1 - Element Parameter Data: | | 2I5 |
| NPAR(1) NPAR(2) | Element type indicator. For a boundary element, NPAR(1) = 7. The total number of boundary elements. | |
| Record No. 2 - Boundary Element Specification: Element records must be in ascending sequence starting with element 1 and ending with element NPAR(2). Omitted elements are generated as described below: | | 5I5, 2E15.6 |
| NEL NP NI KN NCUT SD TRACE | Element number. Node number of node where element is located. Node number of node that with NP defines the axial direction of the boundary element. Direction NP to NI is considered positive. If NI is zero or blank, then, the direction defined by previous input data is used. Element generation parameter. Default value is 1. This is described more fully in Appendix D. Cut number associated with this element. If zero, it is taken to be 'NPROB + 1', i.e., the element is never removed. Specified displacement. Note that the direction from location node to direction node is considered positive. Axial stiffness of element. If zero it is taken to be 10^{10} . | |
| Remarks: | | |

Table 7

| Group (h) - Concentrated Nodal Load Input Data: | | |
|---|---|-------------|
| Variable(s) | Variable Definition or Description | Format |
| Record No. 1 - Concentrated Nodal Load: | | 2I5, 2F10.4 |
| N | The nodal number of node where concentrated force appears. | |
| L | The load case. Possible values of L are from 1 to LL. | |
| R(1) | y or r component of force. | |
| R(2) | z component of force. | |
| Remarks: | If no concentrated load data occurs then a blank line or record must appear here. Only non-zero concentrated nodal load data need be input. Data must appear in non-decreasing nodal number sequence. A node is repeated for each load case with non-zero nodal load at this node. These load cases, for a given node, can be on any order. | |

Table 8

| | | |
|--|---|--------|
| <p>Group (i) - Load Multipliers: 'LL' records (the number of load cases) must appear here. Each record defines one load vector as the sum of the corresponding concentrated load case and as linear combination of the load multipliers and the thermal stress loads, distributed pressure and the shear loads, initial stress loads and gravity loads respectively.</p> | | |
| Record No. 1 - Load Multipliers: | | 6F10.0 |
| STR(1,L) | The thermal stress multiplier for load case L. | |
| STR(2,L) | The y or r component multiplier of the distributed load for the load case L. This also multiplies the y or r component of the boundary element loads. | |
| STR(3,L) | The z component multiplier of the distributed load for load case L. This also multiplies the Z component of the boundary forces loads. | |
| STR(4,L) | The initial stress multiplier for the load case L. | |
| STR(5,L) | The y or r component multiplier of the gravity load for the load case L. | |
| STR(6,L) | The z component multiplier of the gravity load for the load case L. | |
| Remarks: | | |

Output:

Printer Output includes input data such as problem identification, control parameters, material properties, initial stresses, nodal point coordinates and element connectivity, etc., will be printed out if specified. Displacements and stresses of each mining step or subproblem may or may not be printed depending on the print control parameters (please refer to `epfec` input, p.33, for details).

Save File - 'sapout.dat' is to facilitate graphical representation of stresses and displacements or carrying out a stability analysis based on the stresses determined from the model certain data is written onto a disk file and saved for latter processing. There are a total of $(5 + 2 \times \text{NSTEP})$ files to be written on this save file. The logical name is call 'sapout.dat' and its contents are described in the following tables 9a, 9b, 9c and 9d.

Table 9a

| Contents of Save File - 'sapout.dat' | |
|---|---|
| (a) Subproblem or Mining Step Information: | |
| | <p>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: NPROB - total number of subproblems or mining steps, and NRES - the initial stress indicator (not used in post-processing).</p> |
| Co-ordinates of Nodal Points: | |
| | <p>Consists of one recorder header - Format (1H , 4I10, 18A4, 2X). Contents of header are NC, NC1, NUMNP, NDUM, TITLE. Here NC = 0, NC1 = 1, NUMPNP 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 Y and Z co-ordinates of all (NUMNP) nodal points: (Y(I), Z(I), I = 1, NUMNP) in Format (1X, 2E15.7).</p> |
| (c) Material Properties: | |
| | <p>Consists of one record header - Format (1H ,4I10, 18A4, 2X) Contents of header are NC, NC1, NREC, NUMMAT, TITLE. Here NC = 0, NC1 = 2, NREC = number of records stored in this file, NUMMAT is the total number of materials, and TITLE is an 80 character title associated with the problem. For each material there are associated with $(2 \times NTC + 1)$ records: where NTC is the number of temperature records; the first record contains M, NTC, WT, WANG in Format (1H , 2I10, 2E13.5). The rest of the file contents (E(I,J,M), J=1,11) in Format (1H , 8E13.6/1H , 3E13.6)</p> |
| Remarks: Continued over the following page. | |

Contents of Save File - 'sapout.dat' (continued)

(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 = 4, NDUM = 0, and TITLE is a 80 character title associated with the problem.

The rest of the file contains four records of the initial stress coefficients, i.e., AYY, BYY, AZZ, BZZ, AXX, BXX, AYZ, and BYZ in Format (1X, 2E15.7).

Remarks: continued over the following page.

Contents of Save File - 'sapout.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 = 1,
 and TITLE is an eighteen word vector containing the problem title.
 The rest of the file contains 'NUMEL' records with Format (1H ,7I10);
 and contents of each record are: N, (IX(I), I = 1, 4), MAT, NCUT.
 Where N is the element number, IX(1), ... IX(4) are the nodal points
 defining the element, MAT is the material number, and NCUT is the cut number
 associated with the element.

The above data is followed by $2 \times$ NPROB files, i.e., for each subproblem
 (1 to NPROB) there is associated one file containing the nodal displacements
 and a second file 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).
 The contents are NC, NC1, NNP, LL, and HEAD.
 Where NC is the mining step or subproblem number, NC1 = 1,
 NNP equals the number of non-deleted nodal points, LL is the load cases
 and HEAD is an eighteen word vector containing the problem title.
 The header is followed by 'LL \times NNP' records containing the
 displacements. The 'LL' displacements associated with a nodal point are
 grouped together.
 The nodal points associated displacements are stored in reverse order, i.e.,
 The non-deleted nodal point with the largest nodal point number occurs first
 and non-deleted nodal point with smallest nodal number occurs last.
 The contents 'LL' records associated with node N are:
 (N, L, DY(L), DZ(L), L=1,LL) in Format (1H ,I10, I5, 2E15.7).
 Where N is the nodal point number, L is the load case number (1 to LL),
 DY(L) and DZ(L) are the y or r (the horizontal) and z (vertical)
 displacements associated with load case L.

Remarks: continued over the following page.

Contents of Save File - 'sapout.dat' (continued)

(f) Element Stresses:

The stress output associated with subproblem NC is as follows:

One record header - Format (1H ,4I10,18A4,2X)

The contents of header are NC, NC1, NUME, LL, HEAD.

Here NC is the mining step or subproblem number, NC1 = 2, NUME is the number of non-deleted elements, LL is the number of load cases and HEAD is an eighteen word title associated with problem.

The header is followed by records giving the stresses in each non-deleted element for each load case. With each element it is associated NS×LL records where NS is the number of stress calculation points in the element and LL is the number of load cases.

The contents for each record are MM, L, STRLAB, (SIG(I), I=1, 7), YC, ZC. MM is the element number, L is the load case number, STRLAB is a 3 character label defining the stress calculation point (for stress at the centroid STRLAB = 'CEN'). (SIG(1), SIG(2), SIG(3) are the stress components, respectively, in the direction of y (or r), z and x (or t); SIG(4) is the shear stress in the yz (or rz) plane. SIG(5) and SIG(6) are the principal stresses, SIG(7) is the angle in degrees that the major principal stress makes with the y (or r) axis. The angle is measured, counter-clockwise positive, from the y (or r) axis to the major principal stress. YC and ZC are the y (or r) and z co-ordinates of the centroid of each element. No stresses are output for the boundary elements. The format is : (1H ,2I5,A3,9E13.7)

Remarks:

MSHGEN - MESH GENERATING SYSTEM

Excavation Sequence:

PCSAP2D has no capability to simulate construction (such as backfill in mines), however, excavation can be conveniently simulated by associating with each element a number call 'cut' number corresponding to the subproblem in which the element is removed.

