No. 236
July 1981

INTERACTIVE DATA AND GRAPHIC DISPLAY PROCESSORS

PREMSAP and MSAPLOT

FOR THE MICHIGAN STRUCTURAL ANALYSIS PROGRAM

MSAP and SAP IV

by

Movses J. Kaldjian

Department of Naval Architecture
and Marine Engineering
College of Engineering
The University of Michigan
Ann Arbor, Michigan 48109

ABSTRACT

A computer program called PREMSAP to prepare input data for structural analysis and design of structures, including car frames and ships, along with a program called MSAPLOT to display graphically the mathematical models, have been developed for the University of Michigan Structural Analysis Program MSAP. The latter is based on SAP IV and has the same input format and finite element library. A contact element (truss with zero length) which is not found in SAP IV, has been added to MSAP program.

PREMSAP is a format free interactive program with quadrilaterial and linear, nodal point and element mesh generation options for preparing the input data for MSAP for static and dynamic loads. It can be run from any terminal on a time sharing computer system, and provides the engineer an easy and efficient way of communicating the mathematical model of his structure to the computer and obtaining his results in a matter of minutes.

Using the PREMSAP output, or otherwise prepared MSAP data, MSAPLOT plot line, plane and 3/D objects in isometric, perspective and hidden line forms. MSAPLOT allows the user of MSAP to accomplish the following:

1. Check visually on the screen the correctness of the input data of the model being studied.
2. View the deformed structure due to static loads or dynamic mode analyses.
3. Produce ink drawings through the CalComp plotter for inclusion in reports.
4. Draw stress contours for models made of membrane and eight-node brick elements for any one of the six stress components.

Use of PREMSAP and MSAPLOT programs in conjunction with MSAP, has been proved very successful by consultants, researchers and students alike.

TABLE OF CONTENTS

1. Introduction

The interactive pre- and post processor programs presented here are advanced versions of PREMSAP[1,2,9]* and MSAPLOT.[3,4] Their purpose is to prepare and view the input data and the output results of the Michigan Structural Analysis Program (MSAP). The latter is a slightly modified version of SAP IV[5,6] and has identical input format as SAP IV. A new contact element (truss with zero length) has been added to MSAP library of elements. See Appendix for description.

MSAP is a general structural analysis program for static and dynamic response of linear systems. Its input format is rather involved for ordinary application, especially so since there are many options available.

The purpose of PREMSAP is to provide the engineer an easy guide and an efficient method of communicating his mathematical model to the computer and obtain results by avoiding unnecessary and time consuming mistakes.

PREMSAP is an independent format-free interactive data preprocessor program and can be run from any terminal on a time sharing computer system. It is written in FORTRAN IV and is operational on IBM 370/168 and Amdahl 470V/8 at The University of Michigan. PREMSAP does away completely with formatted card preparation both in static and dynamic modes. It has linear and quadrilateral (2/D) nodal point and element mesh generation features.

MSAPLOT on the other hand permits a visual check of the discretized structural model using the input data prepared by PREMSAP for MSAP or prepared otherwise. This is important, for mistakes in element difinition would result in unnecessary expenditure to the user.

MSAPLOT too is an independent unit. It will plot, if desired, the deformed structure after MSAP has been run. It has perspective and hidden line features as well as a quick CalComp drawing option.

---

*Superscripts refer to references at the end of this report.

Once the input data for MSAP is prepared for a given structure, the user calls for MSAPLOT from a graphic display terminal (Tektronix 4010, 4012, etc.) and sees whether his structure is modeled correctly. If not, he adjusts the input data to eliminate the errors. He is then ready to call on MSAP and obtain the displacements and stresses throughout the structure.

Calling MSAPLOT after MSAP has been run, allows the user to view the deformed structure graphically. A drawing of any size of the initial and/or deformed structure in any orientation, in isometric or perspective with or without hidden line option can be obtained by callingon the CalComp plot option of the program's menu.

MSAPLOT is operational on IBM 370/168 and Amdahl 470V/8 at The University of Michigan. It utilizes the graphics libraries CKLIB[7] and MGLIB[8] to manipulate and display various basic finite element patterns.

MSAPLOT is written in standard Fortran IV, and is transferable to other computers.

No attempt will be made here to describe the MSAP program's element components nor its input data statements. For this the reader is referred to SAP IV[5] manual published by The University of California, Berkeley. The latter and MSAP have the same element libraries and idnetical input data formats. See Appendix for Contact element description.

This paper discusses mainly the new features of PREMSAP and MSAPLOT. For other details and additional description of above see References 1, 2, 3, 4 and 9.

2. PREMSAP

2.1 Input/Output

PREMSAP is an interactive computer program developed to lead the user each step of the way to communicate all the details of his problem to the computer.

After the user has defined his problem with proper sketches and dimensions, from a terminal he calls on PREMSAP. The computer responds by printing a statement requesting some specific piece of information from the user regarding his problem. The user answers back by typing the required information and then returns the carriage. When more than one item is requested per line they must be separated by commas. Either integer or floating point numbers can be used throughout.

The process of requesting information by the computer and the user's response to it continues till the last bit of information necessary to complete the input data for MSAP is accomplished.

All the information received above is stored internally by the computer on separate tapes with different formats. The first tape is printed on a file named -CDATA . The second tape is stored either on a temporary or a permanent file and is to be named by the user as he chooses. It contains the information supplied by -CDATA , arranged according to MSAP input format ready to be run in MSAP or MSAPLOT.

It is of course recommended, though not essential for simple problems, to read the SAP IV[5] (MSAP) manual and get acquainted with finite element techniques prior to running PREMSAP on the terminal.

2.2 Program Description

MSAP contains the following ten element types.[5]

1) Three-dimensional truss
2) Three-dimensional beam
3) Plane stress membrane
4) Two-dimensional plane stress, plane strain and axisymmetric
5) Three-dimensional solid
6) Plate and shell

7) Boundary

8) Variable-number-nodes thick shell

9) Pipe element

10) Contact (link) element

The above element types are grouped by PREMSAP into four categories, namely:

<u>Two-dimensional problems</u>

1) truss, plane stress, plane strain, axisymmetric, boundary, and contact elements

2) beam, plate and shell, boundary, and pipe elements

<u>Three-dimensional problems</u>

3) truss, three-dimensional solid, boundary, variable-number-nodes thick shell, contact elements

4) beam, plate and shell, boundary, and pipe elements

The first information to be supplied is the analysis number, i.e., 0 for static, 1 for dynamic, and 2 for dynamic addition. (The Input mode type is eliminated in the current version). The next bit of information to be supplied is the title of the problem on one line. This is followed by the number of joints (nodal points), the number of element types, and the number of load cases. Then through a number of yes or no type questions PREMSAP identifies the category and the nodal mesh generation options of the problem being studied. These are:

. . . IS THE PROBLEM 2-DIMENSIONAL?
. . . ENTER Y FOR YES OR N FOR NO

. . . ARE THERE BEAM, PIPE, THIN SHELL OR PLATE ELEMENTS
. . . INVOLVED IN THE PROBLEM? (ENTER Y OR N)

. . . IS 2-D (QUADRILATERAL) NODAL POINT MESH GENERATION
. . . TO BE PERFORMED? (ENTER Y OR N)

and

. . . IS LINEAR NODAL POINT MESH GENERATION
. . . TO BE PERFORMED? (ENTER Y OR N)

After these questions are answered, the program prompts the user for specific information according to the category of the element and the mesh generation options being used. They include:

1) <u>Nodal Point Mesh Generation</u>

Global nodes I, J, K, L, and nodal number increments

First and last nodal points, and the nodal point mesh generation increment KN for each set

2) <u>Nodal Point Data</u>

The coordinates

The boundary condition codes (0 for free, 1 for constrained)

3) <u>Element Mesh Generation</u>

Global nodes I, J, K, L, and nodal number increments

First and last elements, and the element mesh generation increment KN for each set

Element type number

Number of elements

a) Material properties

b) Geometric properties

Element nodal points

(Repeat item 3 for each different element type)

4) Load Data

Number of joints with concentrated loads and moments

5) Dynamic Analysis data[5]

Number of frequencies required

Analysis type (1 for eigenvalue, 2 for forced dynamic response, 3 for responses spectrum, or 4 for direct integration)

Forcing functions, spectrum data, etc.

As mentioned earlier, all the data the user enters in the computer is stored in the temporary file -CDATA created by the program. The program then reads the data from -CDATA into a second file FILE2 which is to be named by the user himself. FILE2 is the required input for MSAP.

The user is now ready to obtain his results by typing

$LIST FILE2	- for Listing of data
$RUN MSAP 5=FILE2	- for Structural Analysis
$RUN MSAPLOT	- for Graphic Display

2.3 Computer Examples

For detailed description of the preceding the reader is encouraged to study the ten numerical examples given below. These are direct computer outputs and are obtained by running PREMSAP.

The accompanying CalComp plots attest to the validity of PREMSAP results. They were obtained by running the MSAPLOT program.

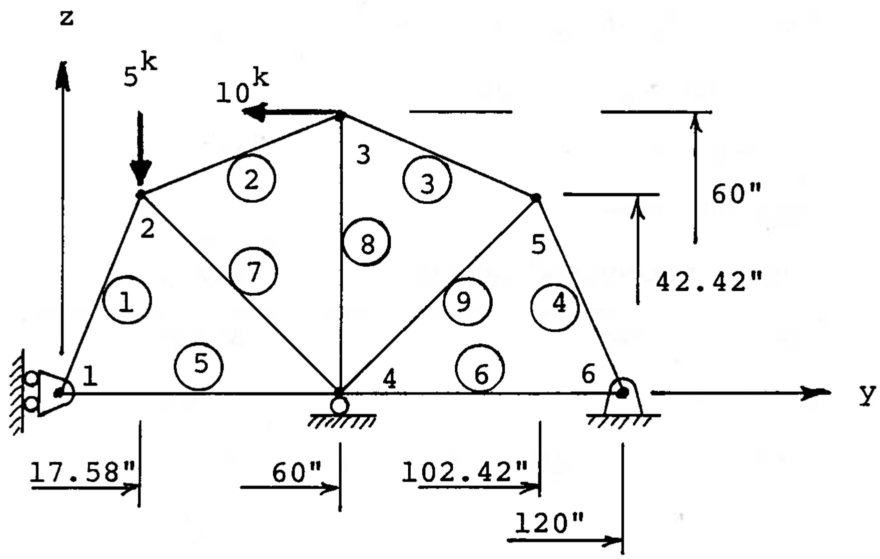
Example	Title	Element Type used
1	2/D Truss	1
2	Frame Structure	2
3	Tower Study (Dynamic)	1,2,7
4	Free-Free Beam (Dynamic)	2,7
5	3/D Membrane Dome	3
6	Open Web Frame	4
7	Precast Wall-Plate (shell)	6,7
8	Solid Flat Bar	5,7,8
9	3/D Pipe	7,12
10	Beam w/ Contact Elements	4,9

EXAMPLE: 1

2/D Truss

Element Type

Truss (1)



Section Properties:

Type	#1	#2
Area	$4in^2$	$6in^2$

Material Properties:

Y. Modulus = 30,000 ksi

Poisson's ratio = 0.3

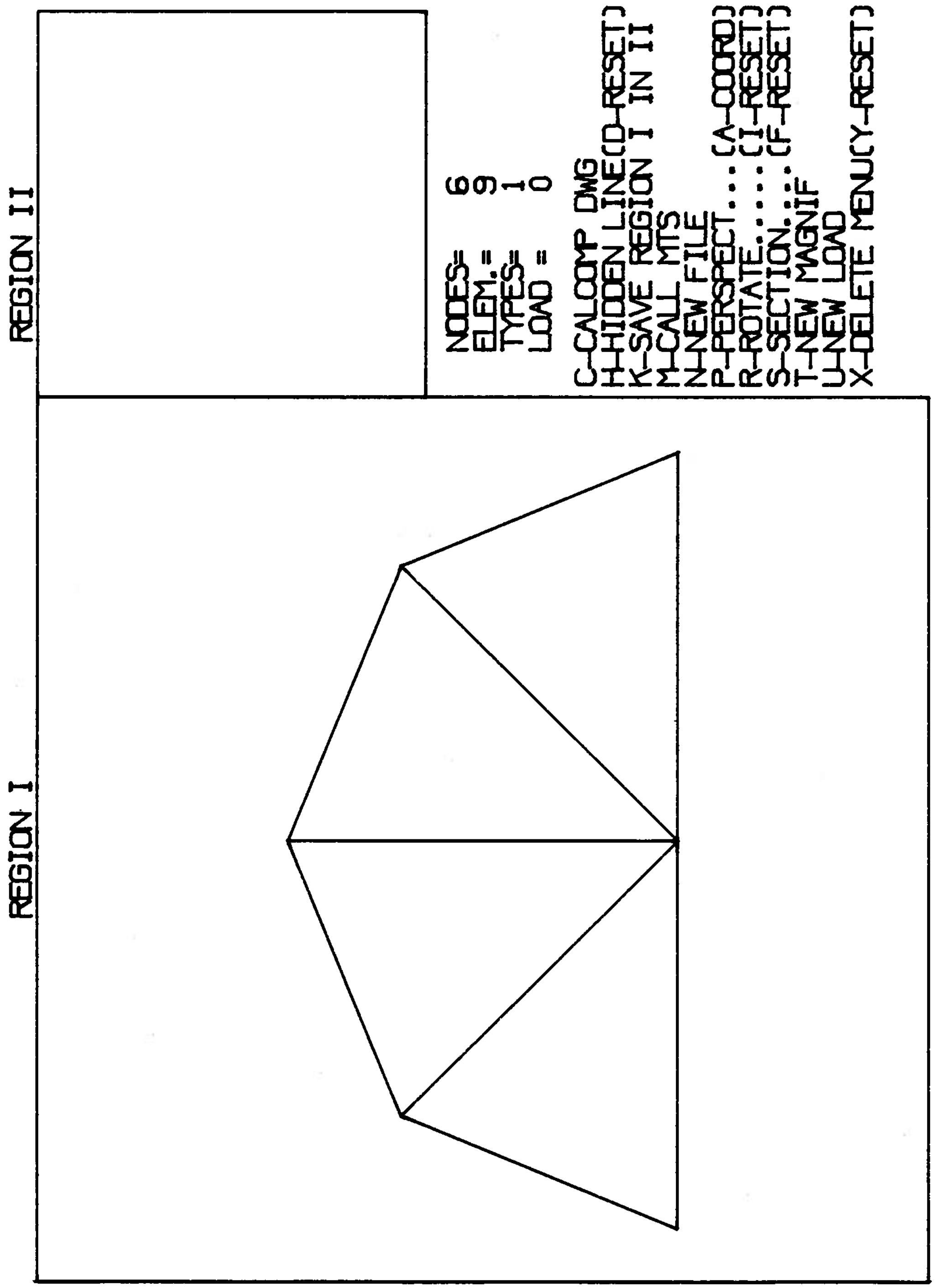
Connectivity:

Elem	I	J	Section No.
1	1	2	2
2	2	3	2
3	3	5	2
4	5	6	2
5	1	4	2
6	4	6	2
7	2	4	1
8	4	3	1
9	4	5	1

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
       2/D truss

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     6,1,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     n

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     n

 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     n

    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y-DIR., IN Z-DIR.
     1:-  0,0,0,1,0
     2:-  17.58,42.42
     3:-  60,60
     4:-  60,0,0,0,1
     5:-  102.42,42.42
     6:-  120,0,0,1,1

    *** ELEMENT DATA

 >> ENTER ELEMENT TYPE NUMBER
 >> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
 >> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
 >> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
 >> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
 >> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
 >> PIPE ELEMENT =12)
     1
```

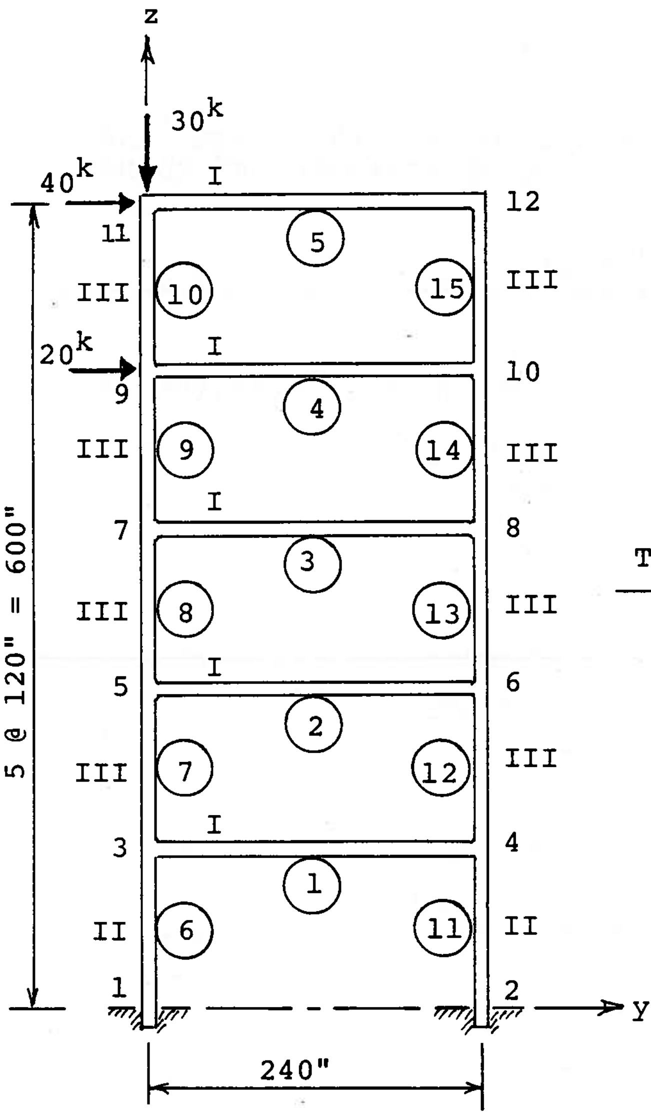
```
>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
     n

   *** TRUSS ELEMENTS

>> ENTER NUMBER OF ELEMENTS AND NUMBER OF DIFF. MATERIALS
>> (INCLUDE DIFF. AREAS IN THE LATTER)
     9,2

>> FOR EACH DIFF. MATERIAL (OR AREA) ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, CROSS-SECTIONAL AREA, AND
>> MAS  & WEIGHT DENSITIES
     1:-   30000, 4, 0, .000283
     2:-   30000, 6,,,

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS. I-(BEGINNING), J-(ENDING), AND MATERIAL I.D. NO.
     1:-   1,2,2
     2:-   2,3,2
     3:-   3,5,2
     4:-   5,6,2
     5:-   1,4,2
     6:-   4,6,2
     7:-   2,4,1
     8:-   4,3,1
     9:-   4,5,1

   ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
     2

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), AND LOADS IN Y-, Z-DIR.
     2, 0, -5
     3, -10

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
     truss

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST TRUSS                   ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=TRUSS        - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT             - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list truss
>     1       2/D TRUSS
>     2         6    1    1    0    0    0
>     3         1   -1    1    0   -1   -1   -1       0.0       0.0       0.0    0       0.0
>     4         2   -1    0    0   -1   -1   -1       0.0    17.580    42.420    0       0.0
>     5         3   -1    0    0   -1   -1   -1       0.0    60.000    60.000    0       0.0
>     6         4   -1    0    1   -1   -1   -1       0.0    60.000       0.0    0       0.0
>     7         5   -1    0    0   -1   -1   -1       0.0   102.420    42.420    0       0.0
>     8         6   -1    1    1   -1   -1   -1       0.0   120.000       0.0    0       0.0
>     9         1    9    2
>    10         1 3.000E+04 0.0       0.0       4.000E+00 2.830E-04
>    11         2 3.000E+04 0.0       0.0       6.000E+00 2.830E-04
>    12              0.0       0.0       0.0       0.0
>    13              0.0       0.0       0.0       0.0
>    14              0.0       0.0       0.0       0.0
>    15              0.0       0.0       0.0       0.0
>    16         1    1    2    2 0.0          0
>    17         2    2    3    2 0.0          0
>    18         3    3    5    2 0.0          0
>    19         4    5    6    2 0.0          0
>    20         5    1    4    2 0.0          0
>    21         6    4    6    2 0.0          0
>    22         7    2    4    1 0.0          0
>    23         8    4    3    1 0.0          0
>    24         9    4    5    1 0.0          0
>    25         2    1 0.0       0.0      -5.000E+00 0.0       0.0       0.0
>    26         3    1 0.0      -1.000E+01 0.0       0.0       0.0       0.0
>    27
>    28              0.0       0.0       0.0       0.0
#END OF FILE

