

144
4-19-77

D. 914

UIIU-ENG-76-2028

CIVIL ENGINEERING STUDIES

STRUCTURAL RESEARCH SERIES NO. 428



ORNL/SUB/4164-2
DIST. CATEGORY UC-77

ISA INTERACTIVE STRESS ANALYSIS A PROGRAM FOR PLANE STRESS, PLANE STRAIN AND AXISYMMETRIC ANALYSIS OF STRUCTURAL CONTINUA USER'S MANUAL

By
H. O. ABDURAHMAN

A Report on an Investigation Carried Out as Part of the Prestressed Concrete Reactor Vessel Program of the Oak Ridge National Laboratory Operated by Union Carbide Corporation for the Energy Research and Development Administration

MASTER

UNIVERSITY OF ILLINOIS
at URBANA-CHAMPAIGN
URBANA, ILLINOIS
JULY, 1976

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

ISA

INTERACTIVE STRESS ANALYSIS
A PROGRAM FOR PLANE STRESS, PLANE STRAIN AND
AXISYMMETRIC ANALYSIS OF STRUCTURAL CONTINUA

USER'S MANUAL

by

H. O. Abdulrahman

NOTICE
This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Energy Research and Development Administration, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

A Report on an investigation carried out as part of the prestressed concrete reactor vessel program of the Oak Ridge National Laboratory operated by Union Carbide Corporation for the Energy Research and Development Administration

UNIVERSITY OF ILLINOIS
URBANA, ILLINOIS
July, 1976

MASTER

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

X/1

Printed in the United States of America. Available from
National Technical Information Service
U. S. Department of Commerce
5285 Port Royal Road, Springfield, Virginia 22161
Price: Printed Copy ~~\$6.00~~, Microfiche \$2.25
4. →

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the Energy Research and Development Administration, nor any of their employees, nor any of their contractors, sub-contractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

ACKNOWLEDGEMENT

This program was developed as part of the investigation of the structural behaviour of prestressed concrete reactor vessels sponsored by the nuclear regulatory commission. The project constitutes a portion of the study "Prestressed Concrete Pressure Vessel Development" carried out at the Oak Ridge National Laboratory under the direction of Dr. J. P. Callahan.

The study is part of the doctoral dissertation by H. O. Abdulrahman under the direction of Professor W. C. Schnobrich.

The program was developed on the DEC System-10 computer furnished by the digital computer laboratory of the University of Illinois.

TABLE OF CONTENTS

	Page
Chapter 1. INTRODUCTION	1
1.1 The Program	1
1.2 Data and Program Structure	2
1.3 The Finite-Element Library	3
1.4 Solution Procedure	4
1.5 Program Capacity	5
Chapter 2. INPUT.	6
2.1 Input Description Syntax	6
2.2 Preparation of the Input	7
2.3 Input Procedure	8
Chapter 3. EXAMPLE PROBLEMS	20
List of Figures	
1. Element of an Axisymmetric Solid	26
2. Local and Global Coordinate Systems	26
3. Natural Coordinates of an Element	26
4. Overlay (Tree) Structure of ISA	27
5. Example Problem (1): Pressure Vessel	28
6. Example Problem (2): Cantilever Beam	29
Appendix A. PROGRAM CAPACITY	31

Chapter 1
INTRODUCTION

1.1 The Program

This program, which is applicable to linear problems, has been developed as the foundation of a nonlinear finite-element program for the analysis of two- and three- dimensional structures. It was developed as part of an investigation of prestressed concrete reactor vessels. Hence it is oriented towards the analysis of reinforced and prestressed concrete structures. The operational part of the program can be used to obtain elastic stress and strain distributions in a two-dimensional continuum; namely plane stress, plane strain and axisymmetric stress conditions.

The goal was to develop a handy analytical tool with a relatively simple user-oriented input and fast turnaround. This is usually accomplished by employing a free-format language translator. This option, however, was not used here because it would have entailed an additional overhead of fast core. The alternative was to use a different concept that serves the same objective while meeting the limitations of core usage set by the Digital Computer Laboratory (DCL) at the University of Illinois. This concept involved designing the program to "talk" to the user. It obtains the data and instructions it needs by asking a series of questions which describe to the user the data he is expected to provide or the decisions he has to make. This is made more expedient by giving the user a chance to recover and correct the most likely mistakes during every phase of the input.

The program was designed for operation interactively on a DEC-10 Computer. It is coded in FORTRAN IV Language. The DEC-10 Fortran compiler permits a quasi-free-format input. This is very advantageous for input from

a teletype terminal. Being interactive, the program enables the user to monitor the processing of his problem closely and make the appropriate decisions. Operation of the program on other computers depends on the availability of interactive (timesharing) facilities.

1.2 Data and Program Structure

Finite-element programs, and particularly nonlinear ones, involve handling and processing of a tremendous amount of data. The memory space required to store the data becomes even larger when extended precision arithmetic is used to minimize truncation errors, as was the case with this program.

With the maximum fast core on the DEC10 set by the DCL to 20-30 Kilowords (80~120 Kilobytes), it would have been impossible to run the program without incorporating the following features:

- (a) All unneeded data are stored in auxiliary devices (disk). This enabled the allocation of the same memory space for different global arrays. Also most of the local variables and arrays share the same "common" memory space.
- (b) The setting up and solution of a finite-element problem involves a series of discrete independent steps. Each step may be carried out by a module which performs a certain function on the data. This modular design eliminates the need of maintaining all the modules in core simultaneously, resulting in an "overlaid" program. The overlay capability of DEC10 was utilized in this program to reduce the amount of fast core needed to run it to less than 25 Kilowords. All the modules reside on a disk and are dynamically brought to core and maintained there, while they are needed, by the system. Future extensions of the program are not expected to require additional core. Since most of the modules are replaced only once, it was observed that overlaying the program did not increase the loading or execution

time appreciably.

The overlay structure of the program is shown in Fig. (4).

1.3 The Finite-Element Library

A large selection of elements for various types of structures have been developed since the inception of the finite-element method. The element employed in this program to date is a four-node linear quadrilateral isoparametric element. Included is the option to employ nonconforming modes if desired. The element has two degrees of freedom per node. The nonconforming modes can be introduced in either or both the x and y directions of the element. In that case they add two or four additional degrees of freedom to the element respectively. These additional degrees of freedom are condensed out of the element stiffness before assembling the structure stiffness matrix so that the final element stiffness matrix remains an 8 x 8.

The element stiffness is evaluated numerically using Gaussian quadrature with two integration points in each direction. This element is used to model two-dimensional continuum stress conditions: axisymmetric, plain stress and plain strain. The element strains and stresses are evaluated at the four integration points and the center.

The nonconforming modes allow the element modeling to more readily respond to sharp strain gradients. However the user is cautioned that for element grids containing some highly nonrectangular elements the use of nonconforming modes can result in a breakdown in the convergence characteristics of the element solution. These nonconforming modes must be used with caution. Nonconforming modes should not be used with triangular elements. However, the program automatically shuts off this option when the triangular elements are specified by coalescing the last two nodes incident on the element.

1.4 Solution Procedure

The procedure used in this program to obtain a computer solution can be summarized in three steps:

(a) Input

Input data are fed conversationally to the program. The program types out sufficient information explaining the nature of the data which the user is expected to type in. This built-in user's manual enables the acquainted user to run the program without reference to this manual.

Whenever a choice has to be selected among several alternatives, and the user does not make that choice, the program will automatically select one of them. This "default" value is usually indicated in the preceding explanatory note typed out by the program, and is always the most frequently expected value. All the default values are invoked by simply pressing the carriage return (CR) key and proceeding to the next step.

The user is permitted considerable flexibility in recovering input errors. The input phase is divided into a sequence of steps. After each step the user is given the chance of making total or partial corrections to his data.

Since the nodal coordinates and element incidences comprise the larger part of the input, and since most mistakes occur in this step, the program is capable of reading this data from a stored file. This file can be saved for future use or modification. The details of preparing this file will be explained later.

(b) Solution

After a successful input, the program proceeds to:

- i - form the element stiffness matrices;
- ii - assemble the structure stiffness matrix;

iii - Solve the resulting system of simultaneous equations.

The theory underlying this step is well known and can be found in most texts on the finite-element method.

(c) Output

The results of the analysis comprise the following:

i - Nodal displacements and loads.

ii - Element strains and stresses.

The user can selectively print any or all of the nodal displacements, the element strains, stresses, principal strains and principal stresses.

The output will be printed on the line printer attached to the computing facility. However, the user can have it written in a disk file in his storage area. This becomes imperative for long-distance terminals. A hard copy of this file can then be produced at the user's choice.

In addition, the user is given the choice of having the nodal displacements typed out on his teletype. This is sometimes useful because the structure displacements give an indication of most of the errors that might occur in setting up the problem or feeding in the data. The faulty run can be aborted by the user before the remaining computations are made.

1.5 Program Capacity

The capacity of the program is currently set to 150 nodes. There is essentially no limit placed on the number of elements other than that controlled by the number of nodes.

The capacity can, however, be altered by a few changes in the main program as explained in Appendix (A).

Chapter 2

INPUT

2.1 Input Description Syntax

The following notational conventions are used to describe the input commands.

(a) Data and information produced by the program on the teletype are demarcated by printing in italics.

This output produced by the program during execution is one of the following:

- i - information to the user about the nature and order of the data awaited by the program;
- ii - printout of data requested by the user;
- iii - or a message indicating the status of the execution.

(b) Input data arguments are surrounded by angle brackets < >.