The 'cut' numbers are generated by the mesh-generating system MSHGEN. However, it is important that when MSHGEN is used an integer must be assigned for the the 'cut' number if a zone of the structure is involved with excavation sequences. For example, elements with a 'cut' number 02 indicates that these elements are to be removed in the second subproblem (the integer 02); elements with a 'cut' number 03 indicates that these elements are to be removed in the third subproblem (the integer 03). Elements with a 'cut' number 'NPROB + 1' means that these elements will never be removed. 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]. The input data required for MSHGEN is simple and extracted below for easy reference. For users who are not familiar 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 are 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 10

| 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: | | |
| NSPNP | Total number of specified nodal points. | |
| NVZONE | Total number of non-void zones. | |
| NSPAN1 | Total number of Spans in ϵ direction. | |
| NSPAN2 | Total number of spans in η direction. | |
| NPROB | The total number of subproblems or incremental mining steps making up the excavation and or construction sequence. Default value is 1. | |
| NSIDNT | Identification indicator, a value of zero or blank means no identification; NSIDNT = 1, indicates identification. Default = 0 for most of the meshes generated. | |
| Line No. 3 - data defining zone subdivisions: | | |
| NSBDV1(I) | Array defining number of subdivisions in each zone in ξ direction. | |
| BSBDV2(I) | Array defining number of subdivisions in each zone in η direction. | |
| Remarks: | | |

Table 11a

| 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 NCODSP(N) | Specified nodal point number. Constraint code of specified nodal point N. The table below defines the possibilities: <table border="0" style="margin-left: 40px;"> <thead> <tr> <th style="text-align: left;"><u>NCODSP(N)</u></th> <th style="text-align: left;"><u>Constraint at Node N</u></th> </tr> </thead> <tbody> <tr> <td>0</td> <td>No constraint on displacements.</td> </tr> <tr> <td>1</td> <td>y-displacement = 0.</td> </tr> <tr> <td>2</td> <td>z-displacement = 0.</td> </tr> <tr> <td>3</td> <td>y-displacement = 0, z-displacement = 0.</td> </tr> </tbody> </table> | <u>NCODSP(N)</u> | <u>Constraint at Node N</u> | 0 | No constraint on displacements. | 1 | y-displacement = 0. | 2 | z-displacement = 0. | 3 | y-displacement = 0, z-displacement = 0. | |
| <u>NCODSP(N)</u> | <u>Constraint at Node N</u> | | | | | | | | | | | |
| 0 | No constraint on displacements. | | | | | | | | | | | |
| 1 | y-displacement = 0. | | | | | | | | | | | |
| 2 | z-displacement = 0. | | | | | | | | | | | |
| 3 | y-displacement = 0, z-displacement = 0. | | | | | | | | | | | |
| XSP(N) | y-coordinate (horizontal) of node N. | | | | | | | | | | | |
| YSP(N) | z-coordinate (vertical) of node N. | | | | | | | | | | | |
| Group (c) - Block of Data Defining Non-void Zones: For each non-void zone the following records must be entered. | | | | | | | | | | | | |
| 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) | | | | | | | | | | | |
| NUMPC | 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 following page. | | | | | | | | | | | |

Table 11b

| 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 on the sides of zone N as follow: | | |
| NSIDE | The side number of the zone N. See Figure 5 for the number associated with sides. | |
| P1 | The pressure at node 1 of zone side 'NSIDE'. | |
| P2 | The pressure at node 2 of zone side 'NSIDE'. | |
| Remarks: | | |
| Note 1: | <p>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 rigid. Order of specification is counter-clockwise starting with the node at $(\text{MIN}(\xi), \text{MIN}(\eta))$.</p> | |
| Note 2: | <p>Material Specification: Different materials 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.</p> | |
| Note 3. | <p>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 not be used. Elements with cut number greater than 'NPROB' (the maximum number of subproblem) will not be deleted.</p> | |

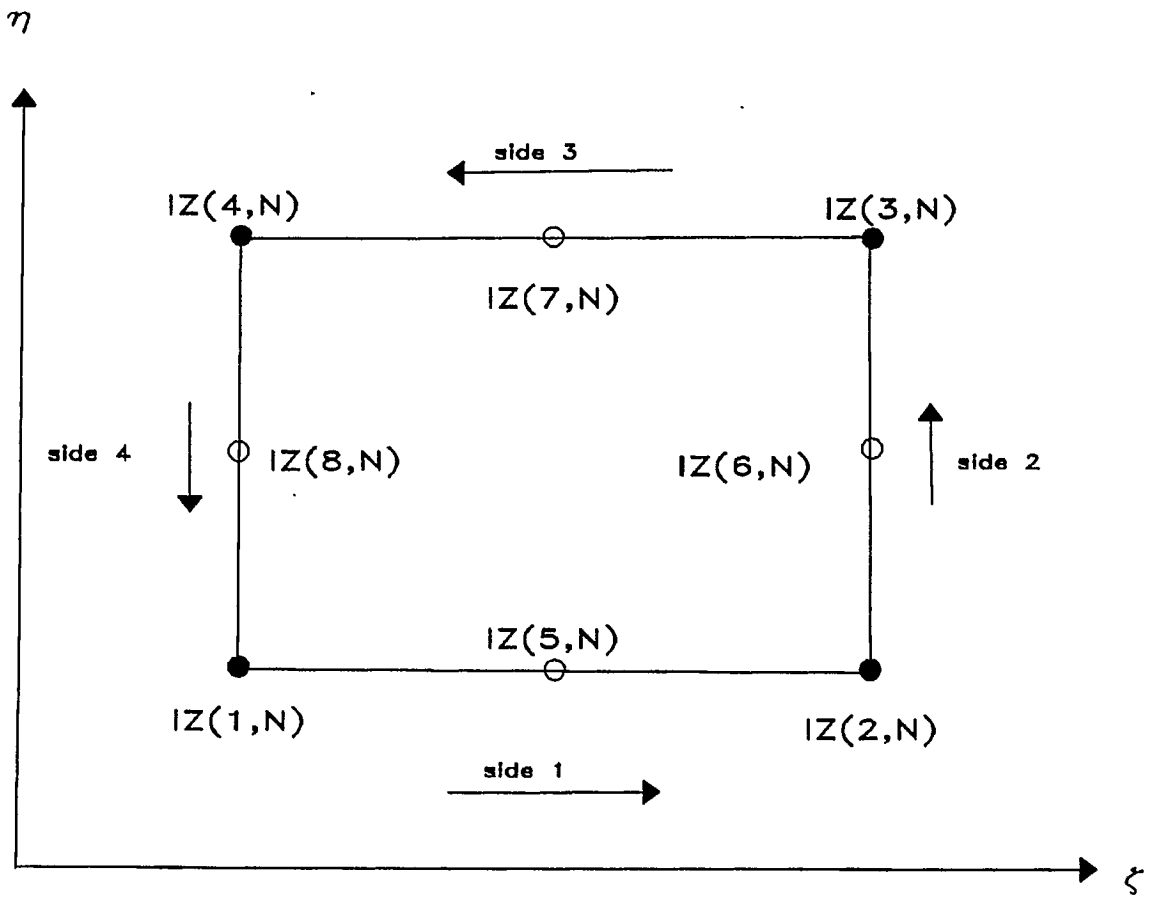


Figure 5. Specification of zone N.

SAP2DC - AN INTERFACE PROGRAM

The mesh-generating system MSHGEN produces only part of the input data required by the program PCSAP2D such as the nodal points coordinates and elements data. Other information such as material properties, concentrated nodal forces, initial stresses, etc., are absent. SAP2DC is an interfacing program which merges this additional information with the output ('genout.dat') of MSHGEN to produce an input file ('sabin.dat') acceptable to PCSAP2D. It should be noted that the following restrictions apply:

- (a) No thermal effects,
- (b) Only one load case,
- (c) No boundary elements, and
- (d) Gravity loads, if any, occur in (-Z) direction.

SAP2DC Input Data Instructions:

The input data for SAP2DC are divided into four groups, namely:

- (a) Problem identification and control information,
- (b) Material properties of rock formations,
- (c) Initial stress coefficients, and
- (d) Concentrated nodal forces.

The detail of input requirement is described in the following tables 12-15. 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 Example Problems Section.