                    1         2         3         4         5         6         7         8
 column    12345678901234567890123456789012345678901234567890123456789012345678901234567890
```


REGION I
REGION II
NODES= 6
ELEM.= 9
TYPES= 1
LOAD = 0
C-CALCOMP DWG
H-HIDDEN LINE(D-RESET)
K-SAVE REGION I IN II
M-CALL MTS
N-NEW FILE
P-PERSPECT.... (A-COORD)
R-ROTATE...... (I-RESET)
S-SECTION..... (F-RESET)
T-NEW MAGNIF
U-NEW LOAD
X-DELETE MENU(Y-RESET)
2/D TRUSS

EXAMPLE: 2

Frame Structure



Element Type

Beam (2)

Material Properties:

Y. Modulus = 30,000 ksi

Poisson's Ratio = 0.3

Section Properties:

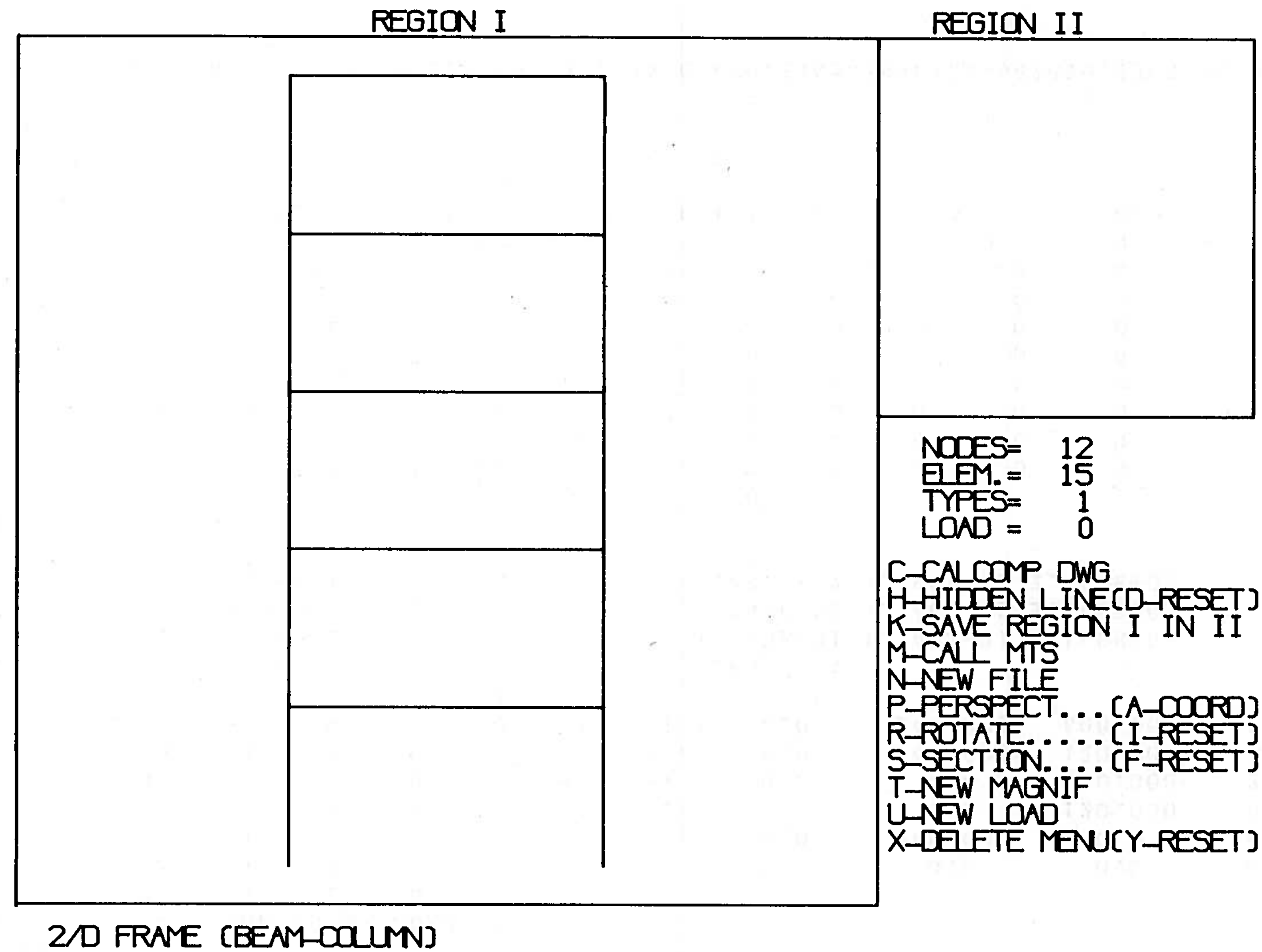
Type	I	II	III
A_{ax}	16.20in^2	17.70	13.20
A_{sh}	7.80in^2	4.25	3.54
I	1140in^4	354	249

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
       2/D frame (beam-column)

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     12,1,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     y

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     n

 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     3,11,2
     4,12,2


    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y-, Z-DIR., AND X-ROT.
     1:-  0,0,0,1,1,1
     2:-  240,0,0,1,1,1
     3:-  0,120,0
    11:-  0,600,0
     4:-  240,120,0
    12:-  240,600
```

```
    *** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
     2

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
     y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     1,5,2
     7,10,2
     12,15,2


    *** BEAM ELEMENTS

>> ENTER NUMBER OF ELEMENTS,NUMBER OF DIFF. GEOMETRY,
>> AND NUMBER OF DIFF. MATERIALS
     15,3,1

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> MASS & WEIGHT DENSITIES
     1:-  30000, .3, 0, .000283

>> FOR EACH DIFF. GEOMETRY ENTER FOLLOWING PER LINE
>> AXIAL AREA, SHEAR AREA, MOMENT OF INERTIA
>> (TO NEGLECT SHEAR DEFORMATIONS SET SHEAR AREAS = 0.)
     1:-  16.20, 7.80, 1140
     2:-  17.70, 4.25,  354
     3:-  13.20, 3.54,  249

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS., I-(BEGINNING), J-(ENDING), K-
>> (K IS ANY POINT A DISTANCE AWAY FROM LINE I-J),
>> AND I.D. NOS., OF MATERIAL AND GEOMETRY
     1:-  4,3,1,1,1
     6:-  3,1,2,1,2
     7:-  5,3,4,1,3
    11:-  2,4,3,1,2
    12:-  4,6,3,1,3
```

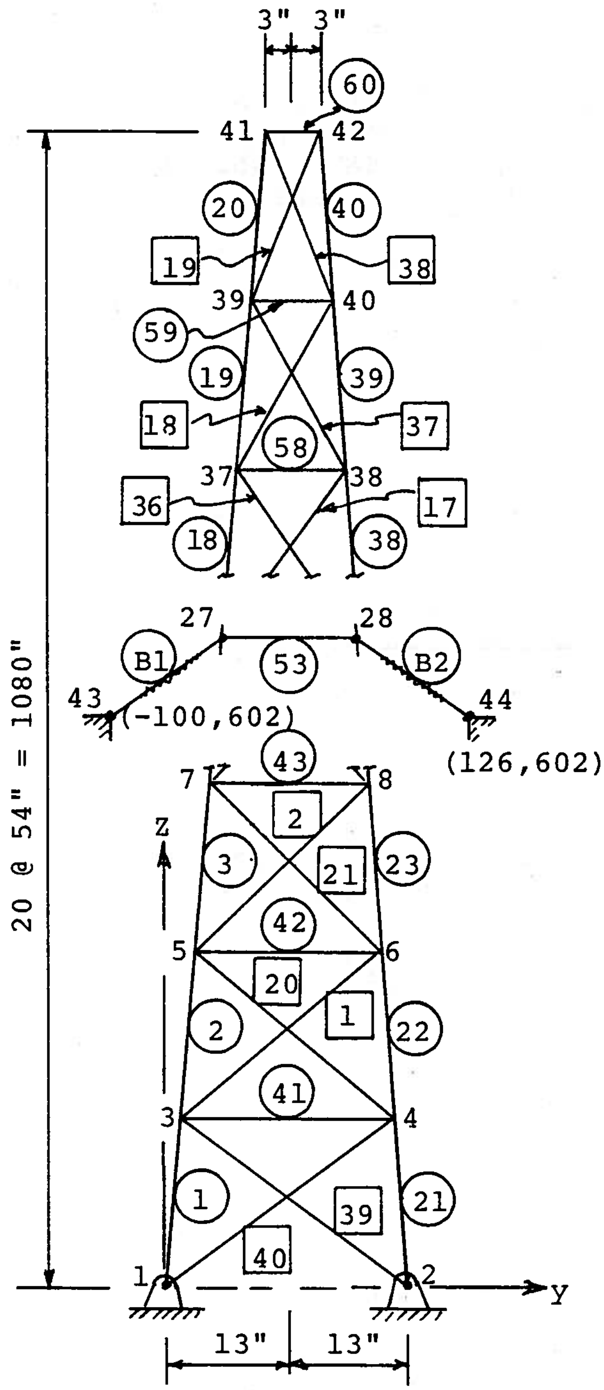
```
   ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
    2

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), LOADS IN Y-, Z-DIR.,
>> AND MOMENT X-X
    9, 20
    11,40,-30

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
    frame

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST FRAME                    ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=FRAME         - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT              - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list frame
>    1       2/D FRAME (BEAM-COLUMN)
>    2        12    1    1    0    0    0
>    3         1   -1    1    1    1   -1   -1      0.0       0.0       0.0      0      0.0
>    4         2   -1    1    1    1   -1   -1      0.0     240.000     0.0      0      0.0
>    5         3   -1    0    0    0   -1   -1      0.0       0.0     120.000    0      0.0
>    6        11   -1    0    0    0   -1   -1      0.0       0.0     600.000    2      0.0
>    7         4   -1    0    0    0   -1   -1      0.0     240.000   120.000    0      0.0
>    8        12   -1    0    0    0   -1   -1      0.0     240.000   600.000    2      0.0
>    9         2   15    3    0    1
>   10         1 3.000E+04 3.000E-01 0.0      2.830E-04
>   11         1 1.620E+01 7.800E+00 0.0      1.140E-01 1.140E-01 1.140E+03
>   12         2 1.770E+01 4.250E+00 0.0      3.540E-02 3.540E-02 3.540E+02
>   13         3 1.320E+01 3.540E+00 0.0      2.490E-02 2.490E-02 2.490E+02
>   14           0.0       0.0       0.0       0.0
>   15           0.0       0.0       0.0       0.0
>   16           0.0       0.0       0.0       0.0
>   17         1    4    3    1    1    1    0    0    0    0     0     0      2
>   18         5   12   11    1    1    1    0    0    0    0     0     0      2
>   19         6    3    1    2    1    2    0    0    0    0     0     0      0
>   20         7    5    3    4    1    3    0    0    0    0     0     0      2
>   21        10   11    9    4    1    3    0    0    0    0     0     0      2
>   22        11    2    4    3    1    2    0    0    0    0     0     0      0
>   23        12    4    6    3    1    3    0    0    0    0     0     0      2
>   24        15   10   12    3    1    3    0    0    0    0     0     0      2
>   25         9    1 0.0        2.000E+01 0.0        0.0       0.0       0.0
>   26        11    1 0.0        4.000E+01-3.000E+01 0.0       0.0       0.0
>   27
>   28           0.0       0.0       0.0       0.0
#END OF FILE
                       1         2         3         4         5         6         7         8
  column      12345678901234567890123456789012345678901234567890123456789012345678901234567890
```


REGION I
REGION II
NODES= 12
ELEM.= 15
TYPES= 1
LOAD = 0
C-CALCOMP DWG
H-HIDDEN LINE(D-RESET)
K-SAVE REGION I IN II
M-CALL MTS
N-NEW FILE
P-PERSPECT....(A-COORD)
R-ROTATE.....(I-RESET)
S-SECTION....(F-RESET)
T-NEW MAGNIF
U-NEW LOAD
X-DELETE MENU(Y-RESET)
2/D FRAME (BEAM-COLUMN)

EXAMPLE : 3

Tower Study (Dynamic)



Element Type

Truss (1)
Beam (2)
Boundary (7)

MAT'L PROPERTIES:
Elastic Mod. = 30,000,000psi
Poison's Ratio = 0.3
Mass Density = .00879
Conc. Mass of 5 units
at N.P. 41 & 42
K for Bound. Elem. = 5750#/"

SECTION PROPERTIES:
Truss: Area = 0.2 in^2

Beam:

Member	Aax	Ash	I
Edge	2.75 in^2	0	2.22 in^4
Horiz	0.20	0	0.01

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
    0,1

 >> ENTER PROBLEM TITLE ON ONE LINE
      dynamic study of tower

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES,
 >> NUMBER OF FREQUENCIES, AND ANALYSIS TYPE CODE
 >> (0= STATIC, 1= EIGENVALUE, 2= FORCED DYNAMIC RESPONSE
 >>  3= RESPONSE SPECTRUM, AND 4= DIRECT INTEGRATION)
    44,3,10,3

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
    y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SH LL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
    y

 >> IS 2 D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
    n

 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
    y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    3,41,2
    4,42,2


   *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y-, Z-DIR., AND X-ROT.
    1:-  0,0,0,1,1,0
```

```
 2:-   26,0,0,1,1,0
 3:-   .5,54,0
41:-   10,1080,0
 4:-   25.5, 54
42:-   16, 1080
43:-   -100, 602, 0,1,1,1
44:-   126, 602, 0,1,1,1

*** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
   7

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   n

*** BOUNDARY ELEMENTS

>> ENTER NUMBER OF ELEMENTS
   2

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODE N (WHERE THE ELEMENT IS PLACED, IN ASCENDING ORDER),
>> NODE I (THIS DEFINES THE DIRECTION OF THE ELEMENT),
>> DISPLACEMENT CODE(0 OR 1), ROTATION CODE(0 OR 1),
>> SPECIFIED DISPLACEMENT, SPECIFIED ROTATION, AND
>> SPRING STIFFNESS (SET TO 1.0*E10 IF LEFT BLANK)
   27,43,1,0,0,0,5750
   28,44,1,0,0,0,5750

>> ENTER ELEMENT TYPE NUMBER
   2

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
```

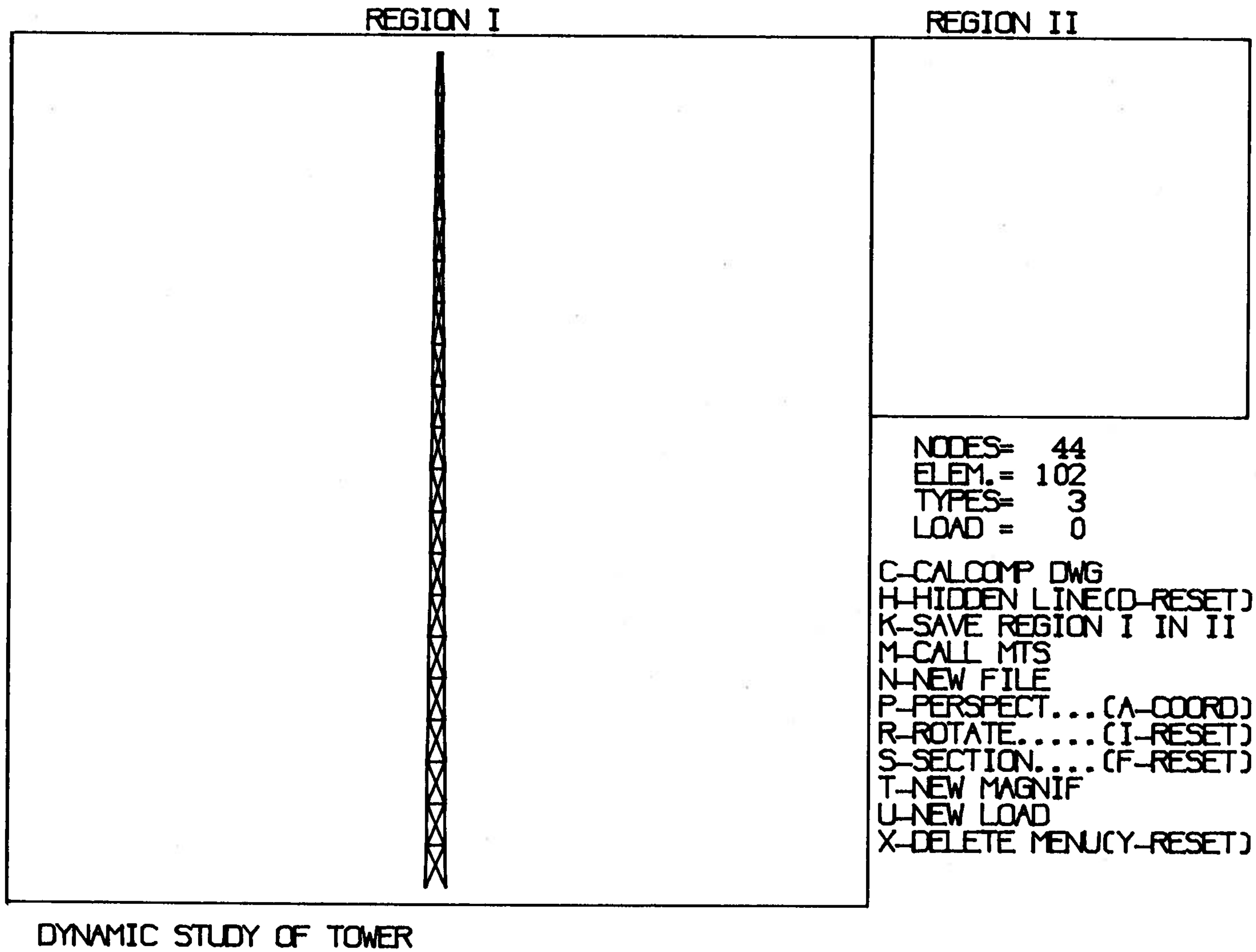
```
    1,20,2
    21,40,2
    41,59,2


   *** BEAM ELEMENTS

>>  NTER NUMBER OF ELEMENTS,NUMBER OF DIFF. GEOMETRY,
>> AND NUMBER OF DIFF. MATERIALS
    60,2,1

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> MASS & WEIGHT DENSITIES
    1:-  30000000, .3, .00879

>> FOR EACH DIFF. GEOMETRY ENTER FOLLOWING PER LINE
>> AXIAL AREA, SHEAR AREA, MOMENT OF INERTIA
>> (TO NEGLECT SHEAR DEFORMATIONS SET SHEAR AREAS = 0.)
    1:-  2.75, .0, 2.22
    2:-  0.2, .0, .01

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS., I-(BEGINNING), J-(ENDING), K-
>> (K IS ANY POINT A DISTANCE AWAY FROM LINE I-J),
>> AND I.D. NOS., OF MATERIAL AND GEOMETRY
    1:-  1,3,43,1,1
   21:-  2,4,43,1,1
   41:-  3,4,42,1,2
   60:-  42,41,39,1,2

>> ENTER ELEMENT TYPE NUMBER
    1

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,19,2
    20,38,2
```

```
   *** TRUSS ELEMENTS

>> ENTER NUMBER OF ELEMENTS AND NUMBER OF DIFF. MATERIALS
>> (INCLUDE DIFF. AREAS IN THE LATTER)
    40,1

>> FOR EACH DIFF. MATERIAL (OR AREA) ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, CROSS-SECTIONAL AREA, AND
>> MASS & WEIGHT DENSITIES
    1:-  30000000, .2, .00879

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS. I-(BEGINNING), J-(ENDING), AND MATERIAL I.D. NO.
    1:-  3,6,1
   20:-  5,4,1
   39:-  2,3,1
   40:-  1,4,1

   ***CONCENTRATED LOAD DATA

>> NOTE: FOR DYNAMIC ANALYSIS
>>       LOAD = MASS, AND MOMENT = ROTATIONAL INERTIA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
    2

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), LOADS IN Y-, Z-DIR.,
>> AND MOMENT X-X
    41,5,5
    42,5,5

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>  ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
    -tower

...ENTER PRINT CODE(0= NO INTERMEDIATE VALUES, 1= YES),
...CONVERGENCE TOLERANCE(0 FOR DEFAULT), AND CUT-OFF
...FREQUENCY(0 FOR DEFAULT)
   ? 0,0,0

...ENTER FACTOR FOR X,Y,Z DIRECTIONS AND
...SPECTRUM TYPE (0= DISPL., 1= ACCEL.)
   ? 0,1,0,1
```

```
...ENTER TITLE FOR SPECTRUM TABLE ON ONE LINE
   ?    spectrum aaa

...ENTER NO. OF DEFINITION  POINTS FOR SPECTRUM AND
...SCALE FACTOR FOR ORDINATES.
   ? 6, 10

... FOR EACH POINT ENTER PERIOD AND ACCELERATION VALUES
    1:-    .0,  4
    2:-    .2, 20
    3:-    .4, 22
    4:-    .6, 17
    5:-    .8, 10
    6:-   1.0,  7