Braces { } are used to enclose a set of alternatives from which the user must make a choice. The alternatives are either separated by vertical bars {Y|N}, or stacked on top of each other $\begin{Bmatrix} Y \\ N \end{Bmatrix}$.

(c) Whenever the program expects some data to be typed in, a statement defining these data is produced on the teletype. The data entered should correspond in order to the variables indicated in the statement. Floating-point numbers may be entered in integer or decimal form. Numerical data may be entered anywhere along the line separated by commas or blanks. The decimal point need not be used in entering floating-point data unless there is a fractional part. The machine will convert them to the appropriate form.

Pressing the CR key terminates the input record. If fewer arguments than those expected by the program are entered, these will be assigned to the leftmost variables. The trailing variables will be set to

zero. If nothing at all is entered, the entire set will be assigned zeroes.

Questions asked by the program expect a yes or no answer (Y or N). The program compares the user's answer with Y, so that CR is equivalent to N. This helps to speed up the input process.

2.2 Preparation of the Input

Before running the program, a finite element model of the structure to be analyzed should be prepared. Judgement has to be exercised to produce a reasonably representative model. This model should show the following:

- i - The conditions on the boundary of the structure. Boundary nodes should be either:
 - a - free to displace in both X and Y directions
 - b - fixed in one direction (roller support or line of symmetry)
 - c - fixed in both directions (hinge support).

Other boundary conditions can be included by appropriate structural modelling.

- ii - Geometry of nodal coordinates relative to a right-handed cartesian coordinate system.
- iii - The node numbering scheme.

This should be planned to minimize the bandwidth of the structure stiffness matrix by minimizing the difference between the numbers of the nodes incident on the elements throughout the structure.

- iv - The element numbering scheme.

This can be in any convenient order. When more than one set of elements are used, elements belonging to each set must continue in sequence counting from the last element in the previous set.

An important consideration in (ii), (iii), and (iv) above is the ordering of the mesh to facilitate easy generation of nodal geometry and

structural topology. Since this reduces the amount of input greatly, it is sometimes given preference over bandwidth minimization. A compromise has to be struck between user time and machine time.

(v) Element and nodal loads

Nodal loads are concentrated loads at the node points in the X or Y directions. Positive values are taken in the positive axis direction. An element load is a uniform load normal to one side of an element. A positive element load is a pressure on that side of the element.

2.3 Input Procedure

After logging into the DECIO system and starting the program (see Appendix), the input is conducted through the following conversation:

(1)* *W E L C O M E*

Enter: TOTAL # OF JOINTS, TOTAL # OF ELEMENTS

<tot. # of jnts> <tot. # of elements>

(2) *Do you have JOINT & ELEMENT data in a file?(Y or N)*

{<Y>|<N>}

If the answer to this question is Y the program transfers to (3), otherwise it will proceed directly to (4).

(3) *File: <filename>*

where filename is the 1 to 5 character name of the file that contains the nodal coordinates and element incidences. The file must be present in the user's disk area and have an extension DAT, for example PROB1.DAT. If any mistake occurs in (1) through (3), the user can terminate execution [control-C twice] and start all over again.

(4) *<>Enter #s of ALL joints fixed in X-direction.*

<n₁> <n₂> <n₃> ... <n_k>

*The parenthesized numbers are used to explain the input process and are not produced by the program.

where n_i are the numbers of the joints restrained against displacement in the global X-direction.

The program expects a maximum of 30 such nodes.

(5) <> *Enter #s of ALL joints fixed in Y-direction.*

< m_1 > < m_2 > < m_3 > ...

Same as (4) for nodes restrained in Y-direction.

(6) ? *Mistakes in this step ?*
Want to MAKE CORRECTIONS ?(Y or N)

If the answer is:

a - Y, the user will be taken back to (4).

b - N or CR, program will proceed to (7) if the answer to (2)

was N, otherwise it will proceed to (8).

(7) <> *For all joints (n) Type:*
NUMBER, X-COORD, Y-COORD, INCREMENT to node numbers (NG)
Missing joints will be generated
Generation factor NG with last joint in a series

<jt#> <x-coord.> <y-coord.> [<ng>]

<jt#> <x-coord.> <y-coord.> [<ng>]

⋮

<last jt#> <x-coord.> <y-coord.> [<ng>]

<n> is the total number of joints entered in (1). Every line should contain the sequence number of the joint, its abscissa and ordinate in a right-handed coordinate system, and the increment NG if the generation option is used. These numbers can be separated by commas or blanks.

The decimal point need not be quoted in the coordinates if they do not contain fractions. When the generation capability is used nodal data is entered for the first and for the last node of the sequence. The generation parameter is included on the last node of each sequence.

Nodal data may be entered in any sequence. The only restriction is that the last node number must be entered at the end since that signals the end of the input. The program generates any sequence of skipped equally spaced nodes on a straight line. Let the first and last nodes in the series be numbered n and m respectively and the increment be NG . Then $\left(\frac{m-n}{NG} - 1\right)$ nodes will be generated between node n and node m with the numbers:

$$n + NG, n + 2 \times NG, \dots, m - NG.$$

Their coordinates will be

$$x_n + \Delta x, x_n + 2\Delta x, \dots$$

$$y_n + \Delta y, y_n + 2\Delta y, \dots$$

$$\text{where } \Delta x = \frac{x_m - x_n}{s}$$

$$\Delta y = \frac{y_m - y_n}{s}$$

$$\text{and } s = \frac{m-n}{NG}$$

Clearly $(m-n)$ has to be an exact multiple of NG .

Had the user opted to write these data in a file they should be written exactly as explained above. In that case the program will skip this step and go directly to (9).

(8) *Want joint data typed? (Y or N)*

Y will cause typeout on the terminal of the following

<i>JOINT #</i>	<i>X-COORD</i>	<i>Y-COORD</i>	<i>X-TRANS</i>	<i>Y-TRANS</i>
<i>1</i>	<i>x₁</i>	<i>y₁</i>	<i>0</i>	<i>1</i>
<i>2</i>	<i>x₂</i>	<i>y₂</i>	<i>0</i>	<i>0</i>
<i>3</i>	<i>x₃</i>	<i>y₃</i>	<i>0</i>	<i>0</i>
<i>4</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>
<i>⋮</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>
<i>⋮</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>	<i>⋮</i>
<i>Last joint</i>	<i>x_n</i>	<i>y_n</i>		

This helps the user to check his input especially when he generates intermediate nodes. A '0' in the X-TRANS or Y-TRANS indicates that the node is free to translate in that direction a 1 indicates a restraint. This should check with input in (4) and (5).

If the user feels that this output will be lengthy and would like instead to look at specific nodes he should type N or CR and go to (9), otherwise the program will take him to (10).

- (9) *To have specific joint-data typed enter joint number
Push CR to end*

When the user types in a joint number, the response is

<n> <x_n> <y_n> <I₁> <I₂>

For example, typing 5 will produce:

5 10.00 15.00 0 1

CR terminates this step and transfers to the next one.

- (10) *? Mistakes in this step?
Want to MAKE CORRECTIONS? (Y or N)*

This allows recovery of any mistake in the nodal coordinates whether they are entered conversationally or read from a file. In both cases corrections have to be typed in.

{_N^Y} will cause transfer to {₍₁₂₎⁽¹¹⁾}

- (11) *For joint data CORRECTIONS enter:
JOINT #, X-coord., Y-coord., NG (if generation is needed)
To FINISH enter data for LAST JOINT*

<jt.#> <x-coord.> <y-coord.> [<NG>]
:
:
<last jt.#>

Data need be entered only for those nodes which were wrongly entered initially. Generation as in (7) is allowed here. This step is concluded by reentering data for the last node. When the last node is entered, program transfers to step (8) for another look at joint data if desired.

(12) *?Want equilibrium equation #s typed? (Y or N)*

Y will cause the following typeout:

EQUILIBRIUM NODE #	EQUATION NUMBERS	
	X	Y
1	n_1	n_2
2	n_3	n_4
3		
4		
⋮		
⋮		
<last jt>		n_k

Where n_1, n_2, \dots are sequence numbers assigned to the degrees of freedom at each node. Restrained degrees of freedom are indicated by zero in the appropriate column.

N or CR will produce the following message and cause transfer to (13).

Joint data input was successful.

(13) *Do you have concentrated nodal loads? (Y or N)*

{Y} will cause transfer to {(14)}
{N} will cause transfer to {(17)}

(14) *Input fixed and proportional joint loads
Start with fixed loads .. push CR to end*

Type: JOINT #, LOAD, DIRECTION (1 for X, 2 for Y)

<jt>	<load>	<directn>
⋮	⋮	⋮
⋮	⋮	⋮

Only those nodes which have applied loads need to be entered.

Proportional loads

Type: JOINT #, LOAD, DIRECTION (1 for X, 2 for Y)

<jt>	<load>	<directn>
⋮	⋮	⋮
⋮	⋮	⋮

Concentrated loads have been classified this way for use in the nonlinear analysis. For an elastic analysis the user may enter his

loads as though they belonged to either category. Proportional loads are printed out with nodal displacements. The equivalent nodal loads due to element side pressures are added to the proportional loads vector.

(15) *Want joint loads typed? (Y or N)*

N will cause transfer to (16).