Table 12

| Group (a) - Problem Identification and Control Data: | | |
|--|--|--------|
| Variable(s) | Variable Definition or Description | Format |
| Line No. 1 - Problem control information: | | |
| TITLE | A 80 character title. Default - if the record is blank, then, the MSHGEN title is used. | A80 |
| Line No. 2 - Control parameters: | | (Free) |
| NP | Type of structure indicator. NP = 0 for plane strain, NP = 1 for axisymmetric, and NP = -1 for plane stress. | |
| NRES | An indicator for input initial stress: If NRES is nonzero, initial stresses need to be specified and PCSAP2D will attempt to calculate initial stresses using the subroutine CALRS4. If NRES is zero, the program will set initial stresses to zero and input will not be required. | |
| IMAT | An anisotropic material properties indicator. If IMAT = 0, then all materials are assumed to be isotropic. If IMAT is not equal to zero then one or more materials have anisotropic properties. | |
| ICF | Concentrated force indicator. If ICF = 0 then no concentrated forces are input. If ICF is nonzero then concentrated forces are input. | |
| ICOMP | Incompatible mode indicator. If ICOMP is nonzero then incompatible modes are suppressed in calculating the stiffness matrix in PCSAP2D. If the structure is axisymmetric incompatible modes are always suppressed. | |
| SCALE | Coordinate scaling factor. Coordinates generated from MSHGEN will be multiplied by this factor. The default is 1.0 | |
| Remarks: | Continued over the following page. | |

Table 12 (continued)

| Group (a) - Problem Identification and Control Data: (continued) | | |
|--|---|--------|
| Variable(s) | Variable Definition or Description | Format |
| Line No. 3 - Output Print Control: | | (Free) |
| NPRINP | Input print indicator. If nonzero, the printing of input data will be suppressed in PCSAP2D. | |
| NPRDIS | Displacement print indicator. If nonzero, the printing of PCSAP2D displacement output will be suppressed. | |
| NPRSTR | Stress print indicator. If nonzero, the printing of PCSAP2D stress output will be suppressed. | |
| Remarks: | | |

Table 13

| Group (b) - Material Properties of Rock Formations: | | |
|--|--|------------------|
| Variable(s) | Variable Definition or Description | Format (Free) |
| Either record No. 1: If IMAT = 0 (isotropic material properties), then there are 'NUMMAT' records to be entered. | | |
| N E RNU WGT | Material identification number. Young's modulus of deformation. Poisson's ratio. Weight density of material. | |
| Or record block No. 2: If IMAT is not equal 0 (anisotropic material properties), then, there are a total of '3×NUMMAT' records to follow assuming that the number of temperature records (NTC(MAT)) which defines the material is 1. The format and content of these records are identical to that given in PCSAP2D. | | |
| First record | | |
| MAT NTC(MAT) WGT(MAT) WANG(MAT) | Material number. The number of temperature records defining this material. the default is 1. The weight density of this material. The angle of anisotropic with respect to Y or R axis. | |
| Remarks: | Continued over the following page. | |

Table 13 (continued)

| Group (b) - Material Properties of Rock Formations: (continued) | | |
|---|--|---------------|
| Variable(s) | Variable Definition or Description | Format (Free) |
| Second record: | | |
| EN | Young's modulus in N-direction. | |
| ES | Young's modulus in S-direction. | |
| ET | Young's modulus in T-direction. | |
| RNUNS | Poisson's ratio associated with N-S directions. | |
| RNUNT | Poisson's ratio associated with N-Y directions. | |
| RNUST | Poisson's ratio associated with S-T directions. | |
| GNS | Shear modulus in N-S direction. | |
| Third record: | | |
| HAN | Coefficient of thermal expansion in N direction. | |
| HAS | Coefficient of thermal expansion in S direction. | |
| HAT | Coefficient of thermal expansion in T direction. | |
| Remarks: Refer to Appendix B for the definition of principal material axis. | | |

Table 14

| Group (d) - Initial Stress Coefficients: Required only if NRES is not equal to zero. | | |
|--|------------------------------------|------------------|
| Variable(s) | Variable Definition or Description | Format (Free) |
| Record No. 1 | | |
| AYY BYY | Coefficient Coefficient | |
| Record No. 2 | | |
| AZZ BZZ | Coefficient Coefficient | |
| Record No. 3 | | |
| AXX BXX | Coefficient Coefficient | |
| Record No. 4 | | |
| AYZ BYZ | Coefficient Coefficient | |
| <p>The initial stresses are assumed to be varying linearly with depth, Z. The above coefficients are better illustrated by the following equations:</p> $\sigma_{yy} = AYY + BYY \times Z$ $\sigma_{zz} = AZZ + BZZ \times Z$ $\sigma_{xx} = AXX + BXX \times Z$ $\sigma_{yz} = AYZ + BYZ \times Z$ <p>Where σ_{yy}, σ_{zz} and σ_{xx} are the initial stresses in the horizontal, vertical and transverse directions, respectively. σ_{yz} is the shearing stress in the yz plane. Z is the depth.</p> | | |

Table 15

| Group (e) - Concentrated Nodal Forces: Required only if ICF is not equal to zero. | | |
|---|--|------------------|
| Variable(s) | Variable Definition or Description | Format (Free) |
| Record No. 1 | | |
| N R1 R2 | The number of node where nodal forces are acting. y component of force (horizontal) at node N. z component of force (vertical) at node N. | |
| Remarks: 1. Repeat record No. 1 as many times as required. 2. The load multipliers, required by PCSAP2D are generated automatically by SAP2DC. | | |

EXAMPLE PROBLEMS

Open Stope Mining

A hypothetical open stope 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 6 shows the schematic diagram of the mining geometry, and its associated zone diagram is shown in Figure 7. Figure 8 shows the corresponding finite element mesh. The input data required by MSHGEN to generate part of the input required by PCSAP2D is shown in Figure 9. Additional input data required by SAP2DC, as shown in Figure 10, is then merged with MSHGEN output data ('genout.dat') to produce an input file which is acceptable to PCSAP2D.

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

$$\sigma_{yy} = a_{yy} + b_{yy} \times Z$$

$$\sigma_{zz} = a_{zz} + b_{zz} \times Z$$

$$\sigma_{xx} = a_{xx} + b_{xx} \times Z$$

$$\sigma_{yz} = a_{yz} + b_{yz} \times Z$$

where σ_{yy} , σ_{zz} are the horizontal and vertical stresses respectively. σ_{xx} is the stress perpendicular to the yz plane and σ_{yz} is the shearing stress in the yz plane. a_{yy} , b_{yy} , a_{zz} are the coefficients relating the stress components with depth. Z is the depth at which the stresses are evaluated.

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.