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST -TOWER                    ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=-TOWER         - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT               - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list -tower
>      1       DYNAMIC STUDY OF TOWER
>      2        44    3    0   10    3    0
>      3         1   -1    1    1    0   -1   -1       0.0       0.0       0.0    0      0.0
>      4         2   -1    1    1    0   -1   -1       0.0    26.000       0.0    0      0.0
>      5         3   -1    0    0    0   -1   -1       0.0     0.500    54.000    0      0.0
>      6        41   -1    0    0    0   -1   -1       0.0    10.000  1080.000    2      0.0
>      7         4   -1    0    0    0   -1   -1       0.0    25.500    54.000    0      0.0
>      8        42   -1    0    0    0   -1   -1       0.0    16.000  1080.000    2      0.0
>      9        43   -1    1    1    1   -1   -1       0.0  -100.000   602.000    0      0.0
>     10        44   -1    1    1    1   -1   -1       0.0   126.000   602.000    0      0.0
>     11         7    2
>     12           1.0
>     13        27   43    0    0    0    1    0    0 0.0       0.0      5.750E+03
>     14        28   44    0    0    0    1    0    0 0.0       0.0      5.750E+03
>     15         2   60    2    0    1
>     16         1 3.000E+07 3.000E-01 8.790E-03 0.0
>     17         1 2.750E+00 0.0       0.0       2.220E-04 2.220E-04 2.220E+00
>     18         2 2.000E-01 0.0       0.0       1.000E-06 1.000E-06 1.000E-02
>     19           0.0       0.0       0.0       0.0
>     20           0.0       0.0       0.0       0.0
>     21           0.0       0.0       0.0       0.0
>     22         1    1    3   43    1    1    0    0    0    0    0    0    2
>     23        20   39   41   43    1    1    0    0    0    0    0    0    2
>     24        21    2    4   43    1    1    0    0    0    0    0    0    2
>     25        40   40   42   43    1    1    0    0    0    0    0    0    2
>     26        41    3    4   42    1    2    0    0    0    0    0    0    2
>     27        59   39   40   42    1    2    0    0    0    0    0    0    2
>     28        60   42   41   39    1    2    0    0    0    0    0    0    0
>     29         1   40    1
>     30         1 3.000E+07 0.0       8.790E-03 2.000E-01 0.0
>     31           0.0       0.0       0.0       0.0
>     32           0.0       0.0       0.0       0.0
>     33           0.0       0.0       0.0       0.0
>     34           0.0       0.0       0.0       0.0

                  1         2         3         4         5         6         7         8
         12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

```
>     35          1     3     6     1 0.0           2
>     36         19    39    42     1 0.0           2
>     37         20     5     4     1 0.0           2
>     38         38    41    40     1 0.0           2
>     39         39     2     3     1 0.0           0
>     40         40     1     4     1 0.0           0
>     41         41     0 0.0         5.000E+00 5.000E+00 0.0       0.0       0.0
>     42         42     0 0.0         5.000E+00 5.000E+00 0.0       0.0       0.0
>     43
>     44             1.0       0.0       0.0       0.0
>     45          0     0    16   0.0       0.0
>     46      0.0         1.000     0.0           1
>     47         SPECTRUM AAA
>     48          6   10.0000
>     49          0.0        4.0000
>     50          0.2000    20.0000
>     51          0.4000    22.0000
>     52          0.6000    17.0000
>     53          0.8000    10.0000
>     54          1.0000     7.0000
#END OF FILE
                         1         2         3         4         5         6         7         8
                12345678901234567890123456789012345678901234567890123456789012345678901234567890
```


REGION I
REGION II
NODES= 44
ELEM.= 102
TYPES= 3
LOAD = 0
C-CALCOMP DWG
H-HIDDEN LINE(D-RESET)
K-SAVE REGION I IN II
M-CALL MTS
N-NEW FILE
P-PERSPECT....(A-COORD)
R-ROTATE.....(I-RESET)
S-SECTION....(F-RESET)
T-NEW MAGNIF
U-NEW LOAD
X-DELETE MENU(Y-RESET)
DYNAMIC STUDY OF TOWER

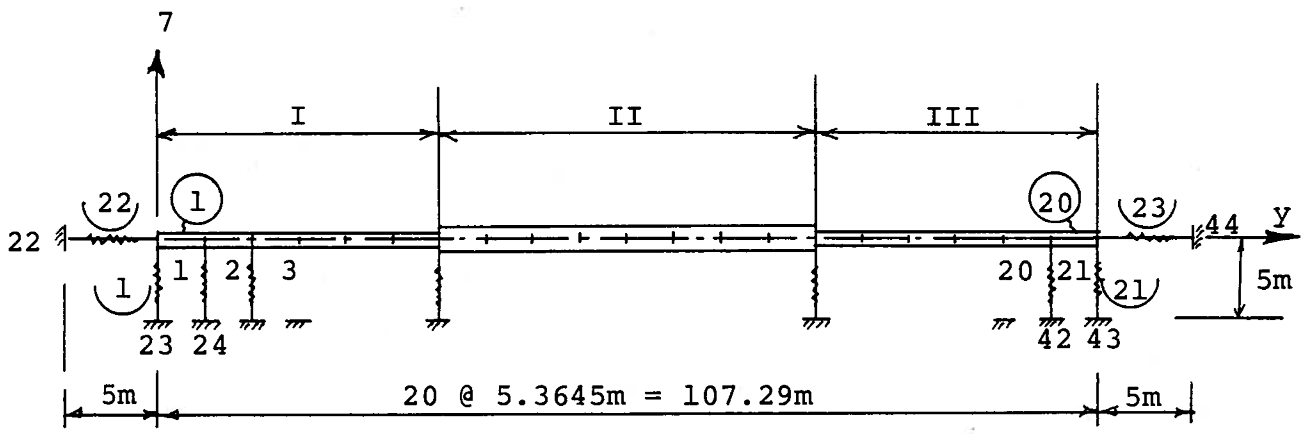
EXAMPLE: 4

Free-free Beam (Dynamic Study)

Element Type

Beam (2)

Boundary (7)



Material Properties:

Y. Modulus = $207 * 10^6$ kg/m^2

Poisson's Ratio = 0.3

Spring Constant = 0.01 kg/m

Section Properties:

Type	I	II	III
A_{ax}	0.400m^2	0.400	0.400
A_{sh}	0.058m^2	0.0823	0.0647
I	1.890m^4	4.229	3.129

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,1

 >> ENTER PROBLEM TITLE ON ONE LINE
         free-free beam (ship)

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES,
 >> NUMBER OF FREQUENCIES, AND ANALYSIS TYPE CODE
 >> (0= STATIC, 1= EIGENVALUE, 2= FORCED DYNAMIC RESPONSE
 >>  3= RESPONSE SPECTRUM, AND 4= DIRECT INTEGRATION)
     45,2,9,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     y

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     n

 >> IS LINEAR NODAL POINT  ESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     1,21,1
     23,43,1


    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y-, Z-DIR., AND X-ROT.
     1:-  0,0,0,0,0,0
    21:-  107.29,0
    22:-  -5,0,0,1,1,1
    23:-  0,-5,0,1,1,1
    43:-  107.29,-5,0,1,1,1
    44:-  112.29,0,0,1,1,1
    45:-  107.29,20,0,1,1,1
```

```
 *** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
    2

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS ( N INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,6,1
    7,14,1
    15,20,1


 *** BEAM ELEMENTS

>> ENTER NUMBER OF ELEMENTS,NUMBER OF DIFF. GEOMETRY,
>> AND NUMBER OF DIFF. MATERIALS
    20,3,1

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> MASS & WEIGHT DENSITIES
    1:-  207100000, .3, 0, 0

>> FOR EACH DIFF. GEOMETRY ENTER FOLLOWING PER LINE
>> AXIAL AREA, SHEAR AREA, MOMENT OF INERTIA
>> (TO NEGLECT SHEAR DEFORMATIONS SET SHEAR AREAS = 0.)
    1:-  .4, .058,  1.890
    2:-  .4, .0823, 4.229
    3:-  .4, .0647, 3.129

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS., I-(BEGINNING), J-(ENDING), K-
>> (K IS ANY POINT A DISTANCE AWAY FROM LINE I-J),
>> AND I.D. NOS., OF MATERIAL AND GEOMETRY
    1:-  1,2,45,1,1
    7:-  7,8,45,1,2
   15:-  15,16,45,1,3

>> ENTER ELEMENT TYPE NUMBER
    7
```

```
>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,21,1


   *** BOUNDARY ELEMENTS

>> ENTER NUMBER OF ELEMENTS
    23

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODE N (WHERE THE ELEMENT IS PLACED, IN ASCENDING ORDER),
>> NODE I (THIS DEFINES THE DIRECTION OF THE ELEMENT),
>> DISPLACEMENT CODE(0 OR 1), ROTATION CODE(0 OR 1),
>> SPECIFIED DISPLACEMENT, SPECIFIED ROTATION, AND
>> SPRING STIFFNESS (SET TO 1.0*E10 IF LEFT BLANK)
    1:-  1,23,1,0,0,0,.01
   22:-  1,22,1,0,0,0,.01
   23:-  21,44,1,0,0,0,.01

   ***CONCENTRATED LOAD DATA

>> NOTE: FOR DYNAMIC ANALYSIS
>>       LOAD = MASS, AND MOMENT = ROTATIONAL INERTIA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
    11

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), LOADS IN Y-, Z-DIR.,
>> AND MOMENT X-X
    1, 55.625, 55.625, 156.67
    3,147.835,147.835, 416.25
    5,366.6,  366.6,   666.15
    7,303.4,  303.4,   573.65
    9,317.65, 317.65,  819.10
    11,311.95,311.95,  892.85
    13,292.40,292.40, 1178.20
    15,411.75,411.75,  961.60
    17,213.12,213.12,  750.65
    19,103.38,103.38,  513.70
    21, 36.32, 36.32,  265.90
```

```
 >> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
 >> ACCORDING TO THE INPUT FORMAT OF MSAP
 >> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
     freebeam

...ENTER PRINT CODE(0= NO INTERMEDIATE VALUES, 1= YES),
...CONVERGENCE TOLERANCE(0 FOR DEFAULT), AND CUT-OFF
...FREQUENCY(0 FOR DEFAULT)
   ? 0

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST FREEBEAM                  ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=FREEBEAM       - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT               - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

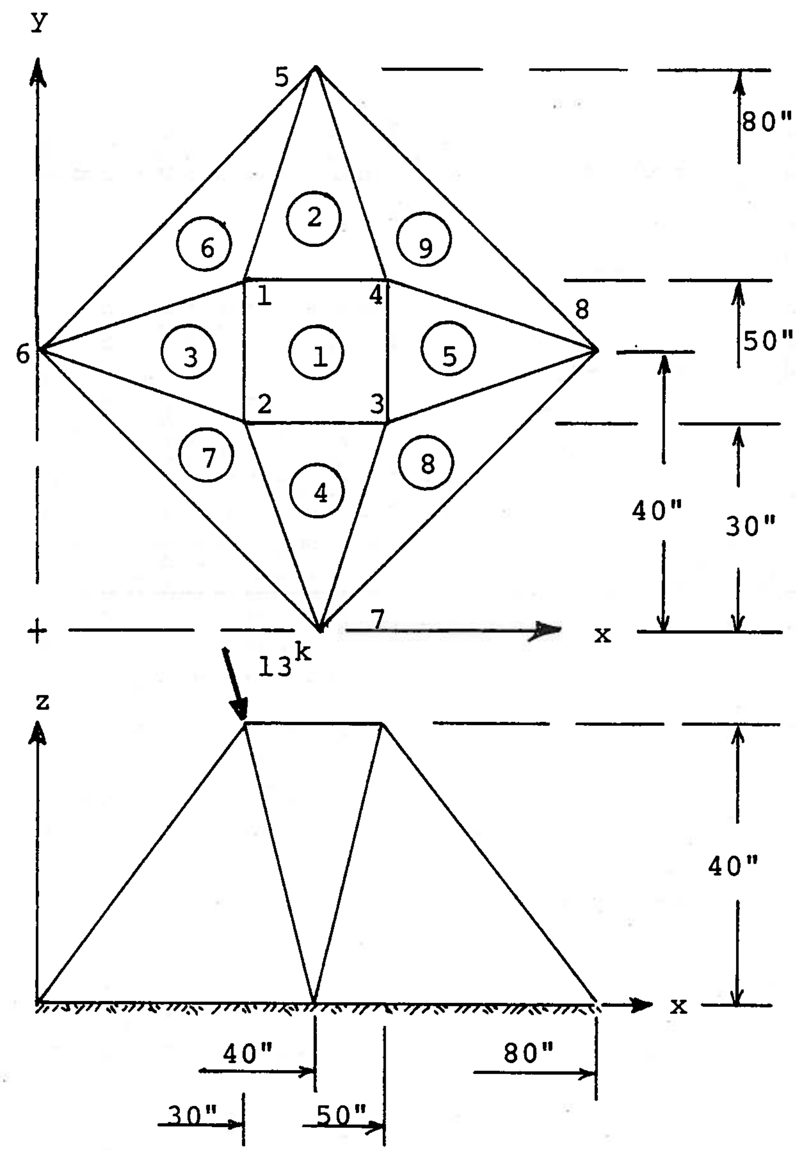
```
#list freebeam
>     1         FREE-FREE BEAM (SHIP)
>     2         45    2    0    9    1    0
>     3          1   -1    0    0    0   -1   -1       0.0        0.0        0.0      0      0.0
>     4         21   -1    0    0    0   -1   -1       0.0      107.290      0.0      1      0.0
>     5         22   -1    1    1    1   -1   -1       0.0       -5.000      0.0      0      0.0
>     6         23   -1    1    1    1   -1   -1       0.0        0.0       -5.000    0      0.0
>     7         43   -1    1    1    1   -1   -1       0.0      107.290     -5.000    1      0.0
>     8         44   -1    1    1    1   -1   -1       0.0      112.290      0.0      0      0.0
>     9         45   -1    1    1    1   -1   -1       0.0      107.290     20.000    0      0.0
>    10          2   20    3    0    1
>    11          1 2.071E+08 3.000E-01 0.0       0.0
>    12          1 4.000E-01 5.800E-02 0.0       1.890E-04 1.890E-04 1.890E+00
>    13          2 4.000E-01 8.230E-02 0.0       4.229E-04 4.229E-04 4.229E+00
>    14          3 4.000E-01 6.470E-02 0.0       3.129E-04 3.129E-04 3.129E+00
>    15            0.0       0.0       0.0       0.0
>    16            0.0       0.0       0.0       0.0
>    17            0.0       0.0       0.0       0.0
>    18          1    1    2   45    1    1    0    0    0    0    0    0    1
>    19          6    6    7   45    1    1    0    0    0    0    0    0    1
>    20          7    7    8   45    1    2    0    0    0    0    0    0    1
>    21         14   14   15   45    1    2    0    0    0    0    0    0    1
>    22         15   15   16   45    1    3    0    0    0    0    0    0    1
>    23         20   20   21   45    1    3    0    0    0    0    0    0    1
>    24          7   23
>    25            1.0
>    26          1   23                  1    0    1 0.0       0.0       0.01
>    27         21   43                  1    0    1 0.0       0.0       0.01
>    28          1   22                  1    0    0 0.0       0.0       0.01
>    29         21   44                  1    0    0 0.0       0.0       0.01
>    30          1    0 0.0       5.563E+01 5.563E+01 1.567E+02 0.0       0.0
>    31          3    0 0.0       1.478E+02 1.478E+02 4.163E+02 0.0       0.0
>    32          5    0 0.0       3.666E+02 3.666E+02 6.661E+02 0.0       0.0
>    33          7    0 0.0       3.034E+02 3.034E+02 5.736E+02 0.0       0.0
>    34          9    0 0.0       3.176E+02 3.176E+02 8.191E+02 0.0       0.0
>    35         11    0 0.0       3.119E+02 3.119E+02 8.929E+02 0.0       0.0
>    36         13    0 0.0       2.924E+02 2.924E+02 1.178E+03 0.0       0.0
>    37         15    0 0.0       4.118E+02 4.118E+02 9.616E+02 0.0       0.0
>    38         17    0 0.0       2.131E+02 2.131E+02 7.506E+02 0.0       0.0
>    39         19    0 0.0       1.034E+02 1.034E+02 5.137E+02 0.0       0.0
>    40         21    0 0.0       3.632E+01 3.632E+01 2.659E+02 0.0       0.0
>    41
>    42            1.0       0.0       0.0       0.0
>    43          0    0   16   0.0       0.0
#END OF FILE
#                        1         2         3         4         5         6         7         8
               12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

EXAMPLE: 5

3/D Membrane Dome

Element Type

Membrane (3)



Y.M. = 30,000 ksi

ν = 0.3

1/4"- Plate

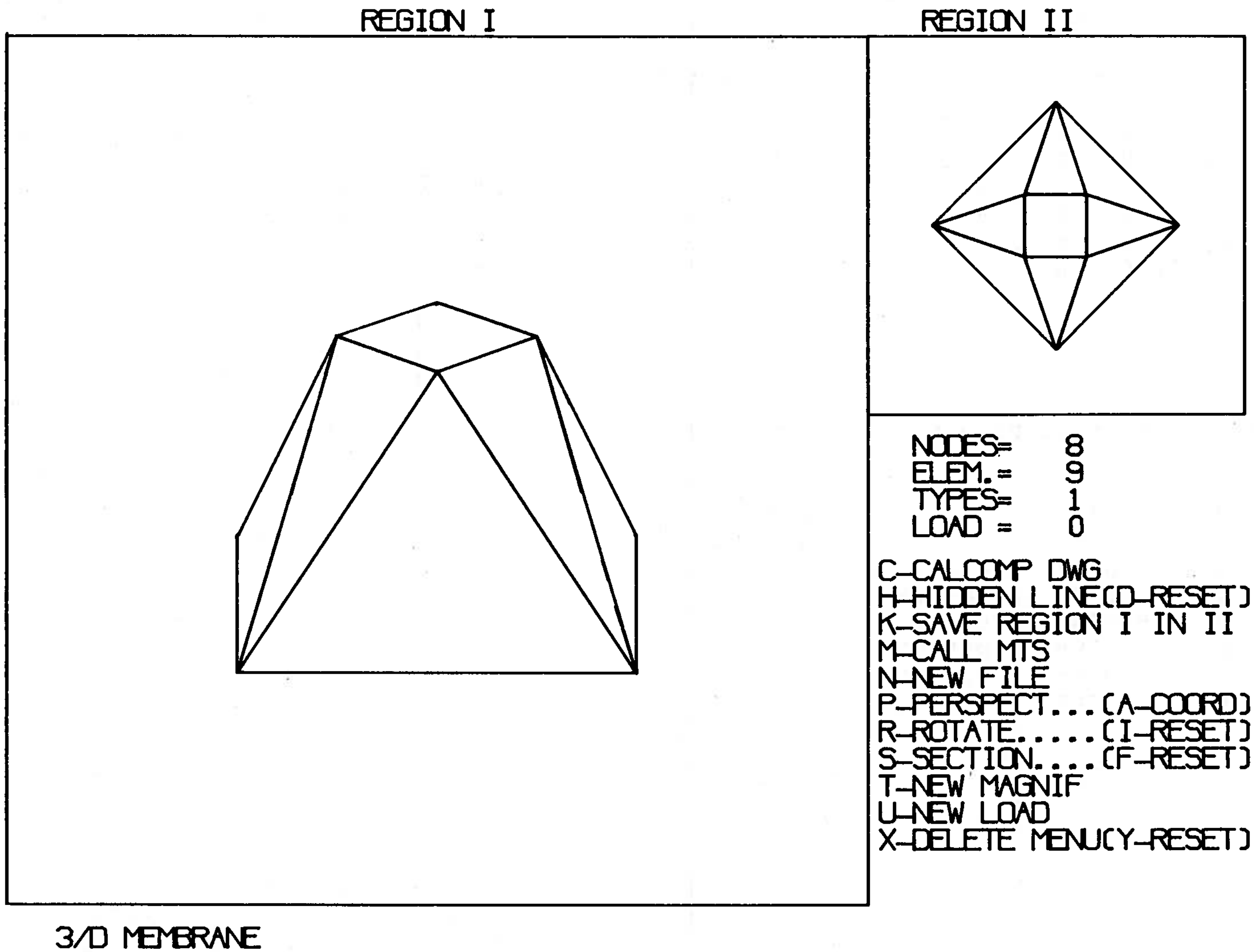
Loads:

N.P.	F_x	F_y	F_z
1	4^k	-3^k	-12^k

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
        3/D membrane

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     8,1,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     n

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     n

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     n

 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     n

    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE X-, Y-, Z-COORD., NODE TEMP., BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN X- ,Y-, Z-DIR.,
 >> AND COORD. TYPE (0 FOR RECTANGULAR, 1 FOR CYLINDRICAL)
     1:-  30,50,40,0,0,0,0
     2:-  30,30,40
     3:-  50,30,40
     4:-  50,50,40
     5:-  40,80,0,0,1,1,1
     6:-  0,40,0,0,1,1,1
     7:-  40,0,0,0,1,1,1
     8:-  80,40,0,0,1,1,1

    *** ELEMENT DATA

 >> ENTER ELEMENT TYPE NUMBER
 >> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
 >> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
 >> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
 >> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
 >> VARIAB E-NUMBER-NODES THICK SHELL =8 ;
 >> PIPE ELEMENT =12)
     3
```

```
>> IS 2-D (QUADRILATERAL) ELEMENT MESH
>> GENERATION TO BE PERFORMED? (ENTER Y OR N)
   n

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
   3,5,1
   6,8,1


  *** PLANE STRESS MEMBRANE ELEMENTS

>> ENTER NUMBER OF ELEMENTS, AND NUMBER OF DIFF. MATERIALS
   9,1

   >> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> WEIGHT & MASS DENSITIES
   1:-  30000, .3, .000283

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODES I, J, K, L(FOR TRIANGULAR ELEM. L=K), MATERIAL I.D. NO.,
>> NSPRT (SEE NOTE), ELEMENT THICKNESS (FOR PLANE STRESS ONLY)
>>  (NSPRT= 0  FOR STRESS OUTPUT AT ELEMENT CENTER
>>         1  FOR NO STRESS OUTPUT
>>         8  FOR STRESS AT CENTER & MIDPOINT OF SIDE I-L
>>        20  FOR STRESS AT CENTER & MIDPOINT OF ALL SIDES)
   1:-  1,2,3,4,1,8,.25
   2:-  4,5,1,1,,,,
   3:-  1,6,2,2,,,,
   6:-  1,5,6,6,,,,
   9:-  4,8,5,5,,,,

  ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
   1

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), AND LOADS IN X-, Y-, Z-DIR.
   1, 4, -3, -12

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
   membrane
```

```
 >> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
 >> $LIST MEMBRANE                 ** FOR LISTING OF DATA **
 >> $RUN CENA:MSAP 5=MEMBRANE      - FOR STRUCTURAL ANALYSIS
 >> $RUN CENA:MSAPLOT              - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED

#list membrane
>      1          3/D MEMBRANE
>      2          8    1    1    0    0    0
>      3          1    0    0    0   -1   -1   -1     30.000    50.000    40.000    0      0.0
>      4          2    0    0    0   -1   -1   -1     30.000    30.000    40.000    0      0.0
>      5          3    0    0    0   -1   -1   -1     50.000    30.000    40.000    0      0.0
>      6          4    0    0    0   -1   -1   -1     50.000    50.000    40.000    0      0.0
>      7          5    1    1    1   -1   -1   -1     40.000    80.000     0.0      0      0.0
>      8          6    1    1    1   -1   -1   -1      0.0      40.000     0.0      0      0.0
>      9          7    1    1    1   -1   -1   -1     40.000     0.0       0.0      0      0.0
>     10          8    1    1    1   -1   -1   -1     80.000    40.000     0.0      0      0.0
>     11          3    9    1    0    0
>     12          1    1 2.830E-04 0.0       0.0
>     13                 3.000E+04 3.000E+04 3.000E+04 3.000E-01 3.000E-01 3.000E-01 1.154E+04
>     14
>     15             0.0       0.0       0.0       0.0
>     16             0.0       0.0       0.0       0.0
>     17             0.0       0.0       0.0       0.0
>     18             0.0       0.0       0.0       0.0
>     19          1    1    2    3    4    1   0.0                 8    0   0.25000
>     20          2    4    5    1    1    1   0.0                 8    0   0.25000
>     21          3    1    6    2    2    1   0.0                 8    1   0.25000
>     22          5    3    8    4    4    1   0.0                 8    1   0.25000
>     23          6    1    5    6    6    1   0.0                 8    1   0.25000
>     24          8    3    7    8    8    1   0.0                 8    1   0.25000
>     25          9    4    8    5    5    1   0.0                 8    0   0.25000
>     26          1    1 4.000E+00-3.000E+00-1.200E+01 0.0       0.0       0.0
>     27
>     28             0.0       0.0       0.0       0.0
#END OF FILE
                         1         2         3         4         5         6         7         8
 column        12345678901234567890123456789012345678901234567890123456789012345678901234567890
```