Y will cause the following typeout for the nodes which have loads applied at them:

FIXED LOADS:

<i>JOINT #</i>	<i>X-DIRECN</i>	<i>Y-DIRECN</i>
<i><n_j></i>	<i><F_x></i>	<i><F_y></i>
<i>⋮</i>	<i>⋮</i>	<i>⋮</i>

PROPORTIONAL LOADS:

<i>JOINT #</i>	<i>X-DIRECN</i>	<i>Y-DIRECN</i>
<i><n_j></i>	<i><P_x></i>	<i><P_y></i>
<i>⋮</i>	<i>⋮</i>	<i>⋮</i>

(16) *?MISTAKES IN THIS STEP?
WANT TO START IT OVER AGAIN? (Y or N)*

{Y} will cause transfer back to (14)
{N} results in the next message and transfers to (17)}

Joint load data input successful

(17) *DATA-INPUT FOR 2-D, FOUR-NODE QUADRILATERAL ISOPARAMETRIC ELEMENT:
How many element sets are in your structure? (max.=5,default=1)*

<number of sets>

The program permits entry of the elements in up to 5 sets. Each set may have different material properties and element characteristics (nonconforming modes). Number of sets is set to 1 by default (CR).

(18) *Enter the stress condition code:*
0 : Axisymmetric (default),
1 : Plane strain,
2 : Plane stress.

For the axisymmetric stress condition the program looks at a slice of a circular cylinder subtending an angle of 1 radian at the axis of symmetry, Fig. (1). The element forms the projection of this slice on any radial plane.

For the plane strain condition the program looks at an element of unit thickness normal to the XY plane.

For the plane stress condition the thickness of the element is specified with the input.

- (19) For element set # *<i>* Enter:
Number of ELEMENTS in the SET

<# of elements in set i>

<i> ranges from 1 to the number entered in (17). The numbers entered for each set should add up to the total number of elements declared in step (1).

- (20) Does this set have ORTHOTROPIC MATERIAL PROPERTIES? (Y OR N)
(Default is ISOTROPIC)

{_N^Y} transfer to {₍₂₂₎⁽²¹⁾}

- (21) ENTER THE 7 ELASTIC MODULII & ANGLE OF ORTHOTROPY
E_x, E_y, E_z, NU_{xy}, NU_{xz}, NU_{yz}, G, BETA

<E_x> <E_y> <E_z> <v_{xy}> <v_{xz}> <v_{yz}> <G>

Where E_x, E_y, E_z: are the elastic modulii in the x, y, and z directions respectively, in the local coordinate system xyz.

v_{xy}, v_{xz}, v_{yz}: are the Poisson's ratios in the xy, xz, and yz planes respectively.

G: is the shear modulus

B: is the angle between the local element xyz and the global XYZ taken as the counter clockwise angle from the Global X to local element x. (Fig. (2))

NOTE: A zero value for either E_x, E_y or E_z may cause a numerical error.

(22) Enter *YOUNG'S MODULUS, POISSON'S RATIO*

<E> <v>

This step is skipped if the answer to (20) is Y.

(23) Enter *NONCONFORMING MODES* parameter:
 1 : introduce in X-direction only
 2 : introduce in both directions
 0 : suppress in both directions

<parm.>

Here X and Y are the natural coordinates of the element, Fig. (3). To introduce the nonconforming modes in the Y-direction only, the incident nodes can be rotated counterclockwise.

(24) Enter element data in element sequence.
 Data for skipped elements will be generated.
 Missing elements will have same *PRESSURE & THICKNESS*
 as the *LAST* element in the generation series.
IELG is the node# increment for generation
 taken from the last element in the series.
 Enter:

ELEMENT #, nodes I,J,K,L, IELG, PRESSURE on I-J, THICKNESS of element.

```
<l> <I1> <J1> <K1> <L1> <IELG1> <pressure1> <thickness1>
  ⋮      ⋮      ⋮
  ⋮      ⋮      ⋮
```

<list element>

The phrase "THICKNESS of element" is typed out for the plane stress condition only.

The elements have to be entered in order from 1 to the highest element number. I, J, K, and L are the numbers of the nodes incident on an element counterclockwise from any corner. Triangular elements may be entered by assigning the same node number to K and L. The thickness need not be entered except for the plane stress condition.

The program generates missing elements in sequence between any two consecutive lines of input if they do not refer to consecutive elements. Thus if two lines read

<n> <I_n> <J_n> <K_n> <L_n> <IELG> <P_n> <thick_n>

<m> <I_m> <J_m> <K_m> <L_m> <IELG> <P_m> <thick_m>

and (m-n)>1, then (m-n-1) elements will be generated in this way:

<n+1> <I_n+IELG_m> <J_n+IELG_m> <K_n+IELG_m> <L_n+IELG_m> <P_m> <thick_m>

<n+2> <I_n+2IELG_m> <J_n+2IELG_m> <K_n+2IELG_m> <L_n+2IELG_m> <P_m> <thick_m>

⋮

<m-1>

The pressure is taken to be an applied uniform load on the side defined by the first two nodes incident on the element (I-J).

Note that the pressure on the sides of intermediate elements, (and the thickness in the plane stress case) are set to the values of the pressure and thickness of the last element. If the element data reside in a file together with the nodal data (i.e., if the answer to step (2) is Y) the file should contain the above information for all element sets in order following nodal data. The program will read it and skip this step.

(25) *Want data for this set typed? (Y or N)*

{N} will cause transfer to (26)
{Y} will result in this typeout:

ELEMENT SET #	:	i						
Ex	=	<Ex>	Ey	=	<Ey>	Ez	=	<Ez>
NU _{xy}	=	<v _{xy} >	NU _{xz}	=	<v _{xz} >	NU _{yz}	=	<v _{yz} >
G	=	<G>	BETA	=	<β>			
Non-conforming Modes Parameter			=	<parm>				

ELEMENT #	INCIDENT NODES				PRESSURE
	I	J	K	L	ON I-J
1	<I ₁ >	<J ₁ >	<K ₁ >	<L ₁ >	< 1 >
⋮					
<last element in set>					

(26) *? Mistakes in this step? (Y or N)*

{ Y } will cause transfer to (27)
{ N } will cause transfer to (29)

(27) *Type:*

- 1 :to start all over again
- 2 :to make specific corrections

<1> will cause return back to (18). All the data entered for the set will be lost and have to be entered again.

<2> Transfer to (28)

(28) *For CORRECTIONS : type*

Element #, I, J, K, L, PRESSURE on I-J, THICKNESS (for plain stress only)
No generation is allowed here
Push CR to finish

The input here is the same as in step (24).

(29) If there are more than one set of elements, and they have not all been processed, programs transfer back to (18) for element set $i+1$, otherwise it proceeds to (30).

(30) *Element data input successful*

FORMING ELEMENT STIFFNESS MATRICES IN SEQUENCE

1, 2, ..., n. The program types out the numbers of the elements as it forms their stiffnesses.

ELEMENT STIFFNESSES FORMED & STORED

FORMING STRUCTURE STIFFNESS MATRIX

TOTAL NUMBER OF EQUATIONS = <n₁>
BANDWIDTH = <n₂>
NO. OF EQUATIONS PER BLOCK = <n₃>
NUMBER OF BLOCKS = <n₄>

GLOBAL STIFFNESS MATRIX ASSEMBLED & STORED

FORMING UPPER TRIANGULAR STIFFNESS MATRIX

UPPER TRIANGULAR MATRIX FORMED & STORED

BACKSUBSTITUTION & REDUCTION OF LOAD VECTOR

SYSTEM OF EQUATIONS SOLVED

This keeps the user informed about the progress of the solution process.

(31) *Enter the 5-digit printout parameter (nnnnn)*

The 5 digits refer respectively to :

Displacements, Strains, Stresses, Principal strains, Principal stresses

- n .EQ. 0 :suppress printing quantity,*
- n .NE. 0 :include it in the printout*

The five-digit parameter is used for selective printout of the five types of output. For example if the user is interested in displacements, stresses and principal stresses only he can enter a number like 10101 or 70905.

The details of the output are shown in the sample problem in Chapter (3).

- (32) *Want above quantities written on a file? (Y or N)*
(Default is printing directly on line printer)

<Y> will cause whatever output requested in (31) to be written on a disk file by the program. This is sometimes convenient. It becomes imperative for the user who doesn't have easy access to the line printers. The output is saved in this way and the user can have it typed at his convenience.

<N> will route the output to the line printer specified by the user.

- (33) *Results will be written on file FINAL.DAT*

This message is produced by the program if the answer to (32) is Y.

- (34) *Type in the title of your problem (60 characters)*

This title will be printed on the heading of the output and with the stress-strain results for every element as shown in the sample problem.

- (35) *Want joint displacements typed on your Teletype? (Y or N)*

This is in addition to having them printed as requested in (31).

{N} will cause transfer to (36).
 {Y} will result in the following output:

NODE	DISPLACEMENTS		PROPORTIONAL LOADS	
	X	Y	X	Y
1	δ_{x1}	δ_{y1}	P_{x1}	P_{y1}
2	δ_{x2}	δ_{y2}	P_{x2}	P_{y2}
3				
4				
⋮	⋮			
⋮	⋮			
⋮	⋮			
<last jt>		.	.	.

δ_{xi} , δ_{yi} are the X and Y displacements of joint i respectively

P_{xi} , P_{yi} are the input proportional nodal forces and/or the nodal forces equivalent to the element pressures.

(36) *COMPUTING STRESSES AND STRAINS*
 STRESSES & STRAINS COMPUTED & STORED
 PRINTING STRESSES & STRAINS ON LINE PRINTER
 STRESS PRINTOUT COMPLETED
BYE
STOP

END OF EXECUTION
CPU TIME: <t> ELAPSED TIME: <m:s>
EXIT

Chapter 3

EXAMPLE PROBLEMS

In this chapter two example problems are presented to demonstrate the details involved in the application of the program. The problems selected have been scaled down and simplified to keep repetitive data to a minimum yet illustrate the main features of the program. The two problems are described below. For each problem the following pages include a listing of:

1. the permanent data file* containing the nodal coordinates and element incidences,
2. the conversation through which the data are fed interactively to the program,
3. part of the output produced by the program, namely the displacements, strains, stresses and their principal values.