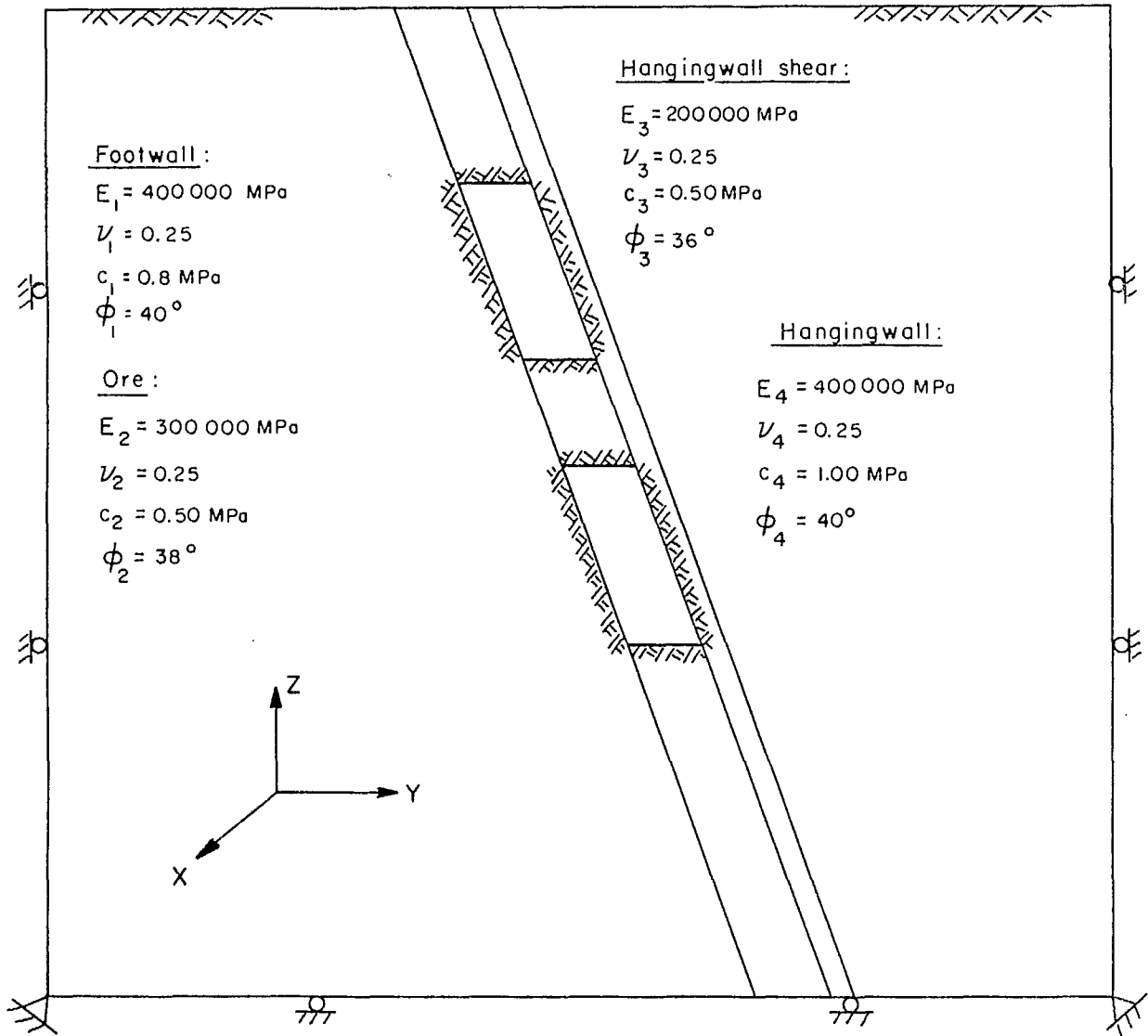


Figure 6 A hypothetical open stope mining layout - a sectional view

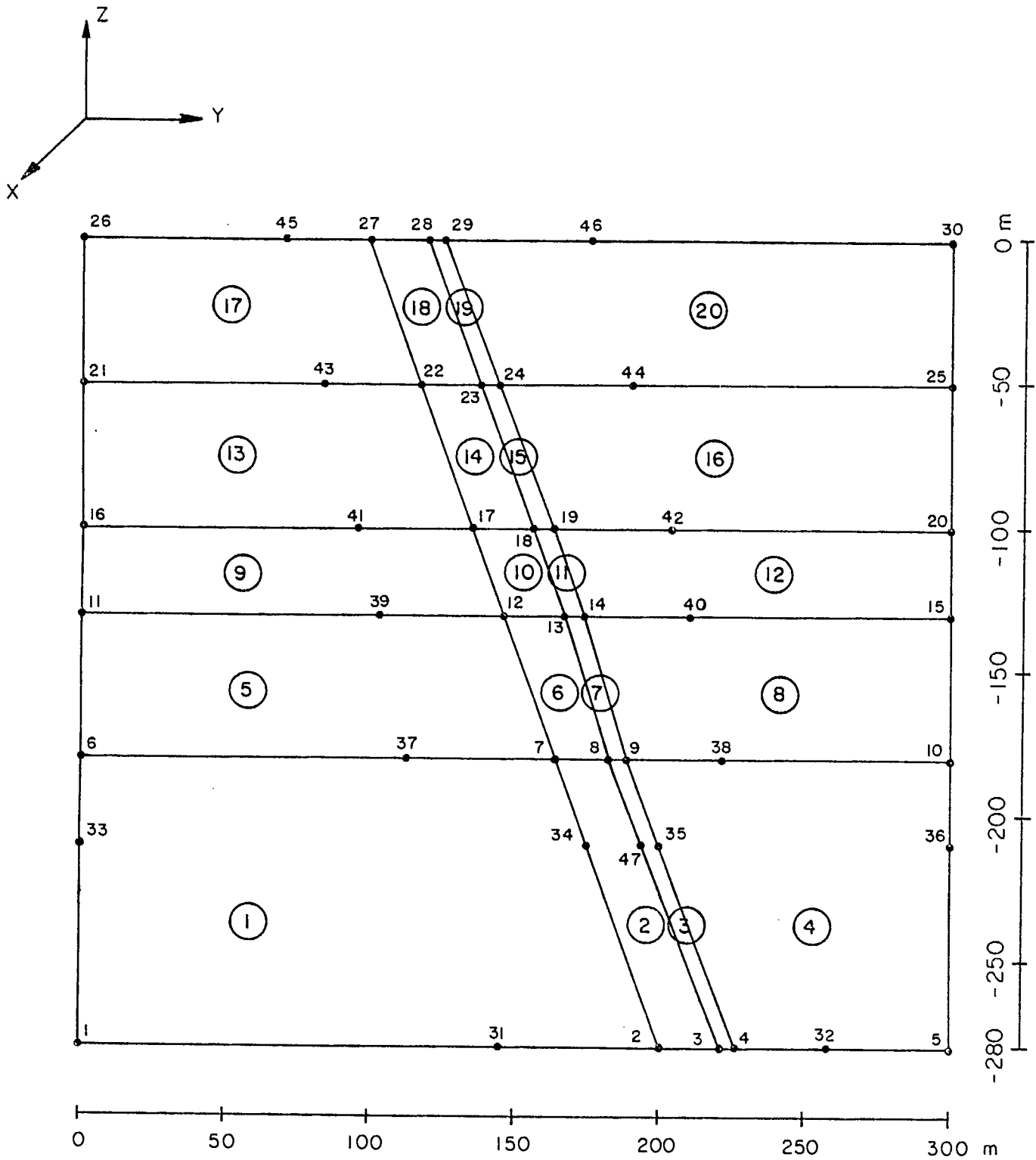


Figure 7 A zone diagram (numbers refer to nodes and circled numbers refer to zones)

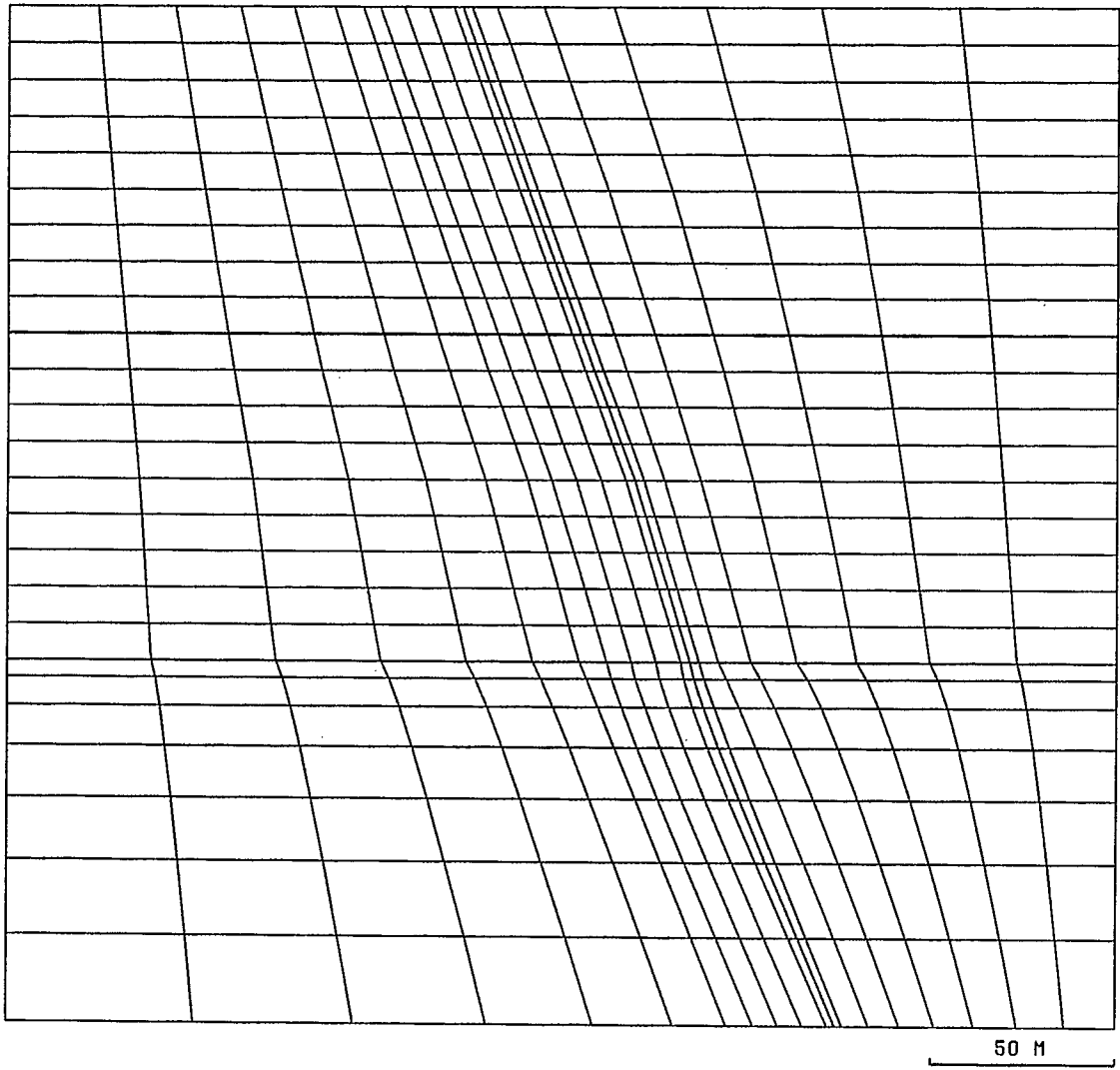


Figure 8 Finite element mesh

'Example problem - a hypothetical open stope mining under gravity loading'

| | | | | | |
|----|----|---|-------|--------|---|
| 47 | 20 | 4 | 5 | 2 | 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 | |

Figure 9 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 | 1 | 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 | 2 | 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 |

Figure 9 MSHGEN input data (continued)

Example Problem - A Hypothetical Open Stope Mining Layout
0,1,0,0,0,1.
1,0,0
1, 4.E+05, .25, 0.029
2, 3.E+05, .25, 0.029
3, 2.E+05, .25, 0.029
4, 4.E+05, .25, 0.029
0.,0.0097
0.,0.0290
0.,0.0097
0.,0.