REGION I
REGION II
NODES= 8
ELEM.= 9
TYPES= 1
LOAD = 0
C-CALCOMP DWG
H-HIDDEN LINE(D-RESET)
K-SAVE REGION I IN II
M-CALL MTS
N-NEW FILE
P-PERSPECT....(A-COORD)
R-ROTATE.....(I-RESET)
S-SECTION....(F-RESET)
T-NEW MAGNIF
U-NEW LOAD
X-DELETE MENU(Y-RESET)
3/D MEMBRANE

EXAMPLE: 6

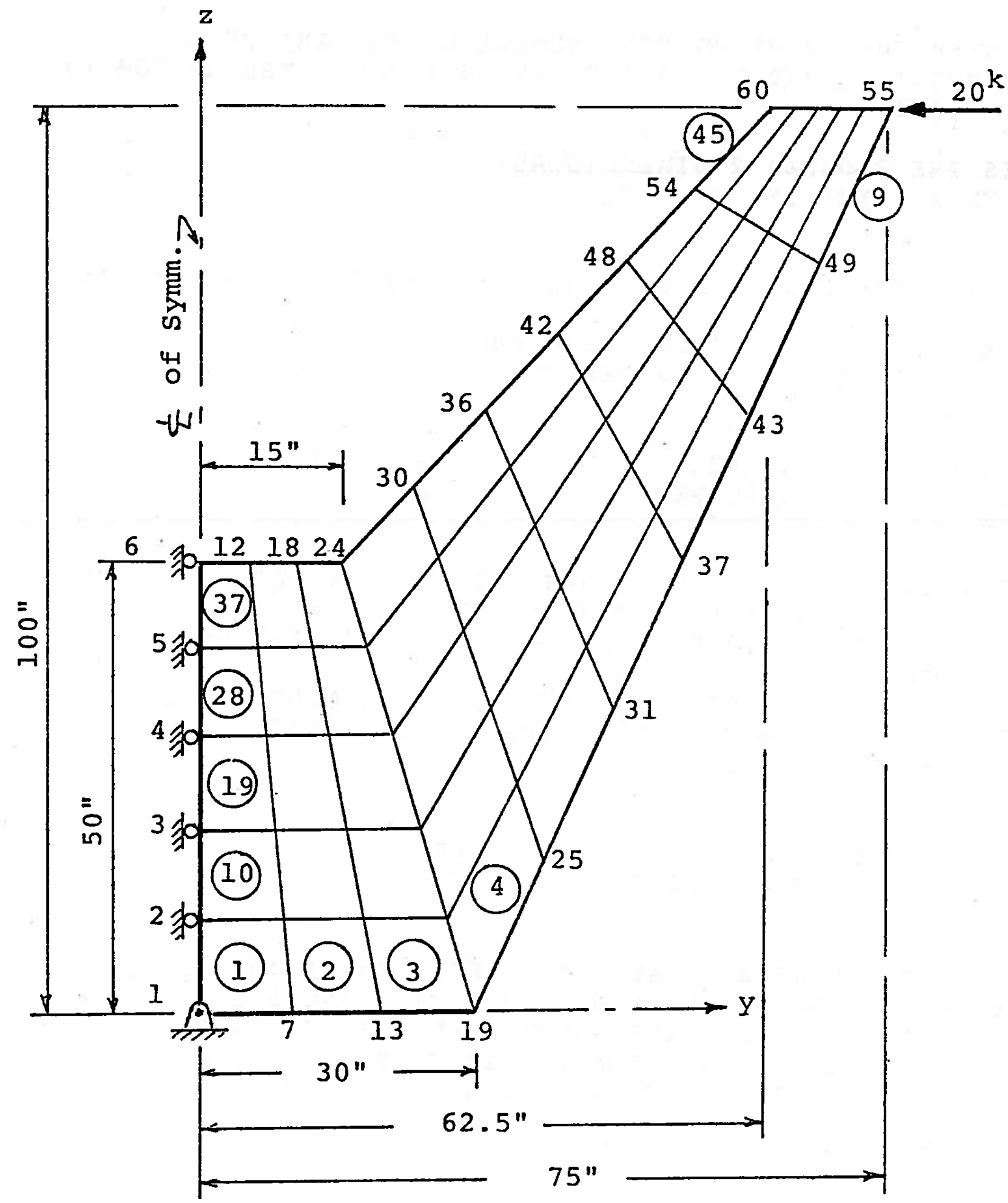
Open Web Frame

Element Type

Plane Stress (4)

1" - thick steel plate

E = 30,000 ksi & ν = .3



```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
    0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
      open web frame

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
    60,1,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
    y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
    n

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
    y

 >> FOR EACH 2 NODAL POINT MESH GENERATION SET
 >> ENTER FOLLOWING SIX ITEMS PER LINE,
 >> GLOBAL NODES I,J,K,L, AND NODAL NUMBER INCREMENTS
 >> KN FOR I-J AND FOR I-L.
 >>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    19,55,60,24,6,1


 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
    y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    2,6,1
    7,12,1
    13,18,1
```

*** JOINT DATA

```
>> FOR EACH JOINT ENTER FOLLOWING PER LINE
>> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
>> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y- IR., IN Z-DIR.
  1:-  0,0,0,1,1
  2:-  0,10,0,1,0
  6:-  0,50,0,1,0
  7:-  10,0
 12:-  0,50
 13:-  20,0
 18:-  10,50
 19:-  30,0
 55:-  75,100
 24:-  15,50
 60:-  62.5,100
```

*** ELEMENT DATA

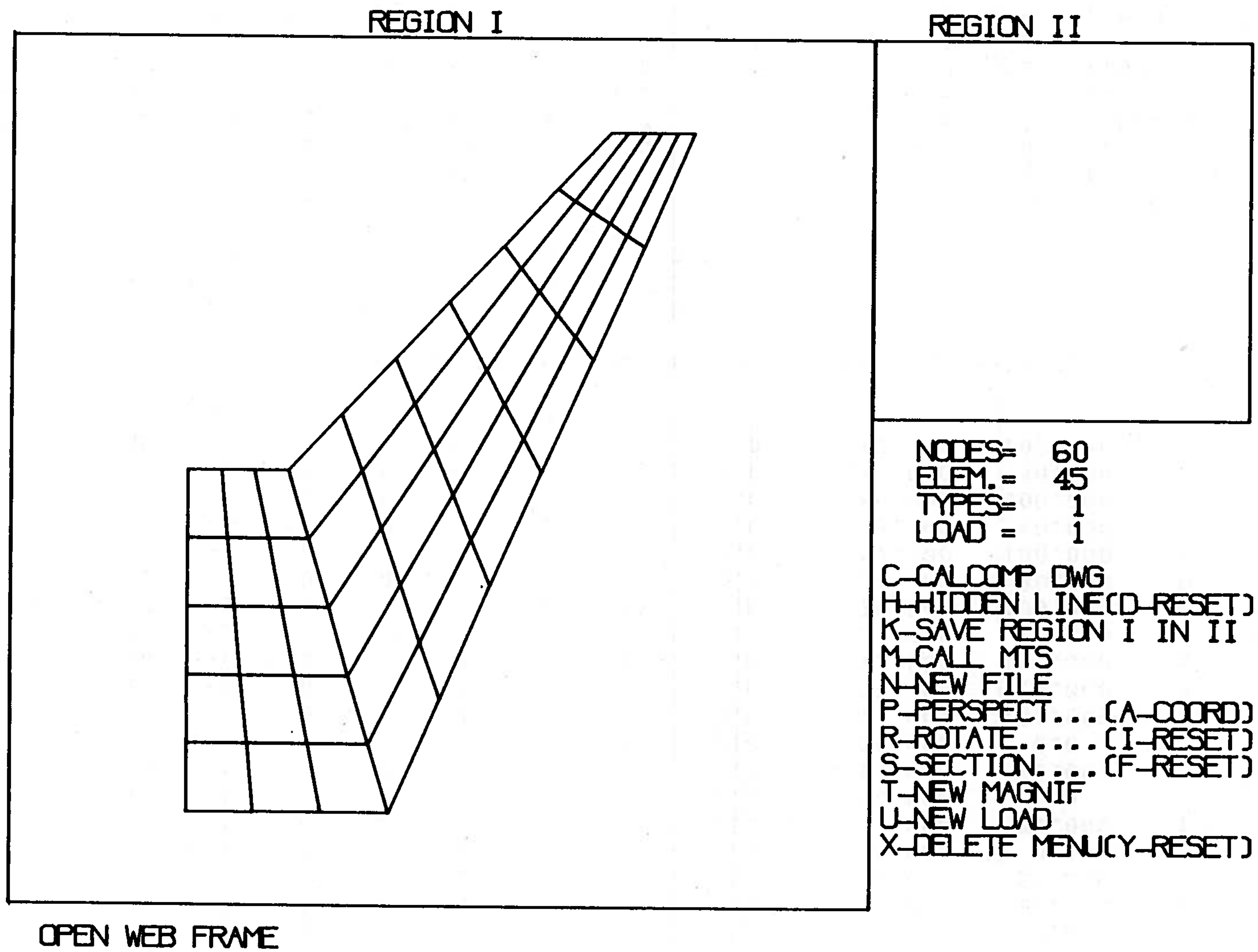
```
>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
   4

>> IS 2-D (QUADRILATERAL) ELEMENT MESH
>> GENERATION TO BE PERFORMED? (ENTER Y OR N)
   y

>> FOR EVERY 2-D ELEMENT MESH GENERATION
>> ENTER FOLLOWING EIGHT ITEMS PER LINE,
>> THE FIRST AND LAST ELEMENT NUMBERS (IN INCREASING
>> SEQ.), GLOBAL NODES I,J,K,L, AND NODAL NUMBER
>> INCREMENTS KN FOR I-J AND FOR I-L.
>>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
   1,45,1,55,60,6,6,1


>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   n
```

*** 2-DIMENSIONAL FINITE ELEMENTS

```
>> ENTER NUMBER OF ELEMENTS, AND NUMBER OF DIFF. MATERIALS
   45,1
```

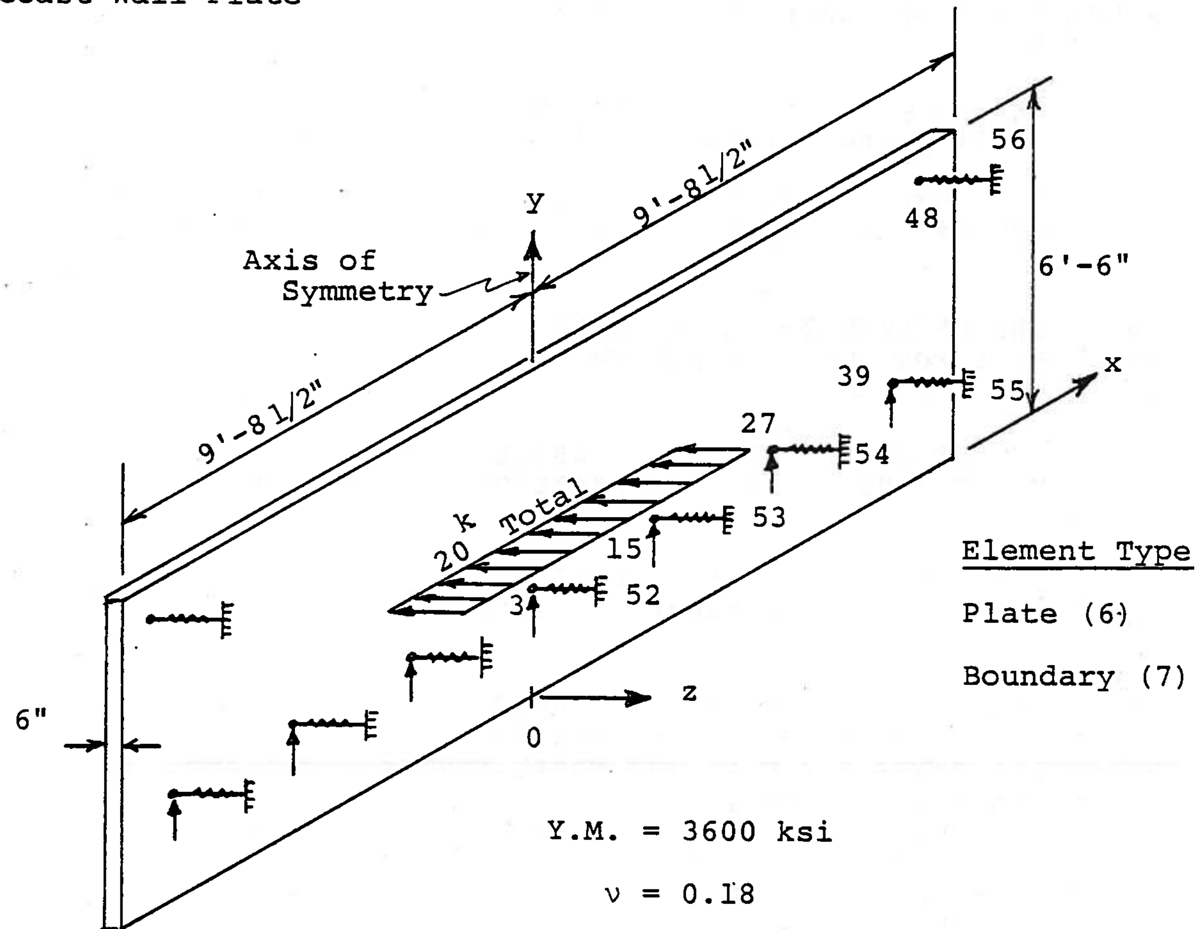
```
>> ENTER ANALYSIS TYPE NUMBER
>> ( 0 = AXISYMMETRIC, 1 = PLANE STRAIN, 2 = PLANE STRESS)
    2

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> WEIGHT & MASS DENSITIES
    1:-  30000, .3, .000283

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODES I, J, K, L(FOR TRIANGULAR ELEM. L K), MATERIAL I.D. NO.,
>> NSPRT (SEE NOTE), ELEMENT THICKNESS (FOR PLANE STRESS ONLY)
>>  (NSPRT= 0  FOR STRESS OUTPUT AT ELEMENT CENTER
>>          1  FOR NO STRESS OUTPUT
>>          8  FOR STRESS AT CENTER & MIDPOINT OF SID  I-L
>>         20  FOR STRESS AT CENTER & MIDPOINT OF ALL SIDES)
    1:-  1,7,8,2,1,8,1

   ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
    1

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), AND LOADS IN Y-, Z-DIR.
    55, -20, 0

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
    -web

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST -WEB                  ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=-WEB       - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT           - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

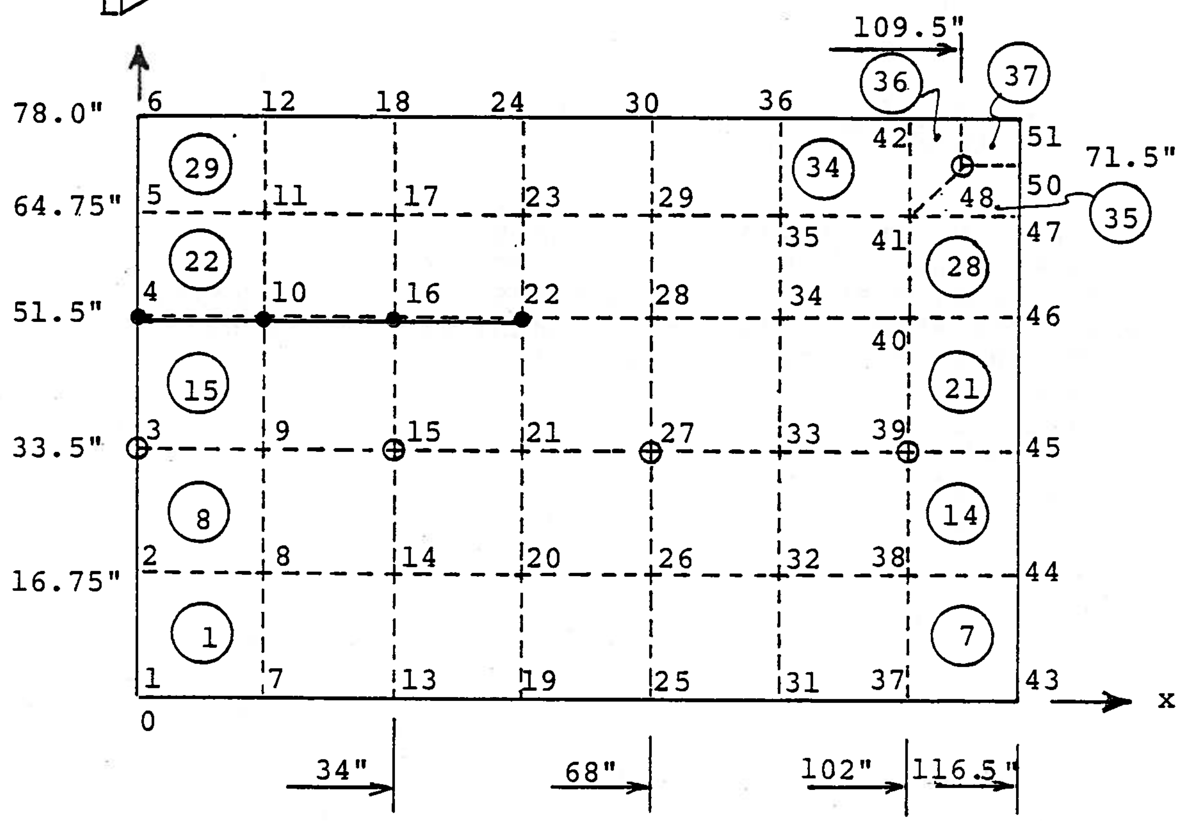
```
#list -web
>      1         OPEN WEB FRAME
>      2          60    1    1    0    0    0
>      3           1   -1    1    1   -1   -1   -1      0.0       0.0       0.0      0       0.0
>      4           2   -1    1    0   -1   -1   -1      0.0       0.0      10.000    0       0.0
>      5           6   -1    1    0   -1   -1   -1      0.0       0.0      50.000    1       0.0
>      6           7   -1    0    0   -1   -1   -1      0.0      10.000     0.0      0       0.0
>      7          12   -1    0    0   -1   -1   -1      0.0       5.000    50.000    1       0.0
>      8          13   -1    0    0   -1   -1   -1      0.0      20.000     0.0      0       0.0
>      9          18   -1    0    0   -1   -1   -1      0.0      10.000    50.000    1       0.0
>     10          19   -1    0    0   -1   -1   -1      0.0      30.000     0.0      0       0.
>     11          55   -1    0    0   -1   -1   -1      0.0      75.000   100.000    6       0.0
>     12          20   -1    0    0   -1   -1   -1      0.0      27.000    10.000    0       0.0
>     13          56   -1    0    0   -1   -1   -1      0.0      72.500   100.000    6       0.0
>     14          21   -1    0    0   -1   -1   -1      0.0      24.000    20.000    0       0.0
>     15          57   -1    0    0   -1   -1   -1      0.0      70.000   100.000    6       0.0
>     16          22   -1    0    0   -1   -1   -1      0.0      21.000    30.000    0       0.0
>     17          58   -1    0    0   -1   -1   -1      0.0      67.500   100.000    6       0.0
>     18          23   -1    0    0   -1   -1   -1      0.0      18.000    40.000    0       0.0
>     19          59   -1    0    0   -1   -1   -1      0.0      65.000   100.000    6       0.0
>     20          24   -1    0    0   -1   -1   -1      0.0      15.000    50.000    0       0.0
>     21          60   -1    0    0   -1   -1   -1      0.0      62.500   100.000    6       0.0
>     22           4   45    1    0    2    0
>     23           1    1 2.830E-04 0.0       0.0
>     24                  3.000E+04 3.000E+04 3.000E+04 3.000E-01 3.000E-01 3.000E-01 1.154E+04
>     25
>     26              0.0       0.0       0.0       0.0
>     27              0.0       0.0       0.0       0.0
>     28              0.0       0.0       0.0       0.0
>     29              0.0       0.0       0.0       0.0
>     30           1    1    7    8    2    1   0.0                      8    6   1.00000
>     31           9   49   55   56   50    1   0.0                      8    6   1.00000
>     32          10    2    8    9    3    1   0.0                      8    6   1.00000
>     33          18   50   56   57   51    1   0.0                      8    6   1.00000
>     34          19    3    9   10    4    1   0.0                      8    6   1.00000
>     35          27   51   57   58   52    1   0.0                      8    6   1.00000
>     36          28    4   10   11    5    1   0.0                      8    6   1.00000
>     37          36   52   58   59   53    1   0.0                      8    6   1.00000
>     38          37    5   11   12    6    1   0.0                      8    6   1.00000
>     39          45   53   59   60   54    1   0.0                      8    6   1.00000
>     40          55    1 0.0      -2.000E+01 0.0       0.0       0.0       0.0
>     41
>     42              0.0       0.0       0.0       0.0
#END OF FILE
                    1         2         3         4         5         6         7         8
  column   12345678901234567890123456789012345678901234567890123456789012345678901234567890
```


REGION I
REGION II
NODES= 60
ELEM.= 45
TYPES= 1
LOAD = 1
C-CALCOMP DWG
H-HIDDEN LINE(D-RESET)
K-SAVE REGION I IN II
M-CALL MTS
N-NEW FILE
P-PERSPECT....(A-COORD)
R-ROTATE.....(I-RESET)
S-SECTION....(F-RESET)
T-NEW MAGNIF
U-NEW LOAD
X-DELETE MENU(Y-RESET)
OPEN WEB FRAME

EXAMPLE: 7

Precast Wall-Plate