Example (1) Thick-walled pressure vessel:

A thick-walled cylindrical concrete vessel, 12.5 in. internal diameter and 20 in. external diameter closed by 10 in. thick slabs is analyzed. The slab is penetrated by six holes as shown in Fig. 5. The vessel is symmetrical about the horizontal plane through its middle. Without the penetrations, it would be axially symmetric about its vertical axis. However, a reasonable approximation of its deformation can be obtained by modelling it as axisymmetric and adjusting the material pro-

*This file may be created by any convenient means. On the DEC System-10, the SOS editor provides the most versatile tool for preparing and editing file texts. The details of using the SOS editor may be found in the DEC System-10 manuals.

perties in the area of the holes. In this analysis the hole area is assumed to be homogeneous and isotropic but with a modulus of elasticity reduced in the ratio of the solid area to the total area in plan. Due to the mentioned symmetry only one quarter of the vessel is analyzed. Boundary conditions along the axes of the vessel are introduced to impose this symmetry.

Example (2): Reinforced Concrete Cantilever Beam.

Fig. 6.a shows the dimensions of the beam and the details of the idealized flexural reinforcement. Shear reinforcement, although necessary and assumed present, is not included in the analysis. Since the strain component perpendicular to the plane of the beam is small, the structure will be treated as a plane stress problem. With this formulation it is possible to have elements with different thicknesses. The contribution of the concrete above the level of the steel is neglected but the steel is considered to be perfectly bonded to the remaining concrete.

The finite-element idealization of the structure is shown in Fig. 6.b. The applied moment is replaced by an equivalent couple. A plot of the deformed shape of the mesh, exaggerated by amplifying the displacements, is shown in Fig. 6.c.

Explanation of the Output:

Data can be input in any system of units as long as these units are consistent for all the input variables. The output data will be in corresponding units.

Positive values of displacements and loads are displacements and loads in the positive direction of the global axes. Positive values of strains and stresses are tensile.

For an axisymmetric analysis, SIGX corresponds to the radial stress, SIGY to the vertical stress, and SIGZ to the hoop stress.

The principal strains and stresses indicated in the output are in the XY plane. The strains and stresses in the Z-direction are principal values because of the absence of shear strains and stresses in XZ and YZ planes. The principal angle indicated in the output is the angle between the direction of the first principal stress and the global X-axis measured in the counterclockwise direction.

The local and the natural coordinates of an element are defined with respect to the four incident nodes IJKL, with the x- or ξ -axis bisecting the side J-K of the element. The location of the four integration points relative to these coordinate systems is shown in Fig. 2 and 3. Note that this orientation rotates if the incident nodes are rotated.

EXAMPLE PROBLEM (1): THICK-WALLED CONCRETE PRESSURE VESSEL

Nodal and Element Data File: VESSEL.DAT

```

1 0 20
5 0 10 1
6 2.5 20
10 2.5 10 1
11 5 20
15 5 10 1
16 7.5 20
20 7.5 10 1
21 10 20
25 10 10 1
26 12.5 20
34 12.5 0 1
35 15 20
43 15 0 1
44 17.5 20
52 17.5 0 1
53 20 20
61 20 0 1
1 1 2 7 6
3 3 4 9 8 1
4 5 10 9 4 0 1000
5 6 7 12 11
7 8 9 14 13 1
8 10 15 14 9 0 1000
9 21 22 27 26
11 23 24 29 28 1
12 25 30 29 24 0 1000
13 26 27 36 35
16 29 30 39 38 1
20 33 34 43 42 1 1000
21 35 36 45 44
28 42 43 52 51 1
29 44 45 54 53
36 51 52 61 60 1
37 11 12 17 16
39 13 14 19 18 1
40 15 20 19 14 0 1000
41 16 17 22 21
43 18 19 24 23 1
44 20 25 24 19 0 1000

```

Interactive Commands Sequence

```

# E L E M E N T
Enter: TOTAL # OF JOINTS, TOTAL # OF ELEMENTS
61 44
Do you have JOINT & ELEMENT data in a file? (Y or N)
Y
File :VESSEL
<> Enter #s of ALL Joints fixed in X-direction .
1 2 3 4 5
<> Enter #s of ALL Joints fixed in Y-direction .
34 43 52 61
? Mistakes in this step ?
Want to MAKE CORRECTIONS ?(Y or N)
? Want joint data typed ? (Y or N)
To have specific Joint-data typed enter joint number
Push CR to end
? Mistakes in this step ?
Want to MAKE CORRECTIONS ?(Y or N)
? Want equilibrium equation #s typed ?(Y or N)
Joint data input was successful .
Do you have concentrated nodal loads?(Y or N)
DATA-INPUT FOR 2-D, FOUR-NODE QUADRILATERAL ISOPARAMETRIC ELEMENT :
How many element sets are in your structure ? (max.=5,default=1)
2
Enter the stress condition code :
0 : Axisymmetric ( default ) .
1 : Plain strain .
2 : Plain stress .

```

```

For element set # 1 Enter :
Number of ELEMENTS in the SET
36
Does this set have ORTHOTROPIC MATERIAL PROPERTIES ?(Y OR N)
(Default is ISOTROPIC)
Enter YOUNG'S MODULUS,POISSON'S RATIO
3500000 .2
Enter NON-CONFORMING MODES parameter :
1 : introduce in X-direction only
2 : introduce in both directions
0 : suppress in both directions
2
Want data for this set typed ?(Y or N)
? Mistakes in this step ? (Y or N)
For element set # 2 Enter :
Number of ELEMENTS in the SET
8
Does this set have ORTHOTROPIC MATERIAL PROPERTIES ?(Y OR N)
(Default is ISOTROPIC)
Enter YOUNG'S MODULUS,POISSON'S RATIO
875000
Enter NON-CONFORMING MODES parameter :
1 : introduce in X-direction only
2 : introduce in both directions
0 : suppress in both directions
2
Want data for this set typed ?(Y or N)
? Mistakes in this step ? (Y or N)
Element data input successful
FORMING ELEMENT STIFFNESS MATRICES IN SEQUENCE
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,
17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32,
33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44,
ELEMENT STIFFNESSES FORMED & STORED
FORMING STRUCTURE STIFFNESS MATRIX
TOTAL NUMBER OF EQUATIONS =113
BANDWIDTH = 21
NO. OF EQUATIONS PER BLOCK=113
NUMBER OF BLOCKS = 1
GLOBAL STIFFNESS MATRIX ASSEMBLED & STORED
FORMING UPPER TRIANGULAR STIFFNESS MATRIX
UPPER TRIANGULAR MATRIX FORMED & STORED
BACKSUBSTITUTION & REDUCTION OF LOAD VECTOR
SYSTEM OF EQUATIONS SOLVED
Enter the 5-digit printout parameter (nnnnn)
The 5 digits refer respectively to :
Displacements,Strains,Stresses,Principal strains,Principal stresses
n .EQ. 0 :suppress printing quantity ,
n .NE. 0 :include it in the printout
12345
Want above quantities written on a file? (Y or N)
(Default is printing directly on line printer)
Y
Results will be written on file FINAL.DAT
Type in the title of your problem (60 characters)
EXAMPLE PROBLEM (1): THICK-WALLED CONCRETE PRESSURE VESSEL
Want joint displacements typed on your teletype ?(Y or N)
COMPUTING STRESSES AND STRAINS
STRESSES & STRAINS COMPUTED & STORED
PRINTING STRESSES & STRAINS ON LPT/D/SA
STRESS PRINTOUT COMPLETED
BYE
STOP

```

Partial Listing of the Program Output

TITLE : EXAMPLE PROBLEM (1): THICK-WALLED CONCRETE PRESSURE VESSELS

NUMBER OF NODES = 61
 TOTAL NUMBER OF ELEMENTS = 44
 NUMBER OF SETS OF ELEMENTS = 2
 NUMBER OF DEGREES OF FREEDOM (EQUATIONS) = 113
 HALF BANDWIDTH OF STIFFNESS MATRIX = 21
 TYPE OF ANALYSIS :
 AXISYMMETRIC

NODAL DISPLACEMENTS AND LOADS :