Figure 10 SAP2DC input data

(Note that number of material types is transferred from MSHGEN program)

Under gravitational loading the initial shearing stress $\sigma_{yz} = 0$, therefore, the coefficients $a_{yz} = b_{yz} = 0$. The vertical stress σ_{zz} is due to gravity only and the horizontal stresses, σ_{yy} and σ_{xx} , 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:

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

At the top of the model, i.e., at the ground surface, $Z = 0$.

We have: $\sigma_{zz} = a_{zz} + b_{zz} \times Z = 0$, therefore, $a_{zz} = 0.0$.

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

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

Also, we have:

$$a_{zz} + b_{zz} \times (-280.0) = -0.029 \times 280.0$$

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

From the following two equations:

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

we obtain:

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

Summarizing, we have:

$$\begin{aligned}
 a_{yy} &= 0.0 \\
 b_{yy} &= 0.0097 \\
 a_{zz} &= 0.0 \\
 b_{zz} &= 0.029 \\
 a_{xx} &= 0.0 \\
 b_{xx} &= 0.0097 \\
 a_{yz} &= 0.0 \\
 b_{yz} &= 0.0
 \end{aligned}$$

These coefficients are shown in Fig. 10.

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_{zz} is due to gravity only. The horizontal stresses, σ_{yy} and σ_{xx} , are consisting of two components, one of which is the tectonic stress uniformly distributed across the depth, say 3 MPa in y-direction and 2 MPa in x-direction, and the other part is due to the Poisson's effect, i.e., $\frac{\nu}{1-\nu}\sigma_{zz}$. Also we assume that the vertical stress is one of the principal stresses. Then the initial shearing stress $\sigma_{yz} = 0$, and therefore, the coefficients $a_{yz} = b_{yz} = 0$.

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:

$$\begin{aligned}
 \sigma_{zz} &= \gamma \times Z \\
 \sigma_{yy} &= -3.0 + \frac{\nu}{1-\nu}\gamma \times Z \\
 \sigma_{xx} &= -2.0 + \frac{\nu}{1-\nu}\gamma \times Z
 \end{aligned}$$

Similarly, at the top of the model, i.e., at the ground surface, $Z = 0$.

We have: $\sigma_{zz} = a_{zz} + b_{zz} \times Z = 0$, therefore, $a_{zz} = 0.0$.

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

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

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

From the following two equations:

$$a_{yy} + b_{yy} \times Z = -3.0 + \frac{0.25}{1 - 0.25}(0.029 \times Z)$$
$$a_{xx} + b_{xx} \times Z = -2.0 + \frac{0.25}{1 - 0.25}(0.029 \times Z)$$

we obtain:

$$a_{yy} = -3.0$$
$$b_{yy} = 0.3333$$
$$a_{xx} = -2.0$$
$$b_{xx} = 0.3333$$

Summarizing, we have:

$$a_{yy} = -3.0$$
$$b_{yy} = 0.3333$$
$$a_{zz} = 0.0$$
$$b_{zz} = 0.029$$
$$a_{xx} = -2.0$$
$$b_{xx} = 0.3333$$
$$a_{yz} = 0.0$$
$$b_{yz} = 0.0$$

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 Figure 11. The input for MSHGEN and SAP2DC are shown, respectively, in Figures. 12 and 13.

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.

The finite element results and closed-form solution are given as follows:

| Finite Element Mesh | NX | NY | Displacements (m) | |
|----------------------|----|----|-------------------|-----------|
| 1 | 5 | 2 | 0.000094 | 0.000068* |
| 2 | 10 | 2 | 0.000096 | 0.000072* |
| 3 | 20 | 4 | 0.000097 | 0.000093* |
| Closed-form Solution | | | 0.000100 | |

Note: * indicates solution with incompatible displacement mode suppressed.
See Type of Elements section for more details.

The output file from SAP2DC or the input file for PCSAP2D, 'sapin.dat', for the mesh No.2 is given in Figure 14.

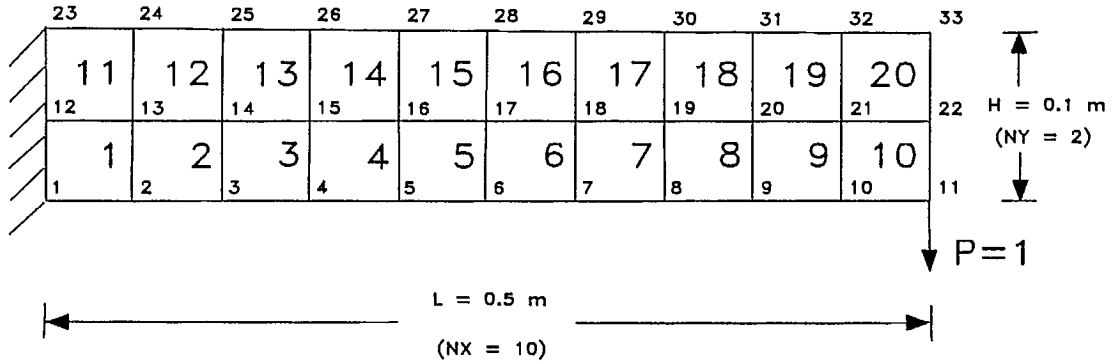


Figure 11 A cantilever beam subjected to a concentrated load

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

Figure 12 MSHGEN input data for cantilever beam example

```
Cantilever beam with concentrated load
0,0,0,1,0,1.
0,0,0
1, 5.E+06, .25, 0.0
11 0.0 -1.0
```

Figure 13 SAP2DC input data for cantilever beam example

Cantilever beam with concentrated load

| | | | | | | | | | | | |
|-----------------------|-----------------------------------|--------------|---------------|----------------------------|--------------|---|---|---|-------|-----|----|
| 33 | 1 | 1 | 1 | 1 | 1 | 1 | 0 | 0 | 1.000 | | |
| 1 | 1 | 1 | 0.000000E+00 | 0.000000E+00 | 1 | | | | | | |
| 2 | 0 | 0 | 0.4999999E-01 | 0.000000E+00 | 1 | | | | | | |
| 3 | 0 | 0 | 0.9999999E-01 | 0.000000E+00 | 1 | | | | | | |
| 4 | 0 | 0 | 0.1500000E+00 | 0.000000E+00 | 1 | | | | | | |
| 5 | 0 | 0 | 0.2000000E+00 | 0.000000E+00 | 1 | | | | | | |
| 6 | 0 | 0 | 0.2500000E+00 | 0.000000E+00 | 1 | | | | | | |
| 7 | 0 | 0 | 0.3000000E+00 | 0.000000E+00 | 1 | | | | | | |
| 8 | 0 | 0 | 0.3500000E+00 | 0.000000E+00 | 1 | | | | | | |
| 9 | 0 | 0 | 0.4000000E+00 | 0.000000E+00 | 1 | | | | | | |
| 10 | 0 | 0 | 0.4500000E+00 | 0.000000E+00 | 1 | | | | | | |
| 11 | 0 | 0 | 0.5000000E+00 | 0.000000E+00 | 1 | | | | | | |
| 12 | 1 | 1 | 0.000000E+00 | 0.500000E-01 | 1 | | | | | | |
| 13 | 0 | 0 | 0.4999999E-01 | 0.500000E-01 | 1 | | | | | | |
| 14 | 0 | 0 | 0.9999999E-01 | 0.500000E-01 | 1 | | | | | | |
| 15 | 0 | 0 | 0.1500000E+00 | 0.500000E-01 | 1 | | | | | | |
| 16 | 0 | 0 | 0.2000000E+00 | 0.500000E-01 | 1 | | | | | | |
| 17 | 0 | 0 | 0.2500000E+00 | 0.500000E-01 | 1 | | | | | | |
| 18 | 0 | 0 | 0.3000000E+00 | 0.500000E-01 | 1 | | | | | | |
| 19 | 0 | 0 | 0.3500000E+00 | 0.500000E-01 | 1 | | | | | | |
| 20 | 0 | 0 | 0.4000000E+00 | 0.500000E-01 | 1 | | | | | | |
| 21 | 0 | 0 | 0.4500000E+00 | 0.500000E-01 | 1 | | | | | | |
| 22 | 0 | 0 | 0.5000000E+00 | 0.500000E-01 | 1 | | | | | | |
| 23 | 1 | 1 | 0.000000E+00 | 0.100000E+00 | 1 | | | | | | |
| 24 | 0 | 0 | 0.4999999E-01 | 0.9999999E-01 | 1 | | | | | | |
| 25 | 0 | 0 | 0.9999999E-01 | 0.100000E+00 | 1 | | | | | | |
| 26 | 0 | 0 | 0.1500000E+00 | 0.100000E+00 | 1 | | | | | | |
| 27 | 0 | 0 | 0.2000000E+00 | 0.100000E+00 | 1 | | | | | | |
| 28 | 0 | 0 | 0.2500000E+00 | 0.100000E+00 | 1 | | | | | | |
| 29 | 0 | 0 | 0.3000000E+00 | 0.9999999E-01 | 1 | | | | | | |
| 30 | 0 | 0 | 0.3500000E+00 | 0.100000E+00 | 1 | | | | | | |
| 31 | 0 | 0 | 0.4000000E+00 | 0.9999999E-01 | 1 | | | | | | |
| 32 | 0 | 0 | 0.4500000E+00 | 0.9999999E-01 | 1 | | | | | | |
| 33 | 0 | 0 | 0.5000000E+00 | 0.100000E+00 | 1 | | | | | | |
| 4 | 20 | 1 | 1 | 0 | | | | | | | |
| 1 | 00.0000E+00 | 0.0 | | | | | | | | | |
| | 0.00.5000E+070.5000E+070.5000E+07 | 0.2500 | 0.2500 | 0.25000.2000E+07 | | | | | | | |
| | 0.0 | 0.0 | 0.0 | | | | | | | | |
| | 0.000000E+00 | 0.000000E+00 | 0.000000E+00 | 0.000000E+00 | 0.000000E+00 | | | | | | |
| | 0.000000E+00 | 0.000000E+00 | 0.000000E+00 | (part of the above record) | | | | | | | |
| 1 | 1 | 2 | 13 | 12 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 2 | 2 | 3 | 14 | 13 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 3 | 3 | 4 | 15 | 14 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 4 | 4 | 5 | 16 | 15 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 5 | 5 | 6 | 17 | 16 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 6 | 6 | 7 | 18 | 17 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 7 | 7 | 8 | 19 | 18 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 8 | 8 | 9 | 20 | 19 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 9 | 9 | 10 | 21 | 20 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 10 | 10 | 11 | 22 | 21 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 11 | 12 | 13 | 24 | 23 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 12 | 13 | 14 | 25 | 24 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 13 | 14 | 15 | 26 | 25 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 14 | 15 | 16 | 27 | 26 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 15 | 16 | 17 | 28 | 27 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 16 | 17 | 18 | 29 | 28 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 17 | 18 | 19 | 30 | 29 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 18 | 19 | 20 | 31 | 30 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 19 | 20 | 21 | 32 | 31 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 20 | 21 | 22 | 33 | 32 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0. |
| 11 | 1 | 0.0000 | -1.0000 | | | | | | | | |
| (A blank line record) | | | | | | | | | | | |
| | 0.0 | 1.0 | 1.0 | 0.0 | -1. | | | | | | |