```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
        wall plate (plate-shell)

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     56,2,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     n

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     y

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH 2 NODAL POINT MESH GENERATION SET
 >> ENTER FOLLOWING SIX ITEMS PER LINE,
 >> GLOBAL NODES I,J,K,L, AND NODAL NUMBER INCREMENTS
 >> KN FOR I-J AND FOR I-L.
 >>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     7,37,38,8,6,1
     10,40,42,12,6,1


 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     4,6,1
     9,33,12
     15,39,12
     43,45,1
```

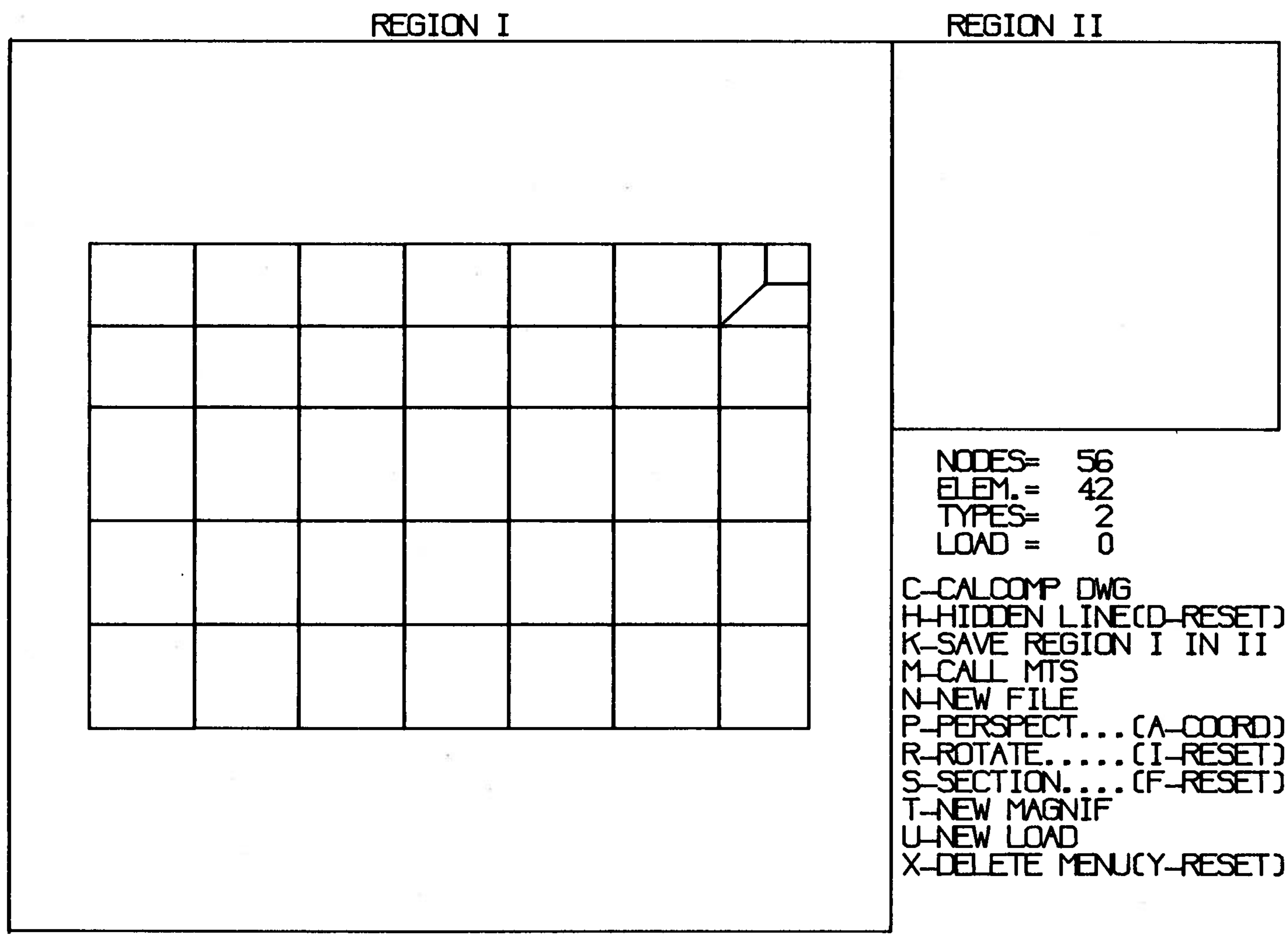
```
>> FOR EVERY 2-D ELEMENT MESH GENERATION
>> ENTER FOLLOWING EIGHT ITEMS PER LINE,
>> THE FIRST AND LAST ELEMENT NUMBERS (IN INCREASING
>> SEQ.), GLOBAL NODES I,J,K,L, AND NODAL NUMBER
>> INCREMENTS KN FOR I-J AND FOR I-L.
>>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,28,1,43,47,5,6,1


>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    29,34,6


   *** PLATE OR THIN SHELL ELEMENTS

>> ENTER NUMBER OF ELEMENTS, AND NUMBER OF DIFF. MATERIALS
    37,1

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> MASS DENSITY
    1:-  3600, .18

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODES I, J, K, L(FOR TRIANGULAR ELEM. L=0), MATERIAL I.D. NO.,
>> AND ELEMENT THICKNESS
    1:-  1,7,8,2,1,6
   29:-  5,11,12,6,1,6
   35:-  41,47,50,48,1,6
   36:-  42,41,48,49,1,6
   37:-  48,50,51,49,1,6

>> ENTER ELEMENT TYPE NUMBER
    7

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    n

   *** BOUNDARY ELEMENTS

>> ENTER NUMBER OF ELEMENTS
    5
```

```
*** JOINT DATA

>> FOR EACH JOINT ENTER FOLLOWING PER LINE
>> THE X-, Y-, Z-COORD., TEMP., BOUND. COND. CODES (0 FOR
>> FREE, 1 FOR CONSTRND) IN X-, Y-, Z-DIR., X-, Y-, Z-ROT.,
>> AND COORD. TYPE (0 FOR RECTANGULAR, 1 FOR CYLINDRICAL)
  1:-  0,0,0,0,0,1,0,0,0,1,-1
  2:-  0,16.75,0,0,1,0,0,0,1
  3:-  0,33.5,0,0,1,1,0,0,1
  4:-  0,51.5,0,0,1,0,0,0,1
  6:-  0,78,0,0,1,0,0,0,1
  7:-  17,0
 37:-  102,0
  8:-  17,16.75
 38:-  102,16.75
  9:-  17,33.5
 33:-  85,33.5
 10:-  17,51.5
 40:-  102,51.5
 12:-  17,78
 42:-  102,78
 15:-  34,33.5,0,0,0,1
 39:-  102,33.5,0,0,0,1
 43:-  116.5,0
 45:-  116.5,33.5
 46:-  116.5,51.5
 47:-  116.5,64.75
 48:-  109.5,71.5,0,0,0,1
 49:-  109.5,78
 50:-  116.5,71.5
 51:-  116.5,78
 52:-  0,33.5,5,0,-1,-1,-1,-1,-1,-1
 53:-  34,33.5,5
 54:-  68,33.5,5
 55:-  102,33.5,5
 56:-  109.5,71.5,5

*** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
  6

>> IS 2-D (QUADRILATERAL) ELEMENT MESH
>> GENERATION TO BE PERFORMED? (ENTER Y OR N)
  y
```

```
 >> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
 >> NODE N (WHERE THE ELEMENT IS PLACED, IN ASCENDING ORDER),
 >> NODE I (THIS DEFINES THE DIRECTION OF THE ELEMENT),
 >> DISPLACEMENT CODE(0 OR 1), ROTATION CODE(0 OR 1),
 >> SPECIFIED DISPLACEMENT, SPECIFIED ROTATION, AND
 >> SPRING STIFFNESS (SET TO 1.0*E10 IF LEFT BLANK)
     3,52,1
     15,53,1
     27,54,1
     39,55,1
     48,56,1

    ***CONCENTRATED LOAD DATA

 >> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
     4

 >> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
 >> JOINT NO.(IN INCREASING SEQ.), LOADS IN X-, Y-, Z-DIR.,
 >> AND MOMENTS ABOUT X-X, Y-Y, Z-Z
     4,0,0,-1.67
     10,0,0,-3.33
     16,0,0,-3.33
     22,0,0,-1.67

 >> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
 >> ACCORDING TO THE INPUT FORMAT OF MSAP
 >> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
     shell

 >> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
 >> $LIST SHELL                    ** FOR LISTING OF DATA **
 >> $RUN CENA:MSAP 5=SHELL         - FOR STRUCTURAL ANALYSIS
 >> $RUN CENA:MSAPLOT              - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list shell
>     1          WALL PLATE (PLATE-SHELL)
>     2          56    2    1    0    0    0
>     3           1    0    1    0    0    0    1     0.0       0.0      0.0      0    0.0
>     4           2    1    0    0    0    1    0     0.0      16.750    0.0      0    0.0
>     5           3    1    1    0    0    1    0     0.0      33.500    0.0      0    0.0
>     6           4    1    0    0    0    1    0     0.0      51.500    0.0      0    0.0
>     7           6    1    0    0    0    1    0     0.0      78.000    0.0      1    0.0
>     8           7    0    0    0    0    0    0    17.000     0.0      0.0      0    0.0
>     9          37    0    0    0    0    0    0   102.000     0.0      0.0      6    0.0
>    10           8    0    0    0    0    0    0    17.000    16.750    0.0      0    0.0
>    11          38    0    0    0    0    0    0   102.000    16.750    0.0      6    0.0
>    12           9    0    0    0    0    0    0    17.000    33.500    0.0      0    0.0
>    13          33    0    0    0    0    0    0    85.000    33.500    0.0     12    0.0
>    14          10    0    0    0    0    0    0    17.000    51.500    0.0      0    0.0
>    15          40    0    0    0    0    0    0   102.000    51.500    0.0      6    0.0
>    16          12    0    0    0    0    0    0    17.000    78.000    0.0      0    0.0
>    17          42    0    0    0    0    0    0   102.000    78.000    0.0      6    0.0
>    18          11    0    0    0    0    0    0    17.000    64.750    0.0      0    0.0
>    19          41    0    0    0    0    0    0   102.000    64.750    0.0      6    0.0
>    20          15    0    1    0    0    0    0    34.000    33.500    0.0      0    0.0
>    21          39    0    1    0    0    0    0   102.000    33.500    0.0     12    0.0
>    22          43    0    0    0    0    0    0   116.500     0.0      0.0      0    0.0
>    23          45    0    0    0    0    0    0   116.500    33.500    0.0      1    0.0
>    24          46    0    0    0    0    0    0   116.500    51.500    0.0      0    0.0
>    25          47    0    0    0    0    0    0   116.500    64.750    0.0      0    0.0
>    26          48    0    1    0    0    0    0   109.500    71.500    0.0      0    0.0
>    27          49    0    0    0    0    0    0   109.500    78.000    0.0      0    0.0
>    28          50    0    0    0    0    0    0   116.500    71.500    0.0      0    0.0
>    29          51    0    0    0    0    0    0   116.500    78.000    0.0      0    0.0
>    30          52   -1   -1   -1   -1   -1   -1     0.0      33.500    5.000    0    0.0
>    31          53    0    0    0    0    0    0    34.000    33.500    5.000    0    0.0
>    32          54    0    0    0    0    0    0    68.000    33.500    5.000    0    0.0
>    33          55    0    0    0    0    0    0   102.000    33.500    5.000    0    0.0
>    34          56    0    0    0    0    0    0   109.500    71.500    5.000    0    0.0

                          1         2         3         4         5         6         7         8
   column        12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

```
>    35         6   37    1
>    36              1                     0.0       0.0       0.0       0.0
>    37      3.721E+03 6.697E+02 0.0       3.721E+03 0.0       1.525E+03
>    38            0.0       0.0       0.0       0.0
>    39            0.0       0.0       0.0       0.0
>    40            0.0       0.0       0.0       0.0
>    41            0.0       0.0       0.0       0.0
>    42            0.0       0.0       0.0       0.0
>    43         1    1    7    8    2         1    6   6.00000              0.0       0.0
>    44         7   37   43   44   38         1    6   6.00000              0.0       0.0
>    45         8    2    8    9    3         1    6   6.00000              0.0       0.0
>    46        14   38   44   45   39         1    6   6.00000              0.0       0.0
>    47        15    3    9   10    4         1    6   6.00000              0.0       0.0
>    48        21   39   45   46   40         1    6   6.00000              0.0       0.0
>    49        22    4   10   11    5         1    6   6.00000              0.0       0.0
>    50        28   40   46   47   41         1    6   6.00000              0.0       0.0
>    51        29    5   11   12    6         1    6   6.00000              0.0       0.0
>    52        34   35   41   42   36         1    6   6.00000              0.0       0.0
>    53        35   41   47   50   48         1    0   6.00000              0.0       0.0
>    54        36   42   41   48   49         1    0   6.00000              0.0       0.0
>    55        37   48   50   51   49         1    0   6.00000              0.0       0.0
>    56         7    5
>    57            1.0
>    58         3   52                   1    0    0 0.0       0.0
>    59        15   53                   1    0    0 0.0       0.0
>    60        27   54                   1    0    0 0.0       0.0
>    61        39   55                   1    0    0 0.0       0.0
>    62        48   56                   1    0    0 0.0       0.0
>    63         4    1 0.0       0.0      -1.670E+00 0.0       0.0       0.0
>    64        10    1 0.0       0.0      -3.330E+00 0.0       0.0       0.0
>    65        16    1 0.0       0.0      -3.330E+00 0.0       0.0       0.0
>    66        22    1 0.0       0.0      -1.670E+00 0.0       0.0       0.0
>    67
>    68            1.0       0.0       0.0       0.0
#END OF FILE
                     1         2         3         4         5         6         7         8
  column    12345678901234567890123456789012345678901234567890123456789012345678901234567890
```



WALL PLATE (PLATE-SHELL)

EXAMPLE: 8

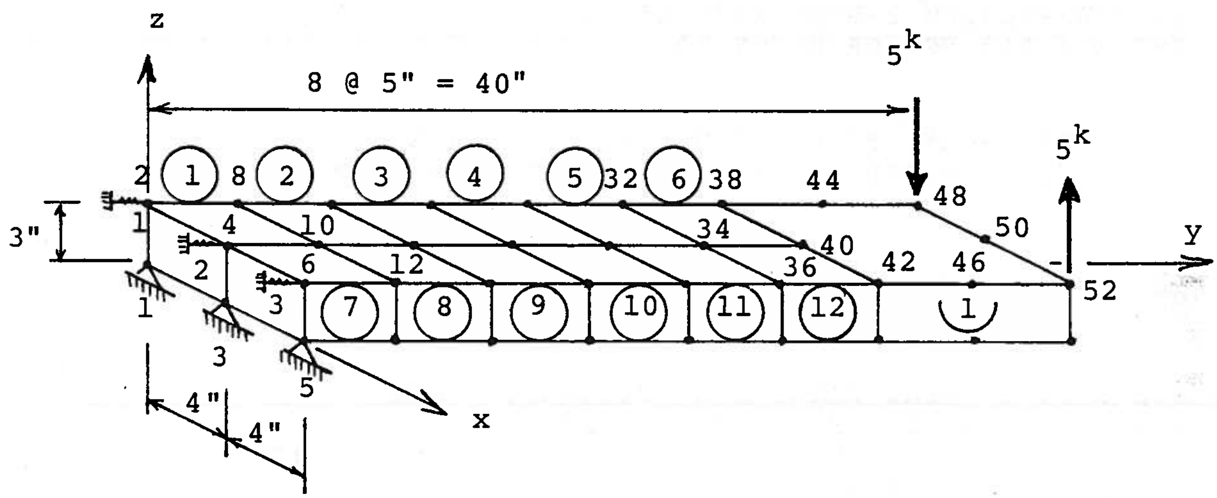
Solid Flat Bar

Element Type

8-Node Brick (5)