NODE	DISPLACEMENTS		PROPORTIONAL LOADS	
	X	Y	X	Y
1	0.000E-01	7.756E-03	0.00	0.00
2	0.000E-01	7.734E-03	0.00	0.00
3	0.000E-01	8.030E-03	0.00	0.00
4	0.000E-01	8.439E-03	0.00	0.00
5	0.000E-01	9.083E-03	0.00	1041.67
6	9.815E-04	7.063E-03	0.00	0.00
7	4.883E-04	7.374E-03	0.00	0.00
8	2.463E-04	7.735E-03	0.00	0.00
9	2.135E-06	8.247E-03	0.00	0.00
10	-2.797E-04	8.712E-03	0.00	6250.00
11	1.559E-03	6.123E-03	0.00	0.00
12	8.112E-04	6.509E-03	0.00	0.00
13	5.138E-04	6.998E-03	0.00	0.00
14	1.363E-04	7.654E-03	0.00	0.00
15	-4.454E-04	8.351E-03	0.00	12500.00
16	2.227E-03	4.565E-03	0.00	0.00
17	1.286E-03	4.846E-03	0.00	0.00
18	1.008E-03	5.250E-03	0.00	0.00
19	5.768E-04	6.086E-03	0.00	0.00
20	-7.546E-04	7.177E-03	0.00	18750.00
21	2.244E-03	3.147E-03	0.00	0.00
22	1.544E-03	3.223E-03	0.00	0.00
23	1.255E-03	3.314E-03	0.00	0.00
24	1.202E-03	3.364E-03	0.00	0.00
25	-3.077E-04	4.580E-03	0.00	25000.00
26	2.053E-03	2.312E-03	0.00	0.00
27	1.521E-03	2.356E-03	0.00	0.00
28	1.268E-03	2.342E-03	0.00	0.00
29	1.072E-03	2.128E-03	0.00	0.00
30	1.030E-03	1.588E-03	15625.00	14583.33
31	2.938E-03	2.852E-04	31250.00	0.00
32	3.836E-03	-2.944E-05	31250.00	0.00
33	4.427E-03	-7.514E-05	31250.00	0.00
34	4.620E-03	0.000E-01	15625.00	0.00
35	1.845E-03	1.753E-03	0.00	0.00
36	1.446E-03	1.762E-03	0.00	0.00
37	1.160E-03	1.677E-03	0.00	0.00
38	1.016E-03	1.464E-03	0.00	0.00
39	1.293E-03	1.223E-03	0.00	0.00
40	2.187E-03	9.864E-04	0.00	0.00
41	3.196E-03	5.020E-04	0.00	0.00
42	3.774E-03	2.109E-04	0.00	0.00
43	3.982E-03	0.000E-01	0.00	0.00
44	1.695E-03	1.380E-03	0.00	0.00
45	1.357E-03	1.403E-03	0.00	0.00
46	1.079E-03	1.394E-03	0.00	0.00
47	1.015E-03	1.351E-03	0.00	0.00
48	1.323E-03	1.269E-03	0.00	0.00
49	1.978E-03	1.072E-03	0.00	0.00
50	2.753E-03	7.864E-04	0.00	0.00
51	3.342E-03	3.949E-04	0.00	0.00
52	3.546E-03	0.000E-01	0.00	0.00
53	1.623E-03	1.064E-03	0.00	0.00
54	1.310E-03	1.127E-03	0.00	0.00
55	1.058E-03	1.244E-03	0.00	0.00
56	1.018E-03	1.410E-03	0.00	0.00
57	1.290E-03	1.533E-03	0.00	0.00
58	1.855E-03	1.499E-03	0.00	0.00
59	2.523E-03	1.198E-03	0.00	0.00
60	3.043E-03	6.703E-04	0.00	0.00
61	3.242E-03	0.000E-01	0.00	0.00

ELEMENT STRAINS AND STRESSES :

***** ELEMENT SET NUMBER (1) *****
 NUMBER OF ELEMENTS IN THE SET = 36
 ELASTIC MODULI :
 EX= 4000000.0 EY= 4000000.0 EZ= 4000000.0
 NUXY= 0.2 NUXZ= 0.2 NUYZ= 0.2
 SHEAR MODULUS = 1666666.7
 ANGLE OF ORTHOTROPY = 0.00
 NONCONFORMING MODES PARAMETER = 2

STRAINS AND STRESSES :

ELEMENT

1 EXAMPLE PROBLEM (1): THICK-WALLED CONCRETE PRESSURE VESSELS

STRAINS POINT	EPSX	EPSY	EPSZ	GAMAXY
IP:1	0.4457D-03	-0.4768D-04	0.6549D-04	0.8854D-04
IP:2	0.2561D-03	-0.1246D-03	0.2745D-03	0.2178D-03
IP:3	0.3318D-03	0.9276D-05	-0.4822D-04	-0.4417D-03
IP:4	0.1422D-03	-0.6762D-04	0.1606D-03	-0.3125D-03
CENT	0.2940D-03	-0.5765D-04	0.1131D-03	-0.1120D-03

STRESSES POINT	SIGX	SIGY	SIGZ	TAUXY
IP:1	0.2001D+04	0.3563D+03	0.7342D+03	0.1476D+03
IP:2	0.1305D+04	0.3589D+02	0.1366D+04	0.3630D+03
IP:3	0.1431D+04	0.3563D+03	0.1646D+03	-0.7362D+03
IP:4	0.7354D+03	0.3589D+02	0.7966D+03	-0.5208D+03
CENT	0.1368D+04	0.1961D+03	0.7654D+03	-0.1866D+03

***** ELEMENT SET NUMBER (2) *****

NUMBER OF ELEMENTS IN THE SET = 8

ELASTIC MODULI :
 EX= 2000000.0 EY= 2000000.0 EZ= 2000000.0
 NUXY= 0.2 NUXZ= 0.2 NUYZ= 0.2
 SHEAR MODULUS = 833333.3
 ANGLE OF ORTHOTROPY = 0.00
 NONCONFORMING MODES PARAMETER = 2

STRAINS AND STRESSES :

ELEMENT

57 EXAMPLE PROBLEM (1): THICK-WALLED CONCRETE PRESSURE VESSELS

STRAINS POINT	EPSX	EPSY	EPSZ	GAMAXY
IP:1	0.2540E-03	-0.1609D-03	0.2693D-03	-0.2641E-03
IP:2	0.2479E-03	-0.1367D-03	0.2655D-03	-0.1933E-03
IP:3	0.1094E-03	-0.1302D-03	0.1649D-03	-0.4192E-03
IP:4	0.2032E-03	-0.1060D-03	0.1909D-03	-0.3485E-03
CENT	0.2286E-03	-0.1334D-03	0.2245D-03	-0.3063E-03

STRESSES POINT	SIGX	SIGY	SIGZ	TAUXY
IP:1	0.6246E+03	-0.6678D+02	0.6502D+03	-0.2201D+03
IP:2	0.6224D+03	-0.1851D+02	0.6518D+03	-0.1611D+03
IP:3	0.4967D+03	-0.6920D+02	0.4593D+03	-0.3494D+03
IP:4	0.4987D+03	-0.1659D+02	0.4782D+03	-0.2904D+03
CENT	0.5586D+03	-0.4477D+02	0.5519D+03	-0.2552D+03

PRINCIPAL STRAINS

POINT	EPS I	EPS II	ANGLE
IP:1	0.2925D-03	-0.1993D-03	-16.24
IP:2	0.2708D-03	-0.1596D-03	-13.35
IP:3	0.3093D-03	-0.2302D-03	-25.50
IP:4	0.2815D-03	-0.1843D-03	-24.21
CENT	0.2847D-03	-0.2189D-03	-29.12

PRINCIPAL STRESSES

POINT	SIG I	SIG II	ANGLE
IP:1	0.6888D+03	-0.1309D+03	-16.24
IP:2	0.6606D+03	-0.5673D+02	-13.35
IP:3	0.6633D+03	-0.2358D+03	-25.50
IP:4	0.5293D+03	-0.1472D+03	-24.21
CENT	0.6521D+03	-0.1382D+03	-20.12

EXAMPLE PROBLEM (2): R. C. CANTILEVER BEAM

Nodal and Element Data File: BEAM.DAT

```
.1 32 4.5
97 0 4.5 6
2 32 4
98 0 4 6
3 32 3
99 0 3 6
4 32 2
100 0 2 6
5 32 1
101 0 1 6
6 32
102 0 0 6
1 1 7 8 2 0 0 .5
16 91 97 98 92 6 0 .5
17 2 8 9 3 0 0 2
32 92 98 99 93 6 0 2
33 3 9 10 4 0 0 2
48 93 99 100 94 6 0 2
49 4 10 11 5 0 0 2
64 94 100 101 95 6 0 2
65 5 11 12 6 0 0 2
80 95 101 102 96 6 0 2
```