Figure 14 PCSAP2D input data for the cantilever example - 'sapiin.dat'

Example with Boundary Elements

An example was devised to illustrate PCSAP2D input. The structure with its finite element discretization is shown in Figure 15. Note that since roller constraints had to be imposed in directions other than the global directions, it was necessary to make use of boundary elements. The input required for this example is shown in Figure 16.

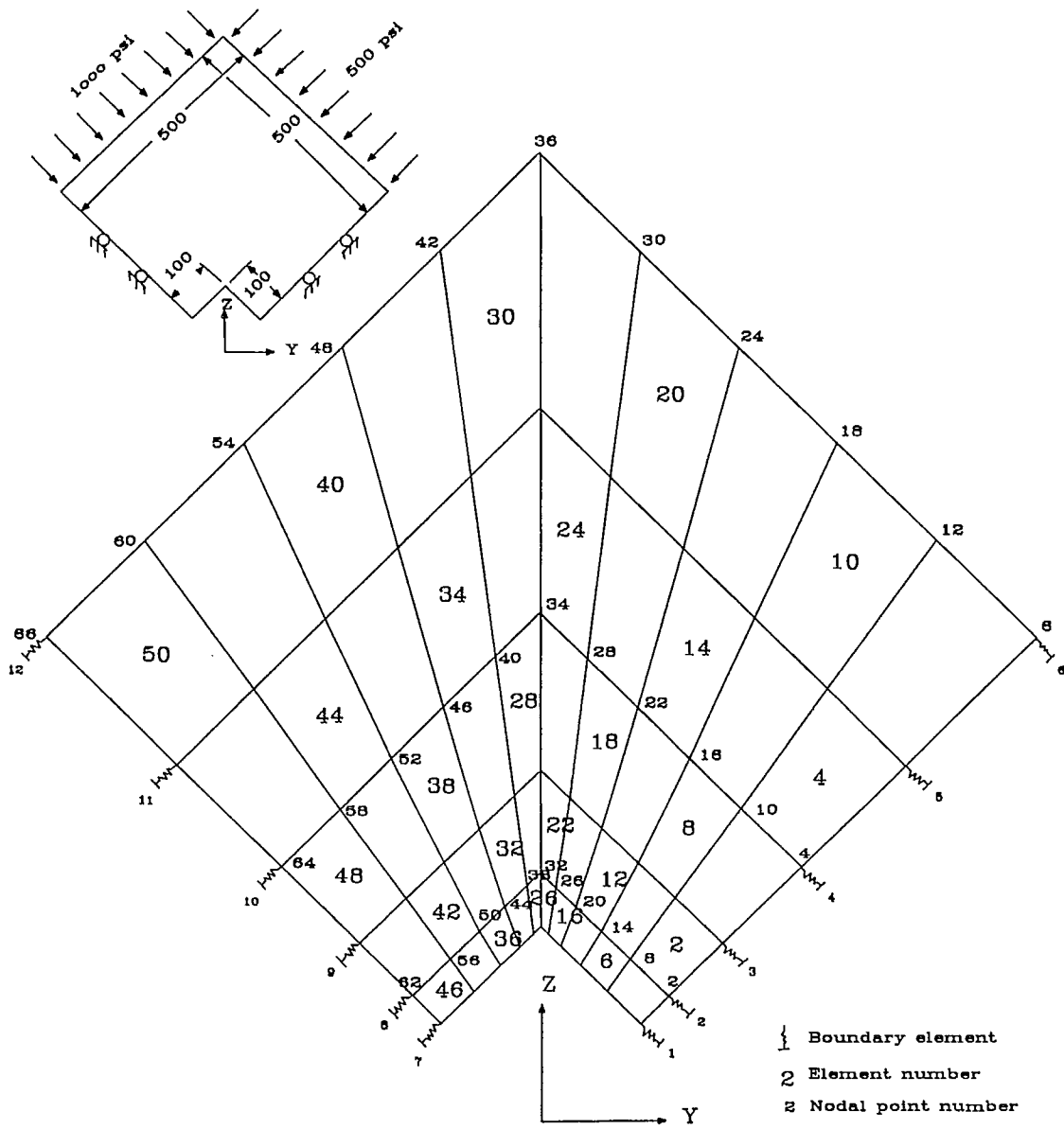


Figure 15. PCSAP2D example with boundary elements.