Boundary (7)

Thick Shell (8)



Material Properties:

Y. Modulus = 10,000 ksi

Poisson's Ratio = 0.3

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
       3/D flat bar

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     52,3,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     n

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     n

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH 2 NODAL POINT M SH GENERATION SET
 >> ENTER FOLLOWING SIX ITEMS PER LINE,
 >> GLOBAL NODES I,J,K,L, AND NODAL NUMBER INCREMENTS
 >> KN FOR I-J AND FOR I-L.
 >>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     2,6,42,38,2,6
     7,11,41,37,2,6
     47,48,52,51,1,2


 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     1,5,2
```

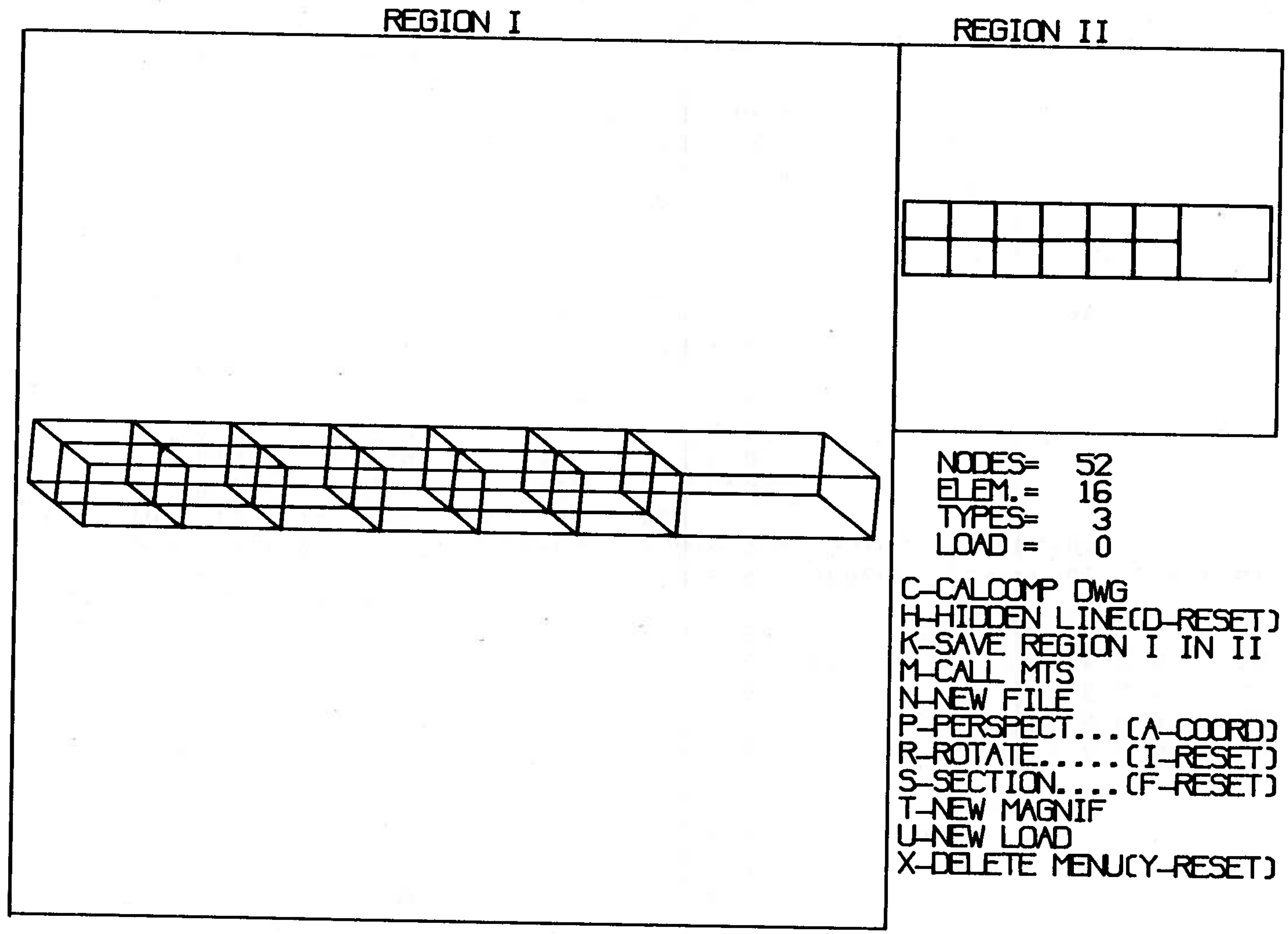
```
*** JOINT DATA

>> FOR EACH JOINT ENTER FOLLOWING PER LINE
>> THE X-, Y-, Z-COORD., NODE TEMP., BOUNDARY COND. CODES
>> (0 FOR FREE, 1 FOR CONSTRAINED) IN X- ,Y-, Z-DIR.,
>> AND COORD. TYPE (0 FOR RECTANGULAR, 1 FOR CYLINDRICAL)
    1:-  0,0,0,0,1,1,1
    5:-  8,0,0,0,1,1,1
    2:-  0,0,3
    6:-  8,0,3
   38:-  0,30,3
   42:-  8,30,3
    7:-  0,5,0
   11:-  8,5,0
   37:-  0,30,0
   41:-  8,30,0
   43:-  0,35,0
   44:-  0,35,3
   45:-  8,35,0
   46:-  8,35,3
   47:-  0,40,0
   48:-  0,40,3
   51:-  8,40,0
   52:-  8,40,3

*** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1 ; BEAM =2 ; ME BRANE(PLANE STRESS) =3 ;
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
>> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
>> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
>> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
>> PIPE ELEMENT =12)
    5

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,6,6
    7,12,6


*** 3-D SOLID ELEMENTS

>> ENTER NUMBER OF ELEMENTS, AND NUMBER OF DIFF. MATERIALS
    12,1
```

```
>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> M DULUS OF ELASTICITY, POISSONS RATIO, AND
>> WEIGHT DENSITY
   1:-  10000, .3

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODE NOS.(CORRESPONDING TO ELEM. NODES 1,2,3,4,5,6,7,8),
>> INTERGRATION ORDER (2 FOR RECTANGULAR ELEM., OR 3 FOR SKEWED ELEM.),
>> MATERIAL I.D. NO., AND TWO FACE NUMBERS
>> (FOR STRESS OUTPUT FROM FOLLOWING NOS. 0,1,2,3,4,5, OR 6)
   1:-  3,9,7,1,4,10,8,2,2,1,6,0
   7:-  5,11,9,3,6,12,10,4,2,1,6,0

>> ENTER ELEMENT TYPE NUMBER
   8

...ENTER NUMBER OF ELEMENTS, NUMBER OF DIF. MATERIALS,
...MAX NUMBER OF TEMP. POINTS FOR ANY MATERIAL,
...NUMBER OF DIFFERENT DISTRIBUTED LOAD SETS,
...MAX NUMBER OF NODES TO DESCRIBE AN ELEMENT, AND
...NUMBER OF SETS OF DATA REQUESTING STRESS OUTPUT
...AT VARIOUS LOCATIONS.
... (INTEGERS ON ONE LINE SEPARATED BY COMMA)
   ? 1,1,1,0,16,1

...FOR EACH MATL ENTER NUMBER OF DIFFERENT TEMPERATURES
...AT WHICH MATL PROPERTIES ARE SPECIFIED, AND
...WEIGHT & MASS DENSITIES
   1:  1, .000283

...FOR EACH DIF. TEMP FOR THIS MATERIAL ENTER
...TEMPERATURE, MODULUS OF ELASTICITY, POISSONS RATIO,
...SHEAR MODULUS AND THERMAL EXPANSION COEFFICIENT.
   1:  72, 30000, .29, 12000, .00005

...SPECIFY OUTPUT LOCATIONS FOR EACH OUTPUT SET
...(LESS THAN OR EQUAL TO 27)
   1:  10,12,14,16,21,22,23

...ARE THERE GRAVITY, THERMAL, OR DISTRIBUTED LOADS
...APPLIED? (ENTER Y OR N)
   ? n

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   ? n

...FOR EACH ELEMENT ENTER
...NUMBER OF NODES DESCRIBING THE ELEMENT DISPLACEMENTS,
...NUMBER OF NODES TO DESCRIBE ELEMENT GEOMETRY,
...MATERIAL ID NUMBER, STRESS OUTPUT SET NUMBER,
...STRESS FREE TEMPERATURE, AND PRESSURE SET NUMBER
   1:  16,8,1,1,72
```

```
      >>LIST DESCRIBING NODE NUMBERS
      ? 52,48,38,42,51,47,37,41,50,44,40,46,49,43,39,45

 >> ENTER ELEMENT TYPE NUMBER
     7

 >> IS LINEAR ELEMENT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FO  EVERY LINEAR ELEMENT MESH GENERATION
 >> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
 >> ELEMENT NUMBERS (IN INCREASING SEQ.)
 >> AND THE MESH GENERATION INCREMENT K
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     1,3,2

    *** BOUNDARY ELEMENTS

 >> ENTER NUMBER OF ELEMENTS
     3

 >> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
 >> NODE N (WHERE THE ELEMENT IS PLACED, IN ASCENDING ORDER),
 >> NODE I (THIS DEFINES THE DIRECTION OF THE ELEMENT),
 >> DISPLACEMENT CODE(0 OR 1), ROTATION CODE(0 OR 1),
 >> SPECIFIED DISPLACEMENT, SPECIFIED ROTATION, AND
 >> SPRING STIFFNESS (SET TO 1.0*E10 IF LEFT BLANK)
     1:-  2,8,1

    ***CONCENTRATED LOAD DATA

 >> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
     2

 >> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
 >> JOINT NO.(IN INCREASING SEQ.), AND LOADS IN X-, Y-, Z-DIR.
     48, 0,0,-5
     52, 0,0,5

 >> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
 >> ACCORDING TO THE INPUT FORMAT OF MSAP
 >> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
     threed

 >> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
 >> $LIST THREED                   ** FOR LISTING OF DATA **
 >> $RUN CENA:MSAP 5=THREED        - FOR STRUCTURAL ANALYSIS
 >> $RUN CENA:MSAPLOT              - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list threed
>    1       3/D FLAT BAR
>    2        52   3   1   0    0    0
>    3         1   1   1   1   -1   -1   -1    0.0      0.0      0.0     0    0.0
>    4         5   1   1   1   -1   -1   -1    8.000    0.0      0.0     2    0.0
>    5         2   0   0   0   -1   -1   -1    0.0      0.0      3.000   0    0.0
>    6         6   0   0   0   -1   -1   -1    8.000    0.0      3.000   2    0.0
>    7        38   0   0   0   -1   -1   -1    0.0     30.000    3.000   0    0.0
>    8        42   0   0   0   -1   -1   -1    8.000   30.000    3.000   2    0.0
>    9         8   0   0   0   -1   -1   -1    0.0      5.000    3.000   0    0.0
>   10        12   0   0   0   -1   -1   -1    8.000    5.000    3.000   2    0.0
>   11        14   0   0   0   -1   -1   -1    0.0     10.000    3.000   0    0.0
>   12        18   0   0   0   -1   -1   -1    8.000   10.000    3.000   2    0.0
>   13        20   0   0   0   -1   -1   -1    0.0     15.000    3.000   0    0.0
>   14        24   0   0   0   -1   -1   -1    8.000   15.000    3.000   2    0.0
    15        26   0   0   0   -1   -1   -1    0.0     20.000    3.000   0    0.0
>   16        30   0   0   0   -1   -1   -1    8.000   20.000    3.000   2    0.0
>   17        32   0   0   0   -1   -1   -1    0.0     25.000    3.000   0    0.0
>   18        36   0   0   0   -1   -1   -1    8.000   25.000    3.000   2    0.0
>   19         7   0   0   0   -1   -1   -1    0.0      5.000    0.0     0    0.0
>   20        11   0   0   0   -1   -1   -1    8.000    5.000    0.0     2    0.0
>   21        37   0   0   0   -1   -1   -1    0.0     30.000    0.0     0    0.0
>   22        41   0   0   0   -1   -1   -1    8.000   30.000    0.0     2    0.0
>   23        13   0   0   0   -1   -1   -1    0.0     10.000    0.0     0    0.0
>   24        17   0   0   0   -1   -1   -1    8.000   10.000    0.0     2    0.0
>   25        19   0   0   0   -1   -1   -1    0.0     15.000    0.0     0    0.0
>   26        23   0   0   0   -1   -1   -1    8.000   15.000    0.0     2    0.0
>   27        25   0   0   0   -1   -1   -1    0.0     20.000    0.0     0    0.0
>   28        29   0   0   0   -1   -1   -1    8.000   20.000    0.0     2    0.0
>   29        31   0   0   0   -1   -1   -1    0.0     25.000    0.0     0    0.0
>   30        35   0   0   0   -1   -1   -1    8.000   25.000    0.0     2    0.0
>   31        43   0   0   0   -1   -1   -1    0.0     35.000    0.0     0    0.0
>   32        44   0   0   0   -1   -1   -1    0.0     35.000    3.000   0    0.0
>   33        45   0   0   0   -1   -1   -1    8.000   35.000    0.0     0    0.0
>   34        46   0   0   0   -1   -1   -1    8.000   35.000    3.000   0    0.0
>   35        47   0   0   0   -1   -1   -1    0.0     40.000    0.0     0    0.0
>   36        48   0   0   0   -1   -1   -1    0.0     40.000    3.000   1    0.0
>   37        49   0   0   0   -1   -1   -1    4.000   40.000    0.0     0    0.0
>   38        50   0   0   0   -1   -1   -1    4.000   40.000    3.000   1    0.0
>   39        51   0   0   0   -1   -1   -1    8.000   40.000    0.0     0    0.0
>   40        52   0   0   0   -1   -1   -1    8.000   40.000    3.000   1    0.0
```

```
>   41        5   12    1    0
>   42        1 1.000E+04 3.000E-01 0.0       0.0
>   43           0.0
>   44           0.0       0.0       0.0       0.0
>   45           0.0       0.0       0.0       0.0
>   46           0.0       0.0       0.0       0.0
>   47           0.0       0.0       0.0       0.0
>   48           0.0       0.0       0.0       0.0
>   49        1    3    9    7    1    4   10    8    2    2    1    6 0 0 0 060   0.0
>   50        6   33   39   37   31   34   40   38   32    2    1    6 0 0 0 060   0.0
>   51        7    5   11    9    3    6   12   10    4    2    1    6 0 0 0 060   0.0
>   52       12   35   41   39   33   36   42   40   34    2    1    6 0 0 0 060   0.0
>   53        8    1    1    1    0    0   16    1    2    2
>   54        1    1 2.830E-04 0.0
>   55    7.200E+01 3.000E+04 3.000E+04 3.000E+04 2.900E-01 2.900E-01 2.900E-01
>   56    1.200E+04 1.200E+04 1.200E+04 5.000E-05 5.000E-05 5.000E-05
>   57       10   12   14   16   21   22   23
>   58           0.0       0.0       0.0       0.0
>   59           0.0       0.0       0.0       0.0
>   60           0.0       0.0       0.0       0.0
>   61           0.0       0.0       0.0       0.0
>   62           0.0       0.0       0.0       0.0
>   63        1   16    8    1    0    1 7.200E+01    1
>   64       52   48   38   42   51   47   37   41   50   44   40   46   49   43   39   45
>   65        0    0    0    0    0
>   66        7    3
>   67           1.0
>   68        2    8                   1    0    2 0.0       0.0
>   69        6   12                   1    0    2 0.0       0.0
>   70       48    1 0.0       0.0      -5.000E+00 0.0       0.0       0.0
>   71       52    1 0.0       0.0       5.000E+00 0.0       0.0       0.0
>   72
>   73           1.0       0.0       0.0       0.0
#END OF FILE
                  1         2         3         4         5         6         7         8
  column 12345678901234567890123456789012345678901234567890123456789012345678901234567890
```



THREE-D CANTILEVER FLAT BAR

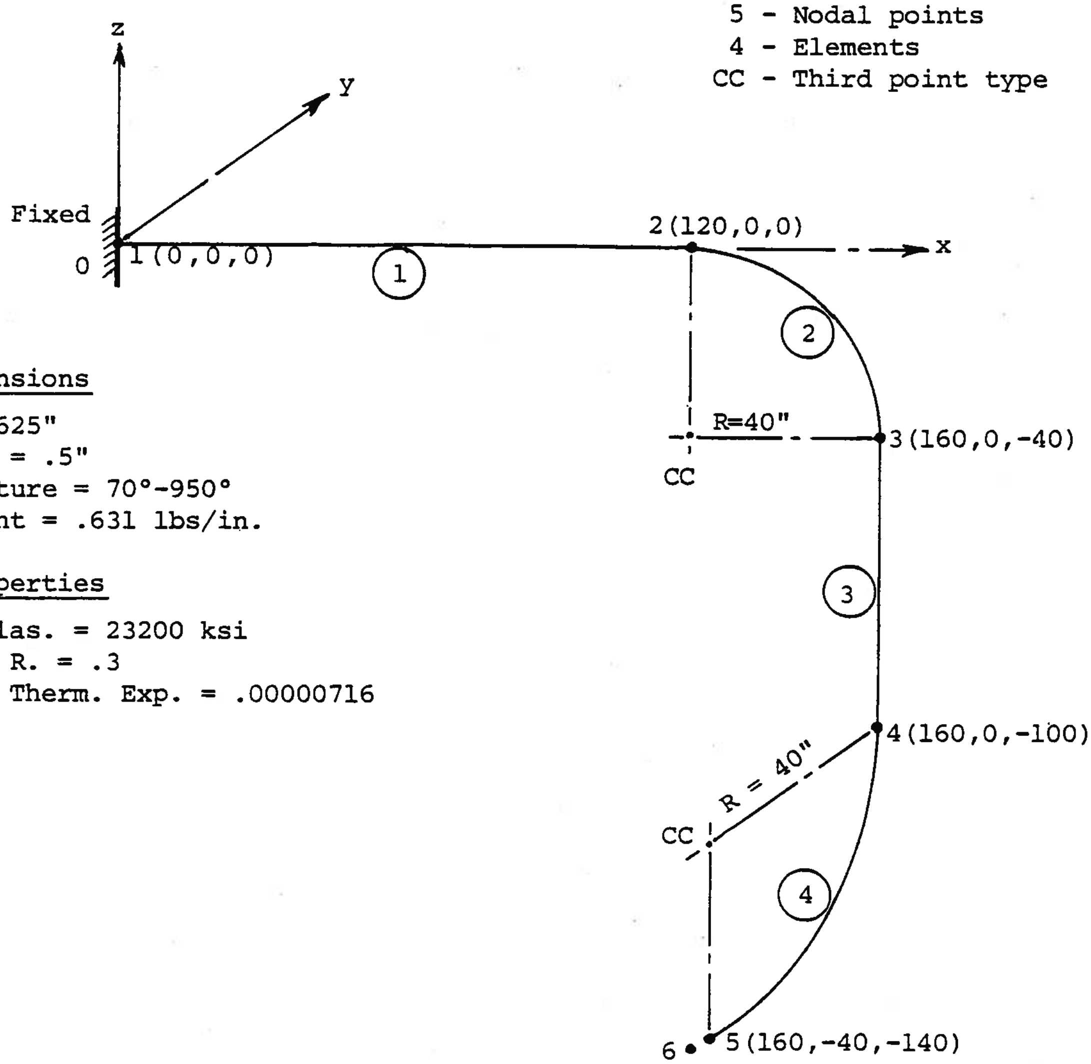
EXAMPLE: 9

3/D Pipe

Element Type

Boundary (7)
Pipe (12)

5 - Nodal points
4 - Elements
CC - Third point type



Pipe Dimensions

O.D. = 8.625"
Thickness = .5"
Δ Temperature = 70°-950°
Unit weight = .631 lbs/in.

Matl. Properties

Mod. of Elas. = 23200 ksi
Poisson's R. = .3
Coeff. of Therm. Exp. = .00000716

NOTE: N.P. 5 is constrained. Its y-displ. = 0.025".