Interactive Commands Sequence

```
W E L C O M E
Enter: TOTAL # OF JOINTS,TOTAL # OF ELEMENTS
102 60
Do you have JOINT & ELEMENT data in a file? (Y or N)
Y
File IBEAM
  <> Enter #s of ALL Joints fixed in X-direction .
97 98 99 100 101 102
  <> Enter #s of ALL Joints fixed in Y-direction .
97 98 99 100 101 102
? Mistakes in this step ?
Want to MAKE CORRECTIONS ?(Y or N)
? Want joint data typed ? (Y or N)
To have specific Joint-data typed enter Joint number
Push CR to end
? Mistakes in this step ?
Want to MAKE CORRECTIONS ?(Y or N)
?Want equilibrium equation #s typed ?(Y or N)
Joint data input was successful .
Do you have concentrated nodal loads?(Y or N)
Y
Input fixed and proportional Joint loads
Start with fixed loads ., push CR to end
Type : JOINT #, LOAD ,DIRECTION (1 for X, 2 for Y)
Proportional loads
Type : JOINT #, LOAD ,DIRECTION (1 for X, 2 for Y)
1 -1000 2
2 -4000 1
6 4000 1
Want joint loads typed ?(Y or N)
? MISTAKES IN THIS STEP ?
WANT TO START IT OVER AGAIN?(Y OR N)
Joint load data input successful
DATA-INPUT FOR 2-D,FOUR-NODE QUADRILATERAL ISOPARAMETRIC ELEMENT :
How many element sets are in your structure ? (max.=5,default=1)
2
Enter the stress condition code :
0 : Axisymmetric ( default ) .
1 : Plain strain .
2 : Plain stress .
2
```

```
For element set # 1 Enter :
Number of ELEMENTS in the SET
16
Does this set have ORTHOTROPIC MATERIAL PROPERTIES ?(Y OR N)
(Default is ISOTROPIC)
Enter YOUNG'S MODULUS,POISSON'S RATIO
29000000 .3
Enter NON-CONFORMING MODES parameter :
1 : introduce in X-direction only
2 : introduce in both directions
0 : suppress in both directions
2
Want data for this set typed ?(Y or N)
? Mistakes in this step ? (Y or N)
For element set # 2 Enter :
Number of ELEMENTS in the SET
64
Does this set have ORTHOTROPIC MATERIAL PROPERTIES ?(Y OR N)
(Default is ISOTROPIC)
Enter YOUNG'S MODULUS,POISSON'S RATIO
3500000 .2
Enter NON-CONFORMING MODES parameter :
1 : introduce in X-direction only
2 : introduce in both directions
0 : suppress in both directions
2
Want data for this set typed ?(Y or N)
? Mistakes in this step ? (Y or N)
Element data input successful
FORMING ELEMENT STIFFNESS MATRICES IN SEQUENCE
1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80,
ELEMENT STIFFNESSES FORMED & STORED
FORMING STRUCTURE STIFFNESS MATRIX
TOTAL NUMBER OF EQUATIONS =192
BANDWIDTH = 16
NO. OF EQUATIONS PER BLOCK=156
NUMBER OF BLOCKS = 2
GLOBAL STIFFNESS MATRIX ASSEMBLED & STORED
FORMING UPPER TRIANGULAR STIFFNESS MATRIX
UPPER TRIANGULAR MATRIX FORMED & STORED
BACKSUBSTITUTION & REDUCTION OF LOAD VECTOR
SYSTEM OF EQUATIONS SOLVED
Enter the 5-digit printout parameter (nnnnn)
The 5 digits refer respectively to :
Displacements,Strains,Stresses,Principal strains,Principal stresses
n .EQ. 0 :suppress printing quantity .
n .NE. 0 :include it in the printout
12345
Want above quantities written on a file? (Y or N)
(Default is printing directly on line printer)
Y
Results will be written on file FINAL.DAT
Type in the title of your problem (60 characters)
REINFORCED CONCRETE CANTILEVER BEAM -----
Want joint displacements typed on your Teletype ?(Y or N)
COMPUTING STRESSES AND STRAINS
STRESSES & STRAINS COMPUTED & STORED
PRINTING STRESSES & STRAINS ON LPT/DISK
STRESS PRINTOUT COMPLETED
BYE
STOP
END OF EXECUTION
CPU TIME: 51.90 ELAPSED TIME: 9:1.78
```

END

Partial Listing of the Program Output

TITLE : EXAMPLE PROBLEM (2) : R.C. CANTILEVER BEAM <PLANE STRESS>

NUMBER OF NODES = 102
 TOTAL NUMBER OF ELEMENTS = 80
 NUMBER OF SETS OF ELEMENTS = 2
 NUMBER OF DEGREES OF FREEDOM (EQUATIONS) = 192
 HALF BANDWIDTH OF STIFFNESS MATRIX = 16

TYPE OF ANALYSIS :

PLAIN STRESS

MODAL DISPLACEMENTS AND LOADS :

NODE	DISPLACEMENTS		PROPORTIONAL LOADS	
	X	Y	X	Y
1	6.145E-04	-4.365E-02	0.00	-1000.00
2	-1.423E-04	-4.362E-02	-4000.00	0.00
3	4.395E-05	-4.338E-02	0.00	0.00
4	-2.762E-04	-4.315E-02	0.00	0.00
5	-4.782E-04	-4.285E-02	0.00	0.00
6	1.188E-03	-4.254E-02	4000.00	0.00
7	8.127E-04	-4.240E-02	0.00	0.00
8	8.159E-04	-4.248E-02	0.00	0.00
9	8.704E-05	-4.262E-02	0.00	0.00
10	-3.215E-04	-4.269E-02	0.00	0.00
11	-5.034E-04	-4.282E-02	0.00	0.00
12	-1.212E-03	-4.285E-02	0.00	0.00
13	1.761E-03	-4.119E-02	0.00	0.00
14	1.208E-03	-4.122E-02	0.00	0.00
15	4.634E-04	-4.124E-02	0.00	0.00
16	-3.488E-04	-4.128E-02	0.00	0.00
17	-1.224E-03	-4.122E-02	0.00	0.00
18	-2.080E-03	-4.112E-02	0.00	0.00
19	2.550E-03	-3.899E-02	0.00	0.00
20	1.791E-03	-3.903E-02	0.00	0.00
21	6.092E-04	-3.908E-02	0.00	0.00
22	-5.268E-04	-3.907E-02	0.00	0.00
23	-1.685E-03	-3.905E-02	0.00	0.00
24	-2.899E-03	-3.900E-02	0.00	0.00
25	2.950E-03	-3.624E-02	0.00	0.00
26	2.177E-03	-3.629E-02	0.00	0.00
27	7.611E-04	-3.631E-02	0.00	0.00
28	-6.557E-04	-3.631E-02	0.00	0.00
29	-2.081E-03	-3.628E-02	0.00	0.00
30	-3.556E-03	-3.624E-02	0.00	0.00
31	3.357E-03	-3.301E-02	0.00	0.00
32	2.514E-03	-3.304E-02	0.00	0.00
33	8.672E-04	-3.306E-02	0.00	0.00
34	-7.511E-04	-3.306E-02	0.00	0.00
35	-2.389E-03	-3.304E-02	0.00	0.00
36	-4.073E-03	-3.301E-02	0.00	0.00
37	3.671E-03	-2.942E-02	0.00	0.00
38	2.738E-03	-2.944E-02	0.00	0.00
39	9.520E-04	-2.945E-02	0.00	0.00
40	-8.218E-04	-2.945E-02	0.00	0.00
41	-2.608E-03	-2.944E-02	0.00	0.00
42	-4.444E-03	-2.942E-02	0.00	0.00
43	3.851E-03	-2.559E-02	0.00	0.00
44	2.880E-03	-2.559E-02	0.00	0.00
...				
88	-3.751E-04	-2.019E-03	0.00	0.00
89	-1.208E-03	-2.057E-03	0.00	0.00
90	-3.073E-03	-2.125E-03	0.00	0.00
91	9.288E-04	-6.828E-04	0.00	0.00
92	6.478E-04	-6.103E-04	0.00	0.00
93	2.215E-04	-5.647E-04	0.00	0.00
94	-1.920E-04	-5.628E-04	0.00	0.00
95	-6.183E-04	-6.034E-04	0.00	0.00
96	-1.135E-03	-7.041E-04	0.00	0.00
97	0.000E-01	0.000E-01	0.00	0.00
98	0.000E-01	0.000E-01	0.00	0.00
99	0.000E-01	0.000E-01	0.00	0.00
100	0.000E-01	0.000E-01	0.00	0.00
101	0.000E-01	0.000E-01	0.00	0.00
102	0.000E-01	0.000E-01	0.00	0.00

ELEMENT STRAINS AND STRESSES :

***** ELEMENT SET NUMBER (1) *****
 NUMBER OF ELEMENTS IN THE SET = 16
 ELASTIC MODULI :
 EX= 29000000.1 EY= 29000000.1 EZ= 29000000.1
 NUXY= 0.3 NUXZ= 0.3 NUYZ= 0.3
 SHEAR MODULUS = 11153846.2
 ANGLE OF ORTHOTROPY = 0.00
 NONCONFORMING NODES PARAMETER = 2

STRAINS AND STRESSES :

1 EXAMPLE PROBLEM (2) : R.C. CANTILEVER BEAM <PLANE STRESS>					
ELEMENT	STRAINS		STRESSES		
	POINT	EPSX	EPSY	EPSZ	GAMAXY
IP:1	-0.1598D-03	-0.5351D-04	0.0000D+00	0.1558D-03	
IP:2	-0.3792D-03	0.1231D-04	0.0000D+00	0.1558D-03	
IP:3	-0.1989D-03	0.7686D-04	0.0000D+00	0.1558D-03	
IP:4	-0.4183D-03	0.1427D-03	0.0000D+00	0.1558D-03	
CENT	-0.2891D-03	0.4459D-04	0.0000D+00	0.1558D-03	
IP:1	-0.5605D+04	-0.3233D+04	0.0000D+00	0.1737D+04	
IP:2	-0.1197D+05	-0.3233D+04	0.0000D+00	0.1737D+04	
IP:3	-0.5605D+04	0.5475D+03	0.0000D+00	0.1737D+04	
IP:4	-0.1197D+05	0.5475D+03	0.0000D+00	0.1737D+04	
CENT	-0.8784D+04	-0.1343D+04	0.0000D+00	0.1737D+04	
IP:1	-0.1237D-04	-0.2010D-03	62.16		
IP:2	0.2723D-04	-0.3942D-03	79.15		
IP:3	0.9734D-04	-0.2194D-03	75.27		
IP:4	0.1533D-03	-0.4290D-03	82.24		
CENT	0.6187D-04	-0.3064D-03	77.49		
IP:1	-0.2316D+04	-0.6523D+04	62.16		
IP:2	-0.2900D+04	-0.1230D+05	79.15		
IP:3	0.1004D+04	-0.6062D+04	75.27		
IP:4	0.7842D+03	-0.1220D+05	82.24		
CENT	-0.9574D+03	-0.9172D+04	77.49		
2 EXAMPLE PROBLEM (2) : R.C. CANTILEVER BEAM <PLANE STRESS>					
ELEMENT	STRAINS		STRESSES		
	POINT	EPSX	EPSY	EPSZ	GAMAXY
IP:1	-0.4226D-03	0.1636D-03	0.0000D+00	-0.6676D-04	
IP:2	-0.2622D-03	0.1155D-03	0.0000D+00	-0.6676D-04	
IP:3	-0.4079D-03	0.1145D-03	0.0000D+00	-0.6676D-04	
IP:4	-0.2474D-03	0.6636D-04	0.0000D+00	-0.6676D-04	
CENT	-0.3350D-03	0.1150D-03	0.0000D+00	-0.6676D-04	
IP:1	-0.1190D+05	0.1173D+04	0.0000D+00	-0.7447D+03	
IP:2	-0.7251D+04	0.1173D+04	0.0000D+00	-0.7447D+03	
IP:3	-0.1190D+05	-0.2510D+03	0.0000D+00	-0.7447D+03	
IP:4	-0.7251D+04	-0.2510D+03	0.0000D+00	-0.7447D+03	
CENT	-0.9577D+04	0.4610D+03	0.0000D+00	-0.7447D+03	
IP:1	0.1655D-03	-0.4245D-03	-86.75		
IP:2	0.1184D-03	-0.7651D-03	-84.99		
IP:3	0.1166D-03	-0.4100D-03	-86.36		
IP:4	0.6987D-04	-0.2510D-03	-83.99		
CENT	0.1174D-03	-0.3375D-03	-85.78		
IP:1	0.1215D+04	-0.1195D+05	-86.75		
IP:2	0.1238D+04	-0.7317D+04	-84.99		
IP:3	-0.2036D+03	-0.1195D+05	-86.36		
IP:4	-0.1726D+03	-0.7330D+04	-83.99		
CENT	0.5159D+03	-0.9832D+04	-85.78		