| EXAMPLE - PCSAP2D WITH BOUNDARY ELEMENTS | | | | | | | | |
|--|---|---|---|-----------|---|----------|---|------|
| 66 | 2 | 1 | 1 | 1 | 1 | 0 | 0 | 12.0 |
| 1 | 0 | 0 | | 70.7110 | | 70.7110 | 0 | |
| 2 | 0 | 0 | | 91.0750 | | 91.0750 | 0 | |
| 3 | 0 | 0 | | 129.5420 | | 129.5420 | 0 | |
| 4 | 0 | 0 | | 186.1100 | | 186.1100 | 0 | |
| 5 | 0 | 0 | | 260.7800 | | 260.7800 | 0 | |
| 6 | 0 | 0 | | 353.5530 | | 353.5530 | 0 | |
| 7 | 0 | 0 | | 47.5180 | | 93.9040 | 0 | |
| 8 | 0 | 0 | | 65.6190 | | 116.5300 | 0 | |
| 9 | 0 | 0 | | 98.2030 | | 160.8800 | 0 | |
| 10 | 0 | 0 | | 145.2670 | | 226.9540 | 0 | |
| 11 | 0 | 0 | | 206.8140 | | 314.7530 | 0 | |
| 12 | 0 | 0 | | 282.8420 | | 424.2760 | 0 | |
| 13 | 0 | 0 | | 28.8500 | | 112.5710 | 0 | |
| 14 | 0 | 0 | | 43.7840 | | 138.3640 | 0 | |
| 15 | 0 | 0 | | 69.5790 | | 189.5020 | 0 | |
| 16 | 0 | 0 | | 106.2350 | | 265.9880 | 0 | |
| 17 | 0 | 0 | | 153.7530 | | 367.8200 | 0 | |
| 18 | 0 | 0 | | 212.1320 | | 494.9990 | 0 | |
| 19 | 0 | 0 | | 14.7080 | | 126.7130 | 0 | |
| 20 | 0 | 0 | | 25.5690 | | 156.5770 | 0 | |
| 21 | 0 | 0 | | 43.6710 | | 215.4100 | 0 | |
| 22 | 0 | 0 | | 69.0130 | | 303.2120 | 0 | |
| 23 | 0 | 0 | | 101.5970 | | 419.9820 | 0 | |
| 24 | 0 | 0 | | 141.4210 | | 565.7220 | 0 | |
| 25 | 0 | 0 | | 5.0910 | | 136.3300 | 0 | |
| 26 | 0 | 0 | | 10.9740 | | 171.1700 | 0 | |
| 27 | 0 | 0 | | 20.4780 | | 238.6020 | 0 | |
| 28 | 0 | 0 | | 33.6020 | | 338.6250 | 0 | |
| 29 | 0 | 0 | | 50.3460 | | 471.2390 | 0 | |
| 30 | 0 | 0 | | 70.7110 | | 636.4450 | 0 | |
| 31 | 0 | 0 | | 0.0000 | | 141.4210 | 0 | |
| 32 | 0 | 0 | | 0.0000 | | 182.1430 | 0 | |
| 33 | 0 | 0 | | 0.0000 | | 259.0790 | 0 | |
| 34 | 0 | 0 | | 0.0000 | | 372.2280 | 0 | |
| 35 | 0 | 0 | | 0.0000 | | 521.5910 | 0 | |
| 36 | 0 | 0 | | 0.0000 | | 707.1680 | 0 | |
| 37 | 0 | 0 | | -5.0910 | | 136.3300 | 0 | |
| 38 | 0 | 0 | | -10.9740 | | 171.1700 | 0 | |
| 39 | 0 | 0 | | -20.4780 | | 238.6020 | 0 | |
| 40 | 0 | 0 | | -33.6020 | | 338.6250 | 0 | |
| 41 | 0 | 0 | | -50.3460 | | 471.2390 | 0 | |
| 42 | 0 | 0 | | -70.7110 | | 636.4450 | 0 | |
| 43 | 0 | 0 | | -14.7080 | | 126.7130 | 0 | |
| 44 | 0 | 0 | | -25.5690 | | 156.5770 | 0 | |
| 45 | 0 | 0 | | -43.6710 | | 215.4100 | 0 | |
| 46 | 0 | 0 | | -69.0130 | | 303.2120 | 0 | |
| 47 | 0 | 0 | | -101.5970 | | 419.9820 | 0 | |
| 48 | 0 | 0 | | -141.4210 | | 565.7220 | 0 | |
| 49 | 0 | 0 | | -28.8500 | | 112.5710 | 0 | |
| 50 | 0 | 0 | | -43.7840 | | 138.3640 | 0 | |

Figure 16 Input data for PCSAP2D example (Figure 15) with boundary elements.

- Figure 16 continued -

| | | | | | |
|----|----------|-----------|-----------|-----------|-----------|
| 51 | 0 | 0 | -69.5790 | 189.5020 | 0 |
| 52 | 0 | 0 | -106.2350 | 265.9880 | 0 |
| 53 | 0 | 0 | -153.7530 | 367.8200 | 0 |
| 54 | 0 | 0 | -212.1320 | 494.9990 | 0 |
| 55 | 0 | 0 | -47.5180 | 93.9040 | 0 |
| 56 | 0 | 0 | -65.6190 | 116.5300 | 0 |
| 57 | 0 | 0 | -98.2030 | 160.8800 | 0 |
| 58 | 0 | 0 | -145.2670 | 226.9540 | 0 |
| 59 | 0 | 0 | -206.8140 | 314.7530 | 0 |
| 60 | 0 | 0 | -282.8420 | 424.2760 | 0 |
| 61 | 0 | 0 | -70.7110 | 70.7110 | 0 |
| 62 | 0 | 0 | -91.0750 | 91.0750 | 0 |
| 63 | 0 | 0 | -129.5420 | 129.5420 | 0 |
| 64 | 0 | 0 | -186.1100 | 186.1100 | 0 |
| 65 | 0 | 0 | -260.7800 | 260.7800 | 0 |
| 66 | 0 | 0 | -353.5530 | 353.5530 | 0 |
| 4 | 50 | 1 | 1 | 1 | 0 |
| 1 | 0 | 0.0 | 0.0 | | |
| | 0.0 | .1000E+07 | .1000E+07 | .1000E+07 | .2000 |
| | 0.0 | 0.0 | 0.0 | | .2000 |
| | | | | | .2000 |
| | | | | | .4176E+06 |
| 1 | 1 | 2 | 8 | 7 | 1 |
| 2 | 2 | 3 | 9 | 8 | 1 |
| 3 | 3 | 4 | 10 | 9 | 1 |
| 4 | 4 | 5 | 11 | 10 | 1 |
| 5 | 5 | 6 | 12 | 11 | 1 |
| 2 | 500.0000 | 500.0000 | | | |
| 6 | 7 | 8 | 14 | 13 | 1 |
| 7 | 8 | 9 | 15 | 14 | 1 |
| 8 | 9 | 10 | 16 | 15 | 1 |
| 9 | 10 | 11 | 17 | 16 | 1 |
| 10 | 11 | 12 | 18 | 17 | 1 |
| 2 | 500.0000 | 500.0000 | | | |
| 11 | 13 | 14 | 20 | 19 | 1 |
| 12 | 14 | 15 | 21 | 20 | 1 |
| 13 | 15 | 16 | 22 | 21 | 1 |
| 14 | 16 | 17 | 23 | 22 | 1 |
| 15 | 17 | 18 | 24 | 23 | 1 |
| 2 | 500.0000 | 500.0000 | | | |
| 16 | 19 | 20 | 26 | 25 | 1 |
| 17 | 20 | 21 | 27 | 26 | 1 |
| 18 | 21 | 22 | 28 | 27 | 1 |
| 19 | 22 | 23 | 29 | 28 | 1 |
| 20 | 23 | 24 | 30 | 29 | 1 |
| 2 | 500.0000 | 500.0000 | | | |
| 21 | 25 | 26 | 32 | 31 | 1 |
| 22 | 26 | 27 | 33 | 32 | 1 |
| 23 | 27 | 28 | 34 | 33 | 1 |
| 24 | 28 | 29 | 35 | 34 | 1 |
| 25 | 29 | 30 | 36 | 35 | 1 |
| 2 | 500.0000 | 500.0000 | | | |

- Figure 16 continued -

| | | | | | | | | | | | |
|-----------------------|-----------|-----------|----|----|-----|---|-----|---|-----|-----|------|
| 26 | 31 | 32 | 38 | 37 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 27 | 32 | 33 | 39 | 38 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 28 | 33 | 34 | 40 | 39 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 29 | 34 | 35 | 41 | 40 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 30 | 35 | 36 | 42 | 41 | 1 | 2 | 1 | 4 | 1 | 0.0 | 0.0 |
| 2 | 1000.0000 | 1000.0000 | | | | | | | | | |
| 31 | 37 | 38 | 44 | 43 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 32 | 38 | 39 | 45 | 44 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 33 | 39 | 40 | 46 | 45 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 34 | 40 | 41 | 47 | 46 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 35 | 41 | 42 | 48 | 47 | 1 | 2 | 1 | 4 | 1 | 0.0 | 0.0 |
| 2 | 1000.0000 | 1000.0000 | | | | | | | | | |
| 36 | 43 | 44 | 50 | 49 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 37 | 44 | 45 | 51 | 50 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 38 | 45 | 46 | 52 | 51 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 39 | 46 | 47 | 53 | 52 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 40 | 47 | 48 | 54 | 53 | 1 | 2 | 1 | 4 | 1 | 0.0 | 0.0 |
| 2 | 1000.0000 | 1000.0000 | | | | | | | | | |
| 41 | 49 | 50 | 56 | 55 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 42 | 50 | 51 | 57 | 56 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 43 | 51 | 52 | 58 | 57 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 44 | 52 | 53 | 59 | 58 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 45 | 53 | 54 | 60 | 59 | 1 | 2 | 1 | 4 | 1 | 0.0 | 0.0 |
| 2 | 1000.0000 | 1000.0000 | | | | | | | | | |
| 46 | 55 | 56 | 62 | 61 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 47 | 56 | 57 | 63 | 62 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 48 | 57 | 58 | 64 | 63 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 49 | 58 | 59 | 65 | 64 | 1 | 2 | 0 | 4 | 1 | 0.0 | 0.0 |
| 50 | 59 | 60 | 66 | 65 | 1 | 2 | 1 | 4 | 1 | 0.0 | 0.0 |
| 2 | 1000.0000 | 1000.0000 | | | | | | | | | |
| 7 | 12 | | | | | | | | | | |
| 1 | 1 | 7 | | | | | | | | | |
| 7 | 61 | 55 | 1 | | | | | | | | |
| 12 | 66 | | | | | | | | | | |
| (A blank line record) | | | | | | | | | | | |
| | 0.0 | 1.0 | | | 1.0 | | 1.0 | | 0.0 | | -1.0 |

GETTING STARTED

As mentioned earlier, PCSAP2D software package consists of the following programs:

- (a) PCSAP2D
- (b) MSHGEN
- (c) SAP2DC
- (c) MSHPLT
- (e) PCPLOT

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

Hardware Requirements:

To run PCSAP2D software package efficiently, an IBM PC/AT compatible is required. The minimum desirable configuration of the system is described in the following table.