```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER.INPUT MODE (0= INTERACTIVE, 1= DATA MODE) AND
 >> ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
 >> ON ONE LINE SEPARATED BY COMMA
     0,0

 >> ENTER PROBLEM TITLE ON ONE LINE
       3/d pipe

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     6,2,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     n

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     y

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     n

 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     n

    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE X-, Y-, Z-COORD., TEMP., BOUND. COND. CODES (0 FOR
 >> FREE, 1 FOR CONSTRND) IN X-, Y-, Z-DIR., X-, Y-, Z-ROT.,
 >> AND COORD. TYPE (0 FOR RECTANGULAR, 1 FOR CYLINDRICAL)
     1:-  0,0,0,950,1,1,1,1,1,1
     2:-  120,0,0,950
     3:-  160,0,-40,950
     4:-  160,0,-100,950
     5:-  160,-40,-140,950,1,0,1,1,1,1
     6:-  160,-41,-140,0,1,1,1,1,1,1

    *** ELEMENT DATA

 >> ENTER ELEMENT TYPE NUMBER
 >> (TRUSS =1 ; BEAM =2 ; MEMBRANE(PLANE STRESS) =3 ;
 >> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4 ;
 >> 8-NODE-BRICK(3D) =5 ; PLATE AND THIN SHELL =6 ;
 >> BOUNDARY ELEM =7 ; OLD THICK SHELL 16-NODE-BRICK =10 ;
 >> VARIABLE-NUMBER-NODES THICK SHELL =8 ;
 >> PIPE ELEMENT =12)
     12
```

```
>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    n

   *** PIPE ELEMENT

>> ENTER NUMBER OF ELEMENTS,NUMBER OF DIFF. MATERIALS,
>> MAX NUMBER OF TEMPATURE POINTS FOR ANY MATERIAL
>> AND NUMBER OF DIFFERENT SECTIONS.
    4,1,1,1

>> FOR EACH MATERIAL ENTER THE NUMBER OF DIFF.
>> TEMPERATURES AT WHICH MATL. PROPERTIES ARE SPECIFIED
    1:-  1
     <> FOR EACH DIFF. TEMPERATURE FOR THIS MATL.
     <> ENTER TEMPERATURE, ELASTIC MODULUS, POISSONS RATIO,
     <> AND COEFF. OF THERMAL EXPENSION
         1:-  950,23200, .3, .00000716

>> FOR EACH SECTION ENTER FOLLOWING PER LINE
>> OUTSIDE DIA., WALL THICKNESS, AND
>> WEIGHT & MASS PER UNIT LENGTH
    1:-  8.625, .5, .631

>> ARE THERE GRAVITY,THERMAL,OR DISTRIBUTED LOADS
>> APPLIED? (ENTER Y OR N)
    y

>> GIVE FRACTION OF GRAVITY LOADS IN X-, Y-, Z-
>> DIRECTIONS, ALSO GIVE FRACTION OF THERMAL AND
>> DISTRIBUTED LOADS
    0,0,0,1,0

>> FOR EACH PIPE ELEMENT ENTER FOLLOWING PER LINE
>> T(FOR STRAIGHT) OR B(FOR BEND); JOINT NUMBERS
>> I-(BEGINNING) AND J-(ENDING);MATERIAL I.D.;
>> SECTION I.D.; STRESS-FREE TEMP. AND INTERNAL PRESSURE
>> (NOTE: ADDL. INFORMATION NEEDED FOR BENDS)
    1:-  t,1,2,1,1,70,0
    2:-  b,2,3,1,1,70
     <> ENTER ADDITIONAL DATA BOR BEND ELEMENT:
     <> RADIUS OF CURVATURE; THIRD POINT TYPE
     <> (TI = TANGENT INTERSECT, OR CC = CENTER OF CURVATURE);
     <> THIRD POINT COORD. IN X-, Y-, Z-DIRECTIONS
         40,cc,120,0,-40
    3:-  t,3,4,1,1,70
    4:-  b,4,5,1,1,70
```

```
<> ENTER ADDITIONAL DATA BOR BEND ELEMENT:
   40,cc,160,-40,-100

>> ENTER ELEMENT TYPE NUMBER
   7

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
   n

   *** BOUNDARY ELEMENTS

>> ENTER NUMBER OF ELEMENTS
   1

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODE N (WHERE THE ELEMENT IS PLACED, IN ASCENDING ORDER),
>> NODE I (THIS DEFINES THE DIRECTION OF THE ELEMENT),
>> DISPLACEMENT CODE(0 OR 1), ROTATION CODE(0 OR 1),
>> SPECIFIED DISPLACEMENT, SPECIFIED ROTATION, AND
>> SPRING STIFFNESS (SET TO 1.0*E10 IF LEFT BLANK)
   5,6,1,0,.025

   ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
   0

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
   -pipe

>> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
>> $LIST -PIPE                    ** FOR LISTING OF DATA **
>> $RUN CENA:MSAP 5=-PIPE         - FOR STRUCTURAL ANALYSIS
>> $RUN CENA:MSAPLOT              - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list -pipe
>      1       3/D PIPE
>      2        6    2    1    0    0    0
>      3        1    1    1    1    1    1    1       0.0       0.0       0.0    0   950.000
>      4        2    0    0    0    0    0    0   120.000       0.0       0.0    0   950.000
>      5        3    0    0    0    0    0    0   160.000       0.0   -40.000    0   950.000
>      6        4    0    0    0    0    0    0   160.000       0.0  -100.000    0   950.000
>      7        5    1    0    1    1    1    1   160.000   -40.000  -140.000    0   950.000
>      8        6    1    1    1    1    1    1   160.000   -41.000  -140.000    0       0.0
>      9       12    4    1    1    1    0    4
>     10        1    1
>     11     9.500E+02 2.320E+04 3.000E-01 7.160E-06
>     12        1 8.625E+00 5.000E-01       6.310E-01 0.0
>     13     0.0
>     14     0.0
>     15     0.0
>     16     1.000E+00
>     17     0.0
>     18       1T    1    2    1    1 7.000E+01 0.0                                        0
>     19       2B    2    3    1    1 7.000E+01 0.0                                        0
>     20     4.000E+01   CC 1.200E+02 0.0      -4.000E+01
>     21       3T    3    4    1    1 7.000E+01 0.0                                        0
>     22       4B    4    5    1    1 7.000E+01 0.0                                        0
>     23     4.000E+01   CC 1.600E+02-4.000E+01-1.000E+02
>     24        7    1
>     25             1.0
>     26        5    6                   1    0    0 2.500E-02 0.0
>     27
>     28             1.0       0.0       0.0       0.0
#END OF FILE
                     1         2         3         4         5         6         7         8
            12345678901234567890123456789012345678901234567890123456789012345678901234567890
```

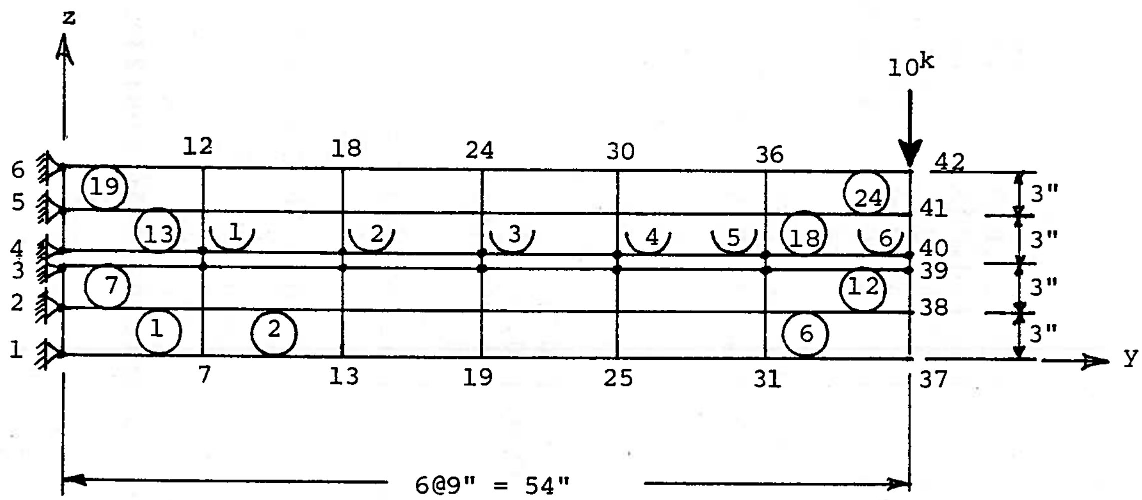
EXAMPLE: 10

Beam with Contact Elements

Element Type

plane stress (4)

Contact (9)



3/4" - thick aluminum plate

E = 10,600 ksi & ν = .3

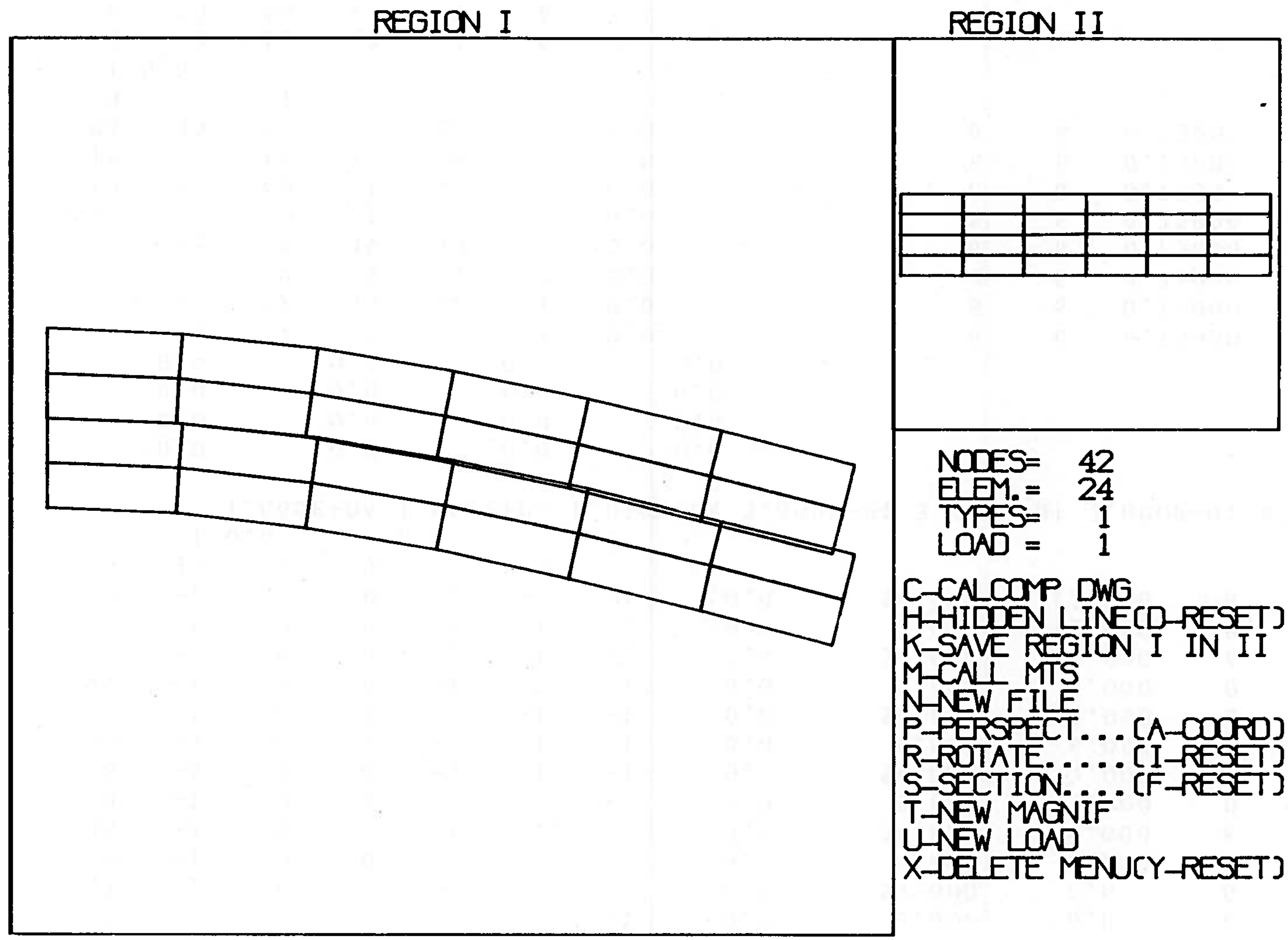
```
#run cena:premsap
#EXECUTION BEGINS
 >> ENTER  ANALYSIS NUMBER (0= STATIC, 1= DYNAMIC, 2= DYNAMIC ADD.)
     0

 >> ENTER PROBLEM TITLE ON ONE LINE
        double beam with contact elements

 >> ENTER NUMBER OF JOINTS, NUMBER OF ELEMENT TYPES, AND
 >> NUMBER OF LOAD CASES (ON ONE LINE SEPARATED BY COMMA)
     42,2,1

 >> IS THE PROBLEM 2-DIMENSIONAL?
 >> ENTER Y FOR YES OR N FOR NO
     y

 >> NOTE: FOR 2-D PROBLEMS, ALL DATA MUST BE PREPARED IN Y-Z PLANE

 >> ARE THERE BEAM, PIPE, THIN SHELL OR PLATE
 >> ELEMENTS INVOLVED IN THE PROBLEM? (ENTER Y OR N)
     n

 >> IS 2-D (QUADRILATERAL) NODAL POINT MESH
 >> GENERATION TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH 2 NODAL POINT MESH GENERATION SET
 >> ENTER FOLLOWING SIX ITEMS PER LINE,
 >> GLOBAL NODES I,J,K,L, AND NODAL NUMBER INCREMENTS
 >> KN FOR I-J AND FOR I-L.
 >>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     7,37,39,9,6,1
     10,40,42,12,6,1


 >> IS LINEAR NODAL POINT MESH GENERATION
 >> TO BE PERFORMED? (ENTER Y OR N)
     y

 >> FOR EACH LINEAR NODAL POINT MESH GENERATION SET ,
 >> ENTER FOLLOWING PER LINE (IN INCREASING SEQ.)
 >> THE FIRST NODAL POINT NUMBER,THE LAST N. P. NUMBER,
 >> AND THE MESH GENERATION INCREMENT KN (POSITIVE)
 >> TERMINATE THIS SEQUENCE WITH A BLANK LINE
     1,3,1,
     4,6,1


    *** JOINT DATA

 >> FOR EACH JOINT ENTER FOLLOWING PER LINE
 >> THE Y-, Z-COORD., NODE TEMP., AND BOUNDARY COND. CODES
 >> (0 FOR FREE, 1 FOR CONSTRAINED) IN Y-DIR., IN Z-DIR.
     1:-  0,0,0,1,1
```

```
 3:-   0,6,0,1,1
 4:-   0,6,0,1,1
 6:-   0,12,0,1,1
 7:-   9,0,
37:-   54,0,
 9:-   9,6
39:-   54,6
10:-   9,6
40:-   54,6
12:-   9,12
42:-   54,12

*** ELEMENT DATA

>> ENTER ELEMENT TYPE NUMBER
>> (TRUSS =1,  BEAM =2,  MEMBRANE(PLANE STRESS) =3,
>> PLANE STRESS, PLANE STRAIN, OR AXISYMMETRIC =4,
>> 8-NODE-BRICK(3D) =5,  PLATE AND THIN SHELL =6,
>> BOUNDARY ELEM =7,  VARIABLE-NUMBER-NODES THICK SHELL =8,
>> CONTACT (LINK) ELEM =9,  OLD 16-NODE-BRICK =10,
>> PIPE ELEMENT =12)
    4

>> IS 2-D (QUADRILATERAL) ELEMENT MESH
>> GENERATION TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY 2-D ELEMENT MESH GENERATION
>> ENTER FOLLOWING EIGHT ITEMS PER LINE,
>> THE FIRST AND LAST ELEMENT NUMBERS (IN INCREASING
>> SEQ.), GLOBAL NODES I,J,K,L, AND NODAL NUMBER
>> INCREMENTS KN FOR I-J AND FOR I-L.
>>  (NOTE: NODAL POINT "I" MUST HAVE SMALLEST NODE NUMBER)
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,12,1,37,39,3,6,1
    13,24,4,40,42,6,6,1


>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    n

*** 2-DIMENSIONAL FINITE ELEMENTS

>> ENTER NUMBER OF ELEMENTS, AND NUMBER OF DIFF. MATERIALS
    24,1

>> ENTER ANALYSIS TYPE NUMBER
>> ( 0 = AXISYMMETRIC, 1 = PLANE STRAIN, 2 = PLANE STRESS)
    2

>> FOR EACH DIFF. MATERIAL ENTER FOLLOWING PER LINE
>> MODULUS OF ELASTICITY, POISSONS RATIO, AND
>> WEIGHT & MASS DENSITIES
    1:-   10600, .3
```

```
>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> NODES I, J, K, L(FOR TRIANGULAR ELEM. L=K), MATERIAL I.D. NO.,
>> NSPRT (SEE NOTE), ELEMENT THICKNESS (FOR PLANE STRESS ONLY)
>>  (NSPRT= 0  FOR STRESS OUTPUT AT ELEMENT CENTER
>>          1  FOR NO STRESS OUTPUT
>>          8  FOR STRESS AT CENTER & MIDPOINT OF SIDE I-L
>>         20  FOR STRESS AT CENTER & MIDPOINT OF ALL SIDES)
    1:-  7,8,2,1,1,8,.75
   13:-  5,4,10,11,1,8,.75

>> ENTER ELEMENT TYPE NUMBER
    9

>> IS LINEAR ELEMENT MESH GENERATION
>> TO BE PERFORMED? (ENTER Y OR N)
    y

>> FOR EVERY LINEAR ELEMENT MESH GENERATION
>> ENTER FOLLOWING PER LINE, THE FIRST AND LAST
>> ELEMENT NUMBERS (IN INCREASING SEQ.)
>> AND THE MESH GENERATION INCREMENT K
>> TERMINATE THIS SEQUENCE WITH A BLANK LINE
    1,6,6


   *** CONTACT (LINK) ELEMENTS

>> ENTER NUMBER OF ELEMENTS AND NUMBER OF DIFF. SPRING CONSTANTS
    6,1

>> FOR EACH DIFF. SPRING ENTER FOLLOWING PER LINE
>> VALUE OF SPRING CONSTANT (SET TO 10E+10 IF LEFT BLANK)
    1:-

>> FOR EACH ELEMENT ENTER FOLLOWING PER LINE
>> JOINT NOS. I-(BEGINNING), J-(ENDING), K-(FOR DIRECTION)
>> AND SPRING CONSTANT I.D. NO.
    1:-  9,10,12,1

   ***CONCENTRATED LOAD DATA

>> ENTER NUMBER OF JOINTS WITH CONCENTRATED LOADS
    1

>> FOR EACH LOADED JOINT ENTER FOLLOWING PER LINE
>> JOINT NO.(IN INCREASING SEQ.), AND LOADS IN Y-, Z-DIR.
    42, 0, -10

>> THE PROGRAM IS READY TO STORE THE DATA IN YOUR OWN FILE
>> ACCORDING TO THE INPUT FORMAT OF MSAP
>> ENTER YOUR OWN FILE NAME(8 CHARACTERS OR LESS)
```

```
     contact

 >> YOU ARE NOW READY TO OBTAIN YOUR RESULTS BY TYPING
 >> $LIST CONTACT                     ** FOR LISTING OF DATA **
 >> $RUN CENA:MSAP 5=CONTACT          - FOR STRUCTURAL ANALYSIS
 >> $RUN CENA:MSAPLOT                 - FOR GRAPHIC DISPLAY
#EXECUTION TERMINATED
```