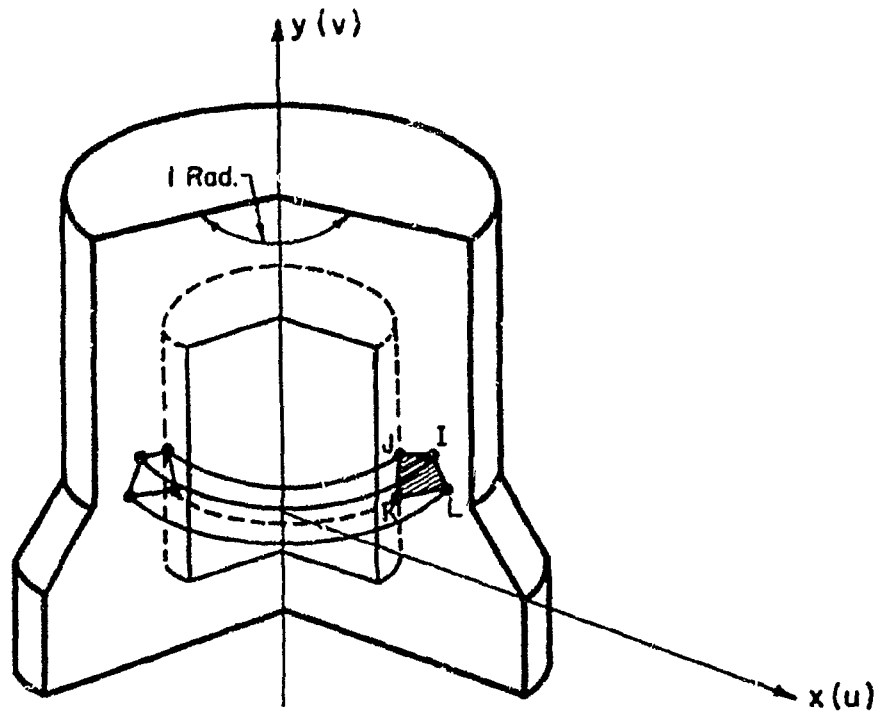


Fig. 1 Element of an Axisymmetric Solid

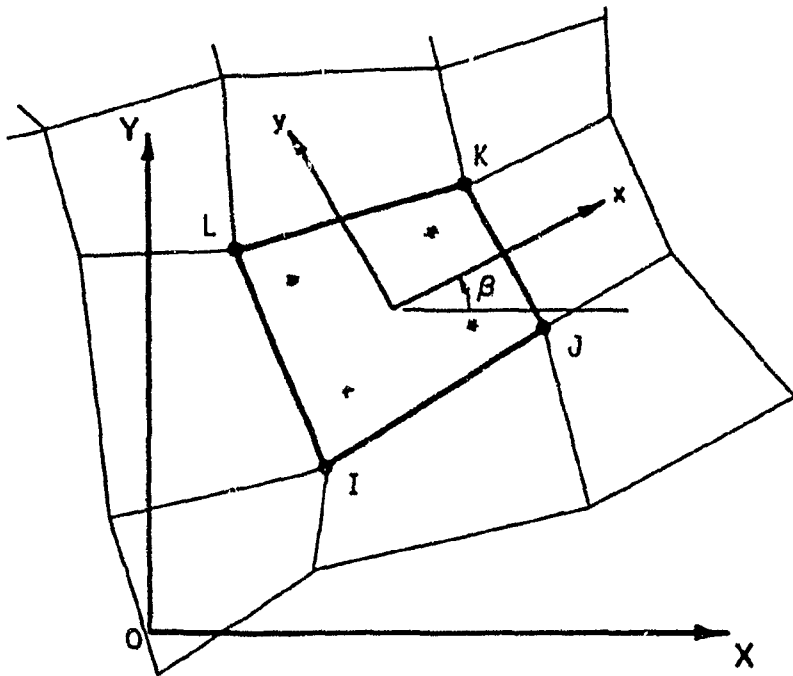


Fig. 2 Local and Global Coordinate Systems

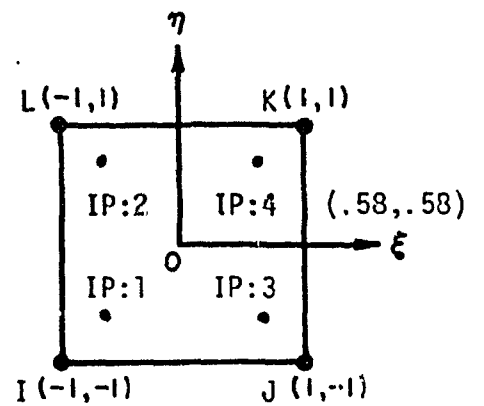


Fig. 3 Natural Coordinates of an Element

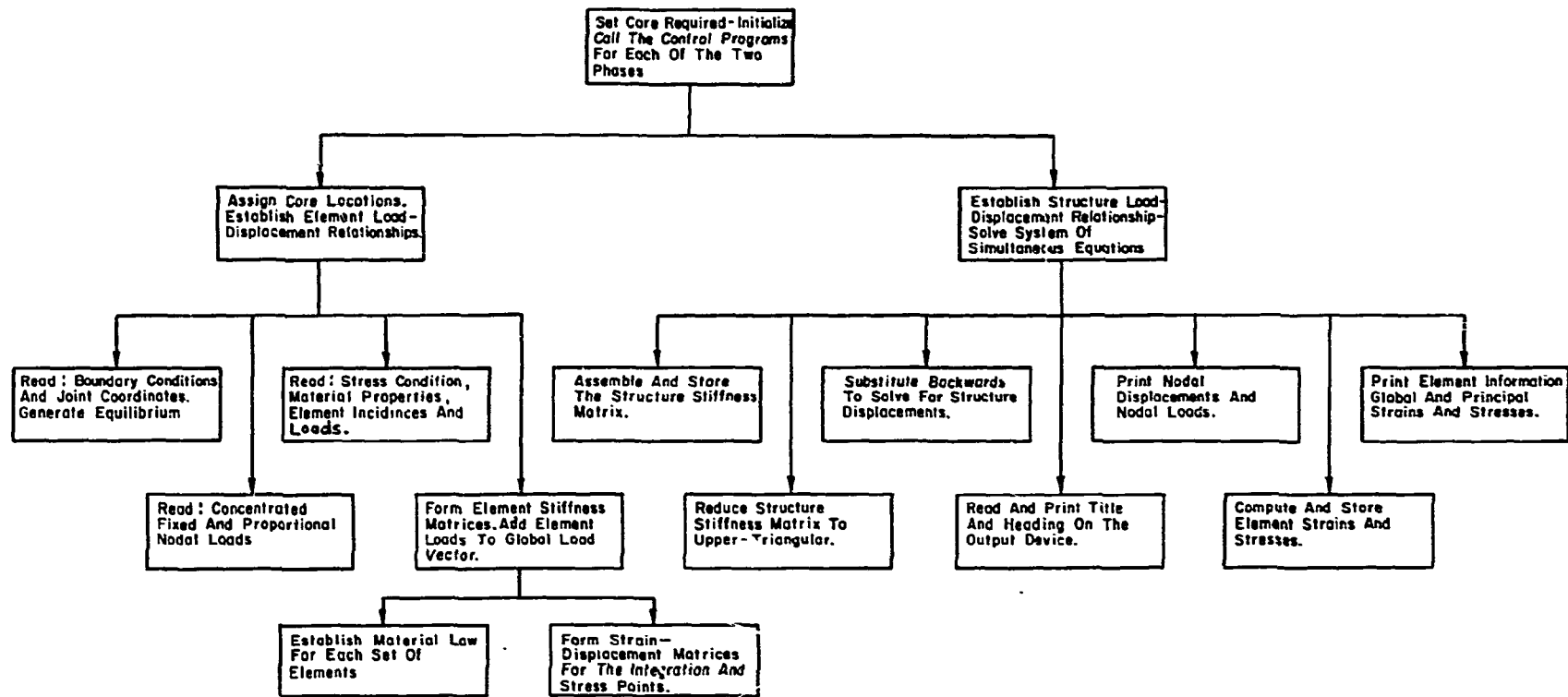
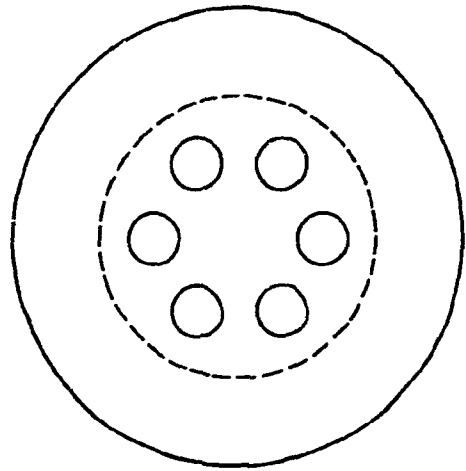
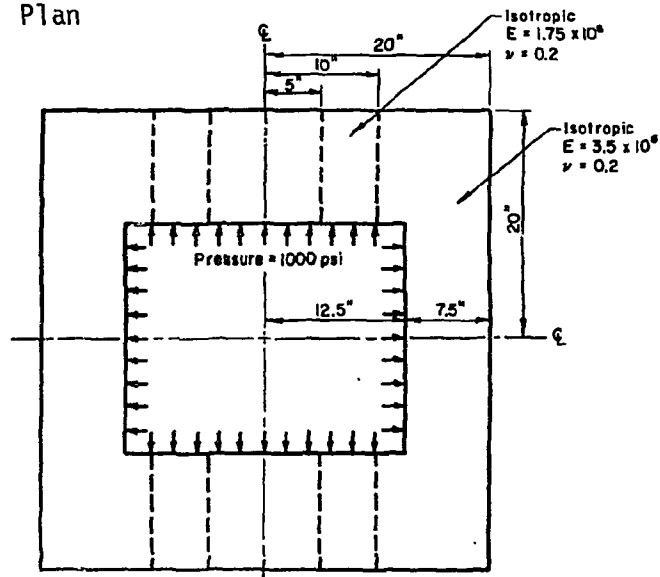