Table 16

| 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 sided floppy diskette drive and a twenty (20) or thirty (30) megabyte hard disk drive | |
| Printer | HP PaintJet printer | |
| Plotter | | |
| Mouse | Microsoft compatible mouse | |
| Remarks: | <ol style="list-style-type: none"> 1. At present the pre- and post processors require a color printer to produce hard copy. 2. To output finite element results a 132 column printer is desirable. | |

Software Requirements:

The main program PCSAP2D and the companion programs are compiled and linked with Ryan-McFarland Fortran compiler. 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, Oregon 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 PCSAP2D Software Package onto Your Personal Computer:

The PCSAP2D 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 simply create a subdirectory named PCSAP2D 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 PCSAP2D
- (b) [path]> restore a: [path]:\pcsap2d\ *.*

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 PCSAP2D 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 PCSAP2D two input files, which are relatively simple, have to be created via screen editing sessions. The two input files are 'genin.dat' and 'sap2dcin.dat', respectively, for the programs MSHGEN and SAP2DC.

MSHGEN takes 'genin.dat' and generates an output file called 'genout.dat'. The interface program SAP2DC takes 'genout.dat' and merges with the input data 'sap2dcin.dat' and produces an input file called 'sapin.dat' for the finite element program PCSAP2D.

Before you execute PCSAP2D, 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 SAP2DC. The interface program SAP2DC will produce an input file, sapin.dat, for PCSAP2D.

After execution of PCSAP2D, two additional files are created, namely: the save file - sapout.dat and the printer-output file - sap2dprt.dat.

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

- (a) genin.dat
- (b) genout.dat
- (c) sap2dcin.dat
- (d) sapin.dat
- (e) sapout.dat

In addition, there are three printer-output files, genprt.dat, sap2dcpirt.dat and sap2dprt.dat, are also created. They contain a information concerning your input data and error messages,if any, 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 SAP2DC takes the input files 'sap2dcin.dat', merges with 'genout.dat' and creates a file called 'sapin.dat'. 'sapin.dat is the input file required by PCSAP2D. After the execution of PCSAP2D, it produces a 'save' file called sap2dout.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. Toews, N.A. and Yu, Y.S.; SAP2D documentation - 2D Linear elastic finite element computer System; Report 74/138; Mining Research Laboratories, CANMET, Energy Mines and Resources Canada; Ottawa; December 1974.
2. Wilson, E.L.; SOLID SAP - A static analysis program for three dimensional solid structures; UC SESM 71-79 University of California, Berkeley, U.S.A.; (1972).
3. Toews, N.A., Yu, Y.S. and Wong, A.S.; SAP2D Documentation (VAX-11/750 Version) - 2D Linear Elastic Finite Element Program; Division Report MRP/MRL 85-35(TR); Mining Research Laboratories, CANMET, Energy Mines and Resources Canada; Ottawa; August 1985.
4. Toews, N.A., Yu, Y.S. and Wong, A.S.; MSHGEN documentation (VAX-11/750 version) - A mesh generating program for finite element analysis; Division Report MRP/MRL 85-32 (TR); Mining Research Laboratories, CANMET, Energy Mines and Resources Canada; Ottawa; February 1985.
5. Yu, Y.S., Toews, N.A. and Wong, A.S.; MSHPLT documentation (VAX-11/750 version) - A mesh plotting program for finite element analysis; Division Report MRP/MRL 85-33 (TR); Mining Research Laboratories, CANMET, Energy Mines and Resources Canada; Ottawa; February 1985.
6. Yu, Y.S., Toews, N.A. and Wong, A.S.; SAP2DC documentation (VAX-11/750 version) - An interface program for the finite element program SAP2D; Division Report MRP/MRL 85-34 (TR); Mining Research Laboratories, CANMET, Energy Mines and Resources Canada; Ottawa; February 1985.
7. PCMSHPLT User's Manual (in preparation)
8. PCPLOT User's Manual (in preparation)

Appendix A. - Nodal Point Generation

If the nodal structure is sufficiently regular it may be possible to generate certain nodal points using the simple SAP2D nodal point generator. This generator works as follows. Suppose that two consecutive records are read the first with nodal number NOLD and with co-ordinates Y(NOLD) and Z(NOLD) and the second record with nodal number N, co-ordinates Y(N) and Z(N) and generation parameter KN. If KN is less than or equal to zero no generation occurs. If KN is positive then NUM nodes will be generated (if NUM is positive) where NUM is the largest integer such that $NOLD + NUM \times KN$ is less than N. The generated points lie equally spaced on the line joining $[X(NOLD), Y(NOLD)]$ and $[X(N), Y(N)]$ and are given nodal numbers $NOLD + KN, NOLD + 2 \times KN, \dots, NOLD + NUM \times KN$ respectively. The generated nodal points have unconstrained displacements unless one or both of the constraint indicators at NOLD is negative, in which case the corresponding generated displacement constraint will also be negative.

Appendix B. - Angle of Anisotropy

In isotropic case :

$$\begin{aligned}EN &= ET = E; \\RNUNS &= RNUNT = RNUST = RNU; \\GNS &= E / 2 (1 + RNU); \text{ and} \\HAN &= HAS = HAT = HA.\end{aligned}$$

In the general case orthotropic temperature dependent material properties are allowed. The records read in above define the properties versus temperature table. Assuming that nodal point temperatures have been entered the element temperature is found by averaging the temperature over the element associated nodal points. The appropriate mechanical properties of this element are then calculated by doing a linear interpolation over the defined mechanical properties-temperature tables. The graph of mechanical properties versus temperature is assumed to be constant at temperatures lower than the lowest tabulated temperatures and temperatures higher than the highest tabulated temperatures respectively. Figure 17 defines the principal material axis N and S of the anisotropic material. The direction T is out of the paper.

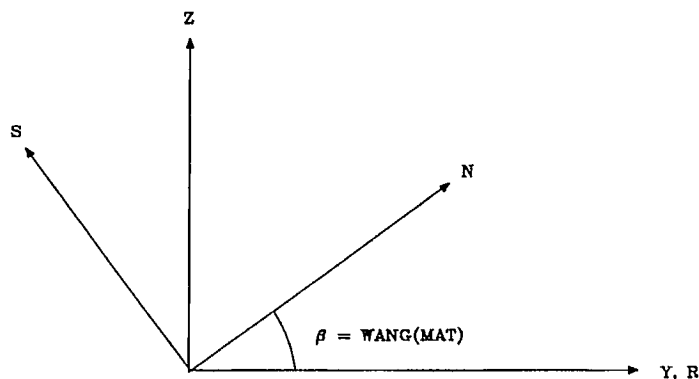


Figure 17. Principal material axis N-S.

Appendix C. - Quadrilateral Element Generation

Suppose that in the previous record read contained NOLD, IO, JO, KO, LO, MATO, NUMPCO, NSO, KGO, TO, THICKO and that current record contains NNEW, IN, JN, KN, LN, MATN, NUMPCN, NSN, KGN, TN, THICKN then if NNEW is greater than NOLD + 1 then all information for element NOLD + 1, ..., NNEW - 1 is generated as follows. Consider element NOLD + M = N then the record generated for this is N, I, J, K, L, MATO, NCUTO, NUMPCO, NSO, KG, THICKO where

$$\begin{aligned} I &= IOLD + M \times KGN \text{ (Note : KGN not KGO is used for generation)} \\ J &= JOLD + M \times KGN \\ K &= KOLD + M \times KGN \\ L &= LOLD + M \times KGN \end{aligned}$$

KG is set to zero to denote a generated element. Distributed loads (if any) are assumed identical to that of previous record.

Appendix D. – Boundary Element Generation

(a) In many problems a given displacement constraint direction remains fixed for a number of elements. To avoid having to introduce (possibly artificial) direction nodes for all these elements the following generation feature has been built in. If the direction node is zero then the direction defined by the last previous record with a non-zero entry is used.

(b) Omitted elements are also generated. In this case the location nodes of generated elements are found by incrementing the location node of the previous record by increasing multiples of the element generation parameter of the current record. Element direction, cut number, specified displacement and axial stiffness are all as on the previous record.