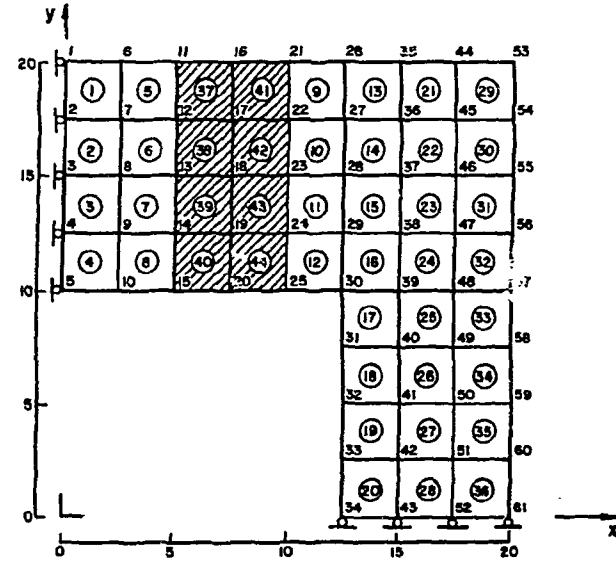
Fig. 4 Overlay (Tree) Structure of ISA



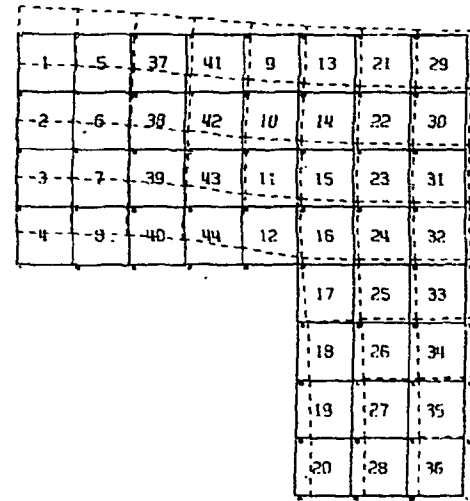
5.a Plan



5.b Vessel Geometry and Loading



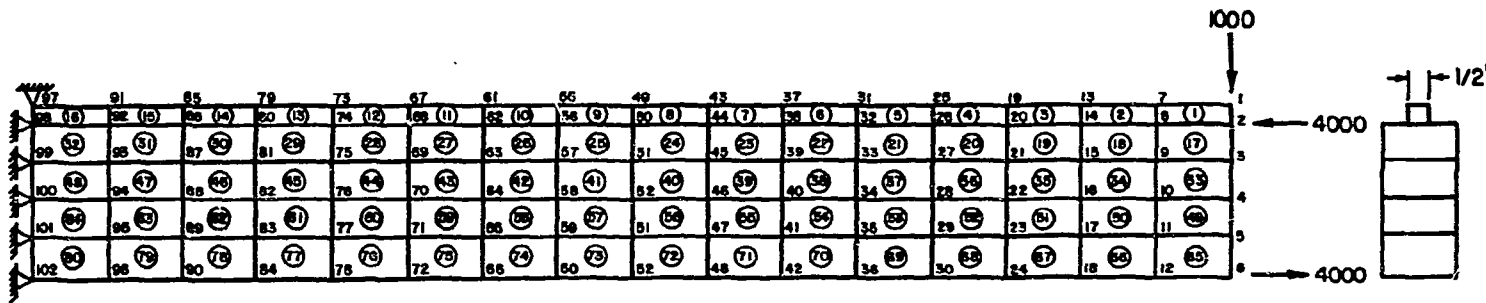
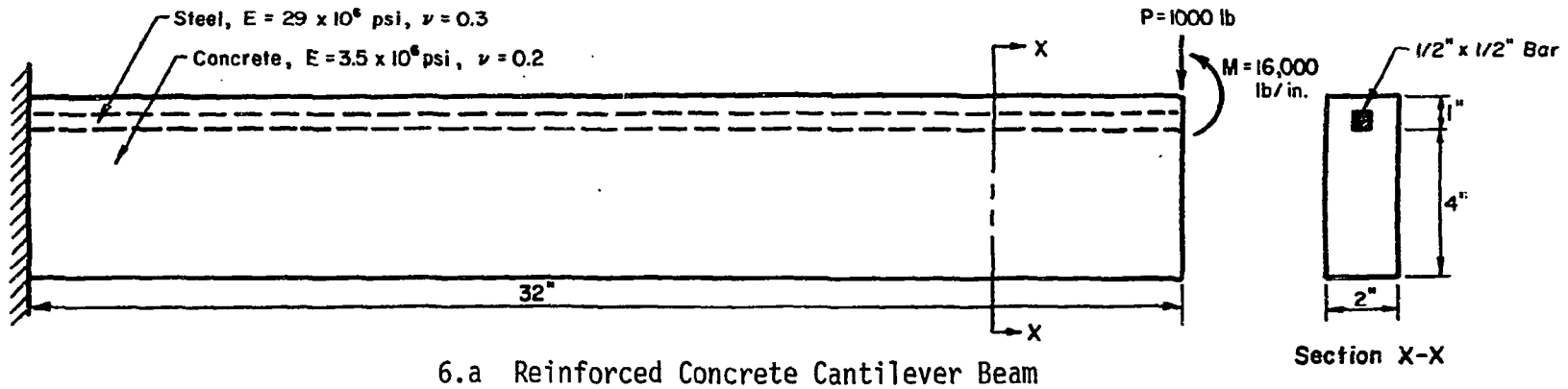
5.c



5.d

5.c and d Finite-element Model of One Quarter of the Vessel and Deformed Mesh Plotted on the Calcomp

Fig. 5 Example Problem (1) Thick-walled Concrete Pressure Vessel with Penetrations



EXAMPLE PROBLEM (1): R.C. CANTILEVER BEAM - PLANE STRESS

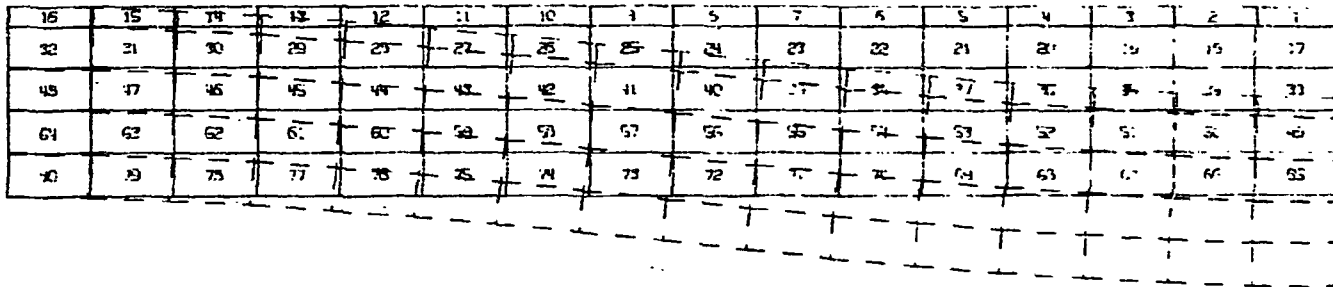


Fig. 6 Example Problem (2): Cantilever Beam

Appendix A
PROGRAM CAPACITY

The capacity of the program is controlled by these statements in the main program:

```
COMMON          POOL (m)
DIMENSION       PLOAD (n), FLOAD (n)
IPOOL = m
MLOAD = n
```

Two requirements have to be met:

$$m > 3 * (\text{number of joints}) + 4 * (\text{total number of elements})$$

$$n \geq 2 * (\text{number of joints})$$

It is advisable however to make n as large as possible because that speeds up the process of solving the equations of equilibrium.