```
#list contact
>    1       DOUBLE BEAM WITH CONTACT ELEMENTS
>    2        42    2    1    0    0    0
>    3         1   -1    1    1   -1   -1   -1      0.0       0.0       0.0      0      0.0
>    4         3   -1    1    1   -1   -1   -1      0.0       0.0       6.000    1      0.0
>    5         4   -1    1    1   -1   -1   -1      0.0       0.0       6.000    0      0.0
>    6         6   -1    1    1   -1   -1   -1      0.0       0.0      12.000    1      0.0
>    7         7   -1    0    0   -1   -1   -1      0.0       9.000     0.0      0      0.0
>    8        37   -1    0    0   -1   -1   -1      0.0      54.000     0.0      6      0.0
>    9         9   -1    0    0   -1   -1   -1      0.0       9.000     6.000    0      0.0
>   10        39   -1    0    0   -1   -1   -1      0.0      54.000     6.000    6      0.0
>   11         8   -1    0    0   -1   -1   -1      0.0       9.000     3.000    0      0.0
>   12        38   -1    0    0   -1   -1   -1      0.0      54.000     3.000    6      0.0
>   13        10   -1    0    0   -1   -1   -1      0.0       9.000     6.000    0      0.0
>   14        40   -1    0    0   -1   -1   -1      0.0      54.000     6.000    6      0.0
>   15        11   -1    0    0   -1   -1   -1      0.0       9.000     9.000    0      0.0
>   16        41   -1    0    0   -1   -1   -1      0.0      54.000     9.000    6      0.0
>   17        12   -1    0    0   -1   -1   -1      0.0       9.000    12.000    0      0.0
>   18        42   -1    0    0   -1   -1   -1      0.0      54.000    12.000    6      0.0
>   19         4   24    1    0    2    0
>   20         1    1 0.0       0.0       0.0
>   21              1.060E+04 1.060E+04 1.060E+04 3.000E-01 3.000E-01 3.000E-01 4.077E+03
>   22
>   23           0.0       0.0       0.0       0.0
>   24           0.0       0.0       0.0       0.0
>   25           0.0       0.0       0.0       0.0
>   26           0.0       0.0       0.0       0.0
>   27         1    1    7    8    2    1   0.0                      8    6   0.75000
>   28         6   31   37   38   32    1   0.0                      8    6   0.75000
>   29         7    2    8    9    3    1   0.0                      8    6   0.75000
>   30        12   32   38   39   33    1   0.0                      8    6   0.75000
>   31        13    4   10   11    5    1   0.0                      8    6   0.75000
>   32        18   34   40   41   35    1   0.0                      8    6   0.75000
>   33        19    5   11   12    6    1   0.0                      8    6   0.75000
>   34        24   35   41   42   36    1   0.0                      8    6   0.75000
>   35         9    6    1
>   36         1 0.0
>   37         1    9   10   12    1    6
>   38         6   39   40   42    1    6
>   39        42    1 0.0       0.0      -1.000E+01 0.0       0.0       0.0
>   40
>   41           0.0       0.0       0.0       0.0
#END OF FILE
```



DOUBLE BEAM WITH CONTACT ELEMENTS

3. MSAPLOT

3.1 Data Input

After the input data of the problem has been prepared according to MSAP (or SAP IV[5]) format, and is stored in a file, MSAPLOT may be run using the following Michigan Terminal Systems (MTS) command:

$RUN CENA:MSAPLOT

When the execution begins the program will prompt the user for his data file name by printing on the screen,

ENTER INPUT DATA FILE NAME

At this stage the user enters the name of the file in which the input data of his problem is stored. After the input data has been entered, the program prints out the following on the screen.

IF DEFLECTED STRUCTURE TO BE DISPLAYED ENTER Y OTHERWISE ENTER N.

The user must enter the letter Y or N at this stage. If Y is entered, the program prints out the message.

ENTER DEFLECTION DATA FILE NAME.

If MSAPLOT is run immediately after running MSAP, the user must enter the temporary file name -DEF which contains all his deflection (or mode shape) information. If -DEF had been copied into a permanent file, say DEFLECT then the latter may be entered at this time instead. -DEF is a temporary file which was created automatically by MSAP to store the displacements for the graphic display of the deflected structure. Unless it is copied into a permanent file it is destroyed after signing off.

After the deflection data file name is entered, the program will print out the message,

ENTER MAGNIFICATION FACTOR.

The magnification factor is used to enlarge the value of deflections (displacements and/or rotations) to make them visible.

If more than one load case, (or frequency) let us say N, had been considered the following message is printed out next.

```
TOTAL NUMBER OF LOAD CASES = N.
ENTER LOAD CASE.
```

The user decides which load case he wants to see displayed and enters its number at this time.

If there are no major errors in the input and/or deflection data files, the program will display at this time the menu along with the x-y plot of the structure under study in Region I of the plot as shown in Fig. 1.

MSAPLOT divides the screen into two display regions. Region I is where the first display is drawn. It is the main display area where the user can blow-up part of the structure or rotate it to obtain a different view or draw a perspective. Region II is used to store a reference picture.

If there is any gross error in the data file the program will either not display at all or display something quite different. For trouble shooting purposes the user may list -GDAT after the program has turned control over to MTS. File -GDAT is a temporary file created automatically by MSAPLOT. It contains the generated nodal point and element information from the input data of the structure. The user may check the list of "-GDAT" to find where he had gone wrong, correct his input file and try again.

3.2 Command Mode

When the program has completed processing the input data, the menu along with the x-y plot of the structure is displayed on the screen (in Region I). It is now ready to receive orders. In this mode, the presence of the graphic cursor (or cross hairs) on the screen indicates that the program is waiting for the user to enter a command. A description of these commands is presented next.

C-CALCOMP

The program control command C is used to produce a CalComp plot. When the user is satisfied with the display on the screen and would like to have an ink drawing made through the University of Michigan CalComp plotter he enters the command C. The program then will prompt him for the scale factor of his CalComp plot. This scale factor can be any number greater than 0.0 and less than or equal to 5.6. The display seen on the screen has a scale factor of 1.0. The above procedure may be repeated as many times as desired.

When the user enters the C command, the data for CalComp is automatically generated by MSAPLOT and stored in a temporary file called -CALC . Note that this file is destroyed at signoff. Therefore to obtain a CalComp plot, the user returns control to MTS by entering the command M and proceeds as follows.

To view what he has placed in the file -CALC he may enter the following command:

```
$RUN *PLOTSEE
 EXECUTION BEGINS                      (MTS reply)
 ENTER FILE NAME                       (MTS question)
```

The user should enter at this time the file name -CALC .

The program then displays the user's CalComp plot for his inspection. When user is ready to inspect his next plot, if such is the case, he should enter at this time the letter N. MTS will respond by printing again,

```
ENTER FILE NAME                        (MTS question)
```

User enters his next file name. To terminate this sequence, he should enter

    $ENDFILE   (or CONTROL-C)

If user decides to obtain a CalComp plot, he should enter the following statements,

    $RENAME -CALC CALC[a]
    $RUN *CCQUEUE
     EXECUTION BEGINS   (MTS reply)
     ENTER FILE NAME   (MTS question)

The user then enters his file name

    CALC

MTS will now give the user the approximate plotting time for the CalComp plot. If user wants the plot(s) he should reply with a Y, otherwise reply with a N. If user confirms the plot, MTS will give him the receipt number for his plot _____. Write this number down and don't lose it!

MTS will then come back with

    ENTER FILE NAME   (MTS question)

If user has another file to be plotted, he may enter it at this time, otherwise he should reply with

    $ENDFILE   (or CONTROL-C)

H-HIDDEN LINE (D-RESET)

The hidden lines in the Region I picture are removed by the command H. After typing H the user is asked to print out the resolution he desires, otherwise carriage return defaults it to a fixed value and the hidden line drawing is plotted.

(a) This is one way of creating a permanent file.

K-SAVE REGION I IN II

The picture in Region I may be permanently stored in Region II by the command K and can be gotten rid of only by storing a new one on top of it.

M-CALL MTS

M is a program control command, with it the user can stop the execution of the program and return control to MTS. The program may be restarted with the MTS command $RESTART.

N-NEW FILE

The control command N causes a return to the input section of the main program and prompting begins again. The user can enter this command and display more than one structure without having to call MSAPLOT again.

P-PERSPECTIVE (A-COORD)

To convert the isometric drawings of Region I into perspective drawings either of the commands P and A may be used. When P is used the program picks up the x and y coordinates directly from the cursor on the screen and asks the user to supply the z distance by typing its value. The first point is the location of the point of view of the observer, and the second is the location of the point of interest.

If command A is used instead of P the x, y and z values of the above two points must be typed in. Note, the geometric center of the structure is taken as the origin of the coordinate system of the display.

R-ROTATE (I-RESET)

The display manipulation command R is used to perform rotations on the structure to obtain a different viewing angle. All rotations are relative to the previous picture shown in Region I.

After R has been entered, the program will prompt the user for the value of the angles in degrees to be rotated about the x, y and z axes respectively. Rotations are always specified with respect to the current Region I display axes. Command I will restore the initial plot.

S-SECTION (F-RESET)

The display manipulation command S enables the user to select a part of the structure enclosed by a rectangle from Region I. The parameters of a section to be blown-up are the lower left-hand and the upper right-hand corners respectively of the rectangle. This is done by moving the cursor (or cross hair) first to the lower corner and entering the letter S a second time, then moving the cursor to the upper corner and entering the letter S a third time. After both corners have been specified, the program outlines the rectangle to be enlarged and awaits for the user's command to display it. To restore the previous display type F.

T-NEW MAGNIF

The program control command T is used to allow a change in the magnification factor of the displacements, i.e., when the user is not satisfied with his current magnification factor effects on the deformed structure.

U-NEW LOAD (or new frequency)

The program control command U is used to display various load cases (or mode shapes as the case may be) when more than one loading (or frequency) is considered in the structural analysis.

X-DELETE MENU (Y-RESET)

Use of command X deletes all printed information including the menu and command Y will restore it back.

3.3 STRESS CONTOUR

In most structural analyses the designer is primarily concerned with the maximum values of the stress. As a result most of the finite element analysis results are essentially discarded and never put to use. Option Z (contour mapping of stresses) combats this deficiency by converting the stress outputs into equivalent contour maps for visual observation through Christiansen's MOVIE.BYU program [10].

Any of the six stress components may be plotted, as well as the direction of the principal stresses for membrane and eight-mode-brick elements, i.e. element types 3, 4, and 5. Each of the contour lines represent a constant stress. The program will divide the maximum stress difference into as many as 27 constant stress lines. At a glance the designer can identify the critical areas of this model and proceed to study them in detail.

In order to obtain a contour plot, stresses must be supplied at the nodal (grid) points. However in finite element analyses stresses are obtained at the center, or at the mid-point of the edges of the elements, hence an averaging of these stresses is performed to convert them to equivalent nodal point stresses.

The latter is accomplished internally by MSAP and is saved on a temporary file. Option Z of the MSAPLOT menu requires three such files (all generated by MSAP) in order to draw the displacement and stress contour plots.

The names of these files are:

-RGEOM	(geometric data)
-RDISP	(displacement data)
-RSTRS	(stress data)

They must be copied to permanent files if the plotting is to be done at a later time.

3.4 MSAPLOT PLOTS

Example drawings are presented in Figs. 1 through 11. These were generated using different element types and structures through the application of the MSAPLOT options described above and are self explanatory.

Figures 2, 8 and 10 demonstrate contour mapping of stresses for various structures and stress components. They are easy to follow and show at a glance where the critical areas are. MSAP stress output in these areas may now be examined in detail.

References

1. Kaldjian, M. J., "Interactive Data Preprocessor Program for Michigan SAP (MSAP)," Journal of Computers and Structures, June 1976.

2. Kaldjian, M. J., "Interactive and Data Mode Preprocessor with Mesh Generation for Michigan SAP," Proc., First SAP User's Conference, Univ. of Southern California, Los Angeles, June 1976.

3. Kaldjian, M. J., "Three Dimensional Interactive Graphic Display Program for Michigan SAP (MSAP)," Journal of Computers and Structures, June 1976.

4. Kaldjian, M. J., "Interactive Pre- and Post Processors for SAP with Quadrilateral Mesh Generation and Hidden Line Graphic Display," Proc., Second SAP User's Conference, Univ. of Southern Calif. Los Angeles,

5. Bath, K. J., E. L. Wilson, and F. E. Peterson, "SAP IV - A Structural Analysis Program for Static and Dynamic Response of Linear Systems," EERC 73-11 Earthquake Engg. Research Center, College of Engg., Univ. of Calif., Berkeley, April, 1974.

6. Wilson, E. L., "SOLID SAP - A Static Analysis Program for 3-D Solid Structures," US-SESM 71-19, Dept. of Civil Engg., Univ. of Calif., Berkeley, Dec. 1972.

7. Conklin, J.W., M. Barnett, "A Basic Software Package for the Computek Terminal on MTS."

8. Van Roekel, J., "MGI User's Guide - An Introduction to the Michigan Graphics Interpreter," Dept. of Aerospace Engg., The University of Michigan, Aug. 1973.

9. Perrone, N., and W.D. Pilkey, Structural Mechanics Software Series I, The University Press of Virginia, Charlottesville, VA. 1977. (Kaldjian, M. J., "Interactive and Data Mode Preprocessor for SAP", pp. 227-274)

10. Christiansen, H., and M. Stephenson, "Graphics Utah Style-77", A Workshop on Interactive Computer Graphics, August 15-19, 1977, Snowbird Resort, Utah.

11. Kaldjian, M.J., "Interactive Data and Graphic Display Processors for MSAP," SAE Proc., 3rd International Conf. on Vehicle Structural Mechanics, Troy, Michigan, Oct. 1979.

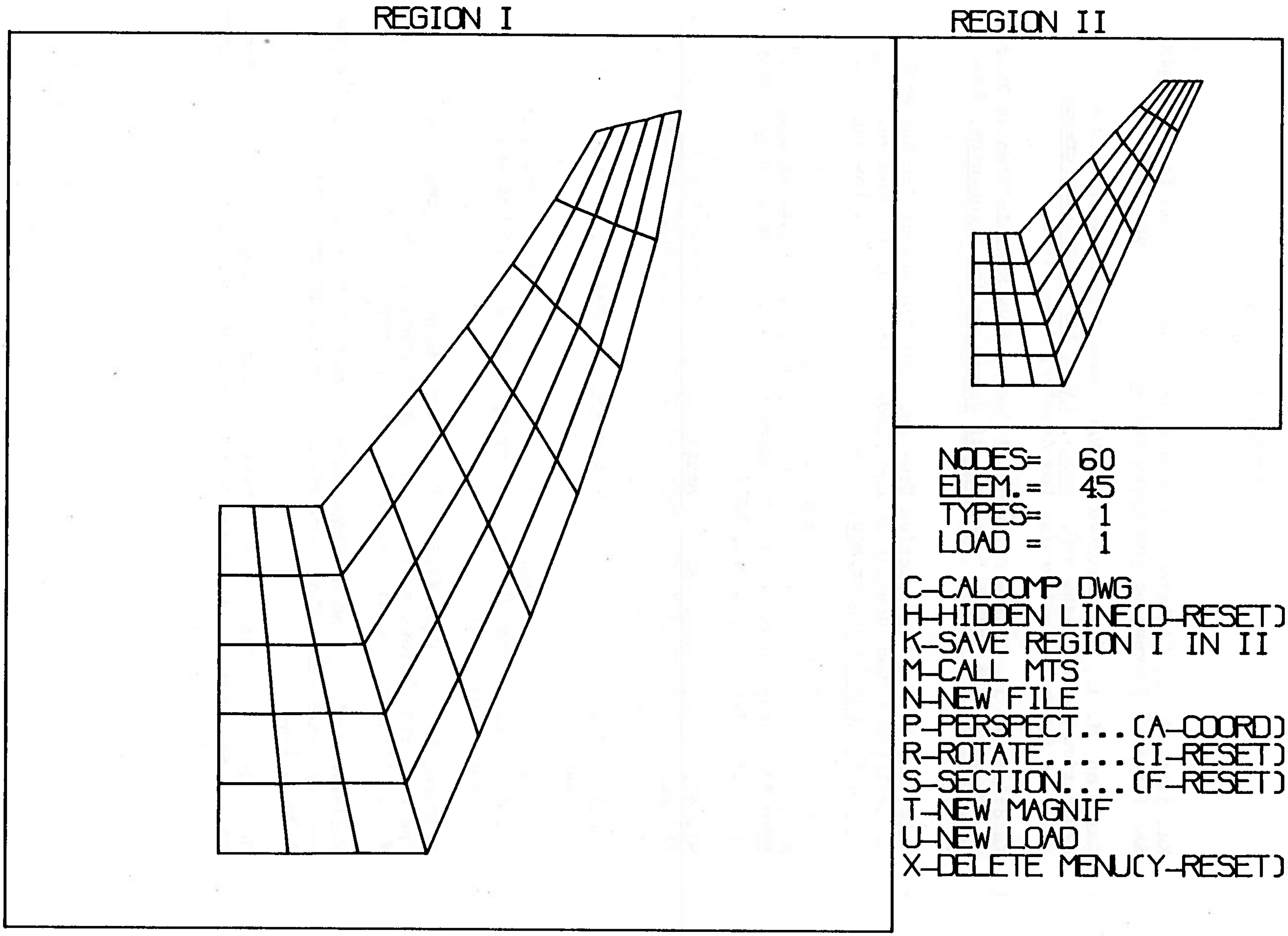


FIGURE 1. Web Frame with Deflection

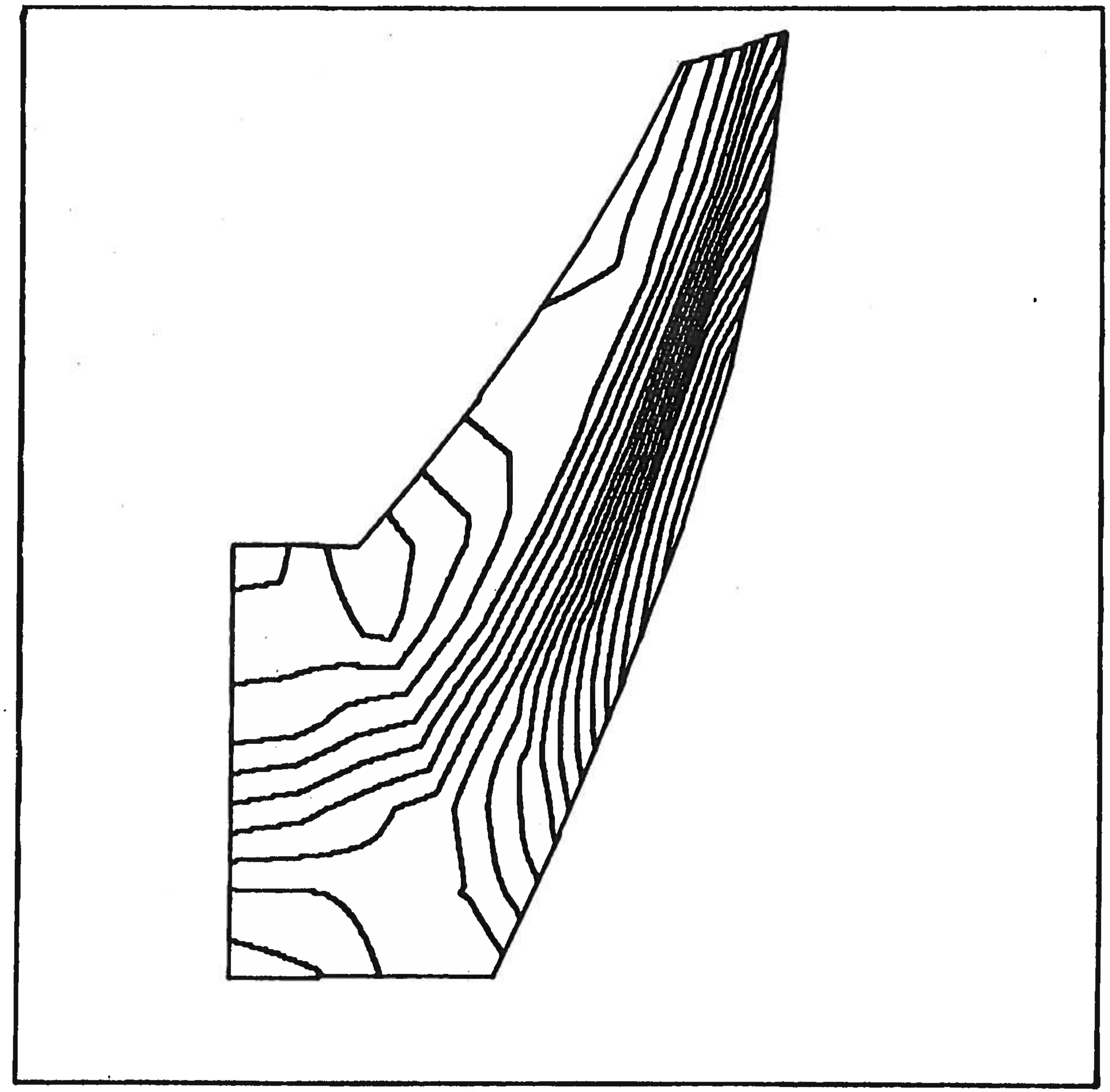


FIGURE 2. Web Frame with Principal Tensile Stress Contours

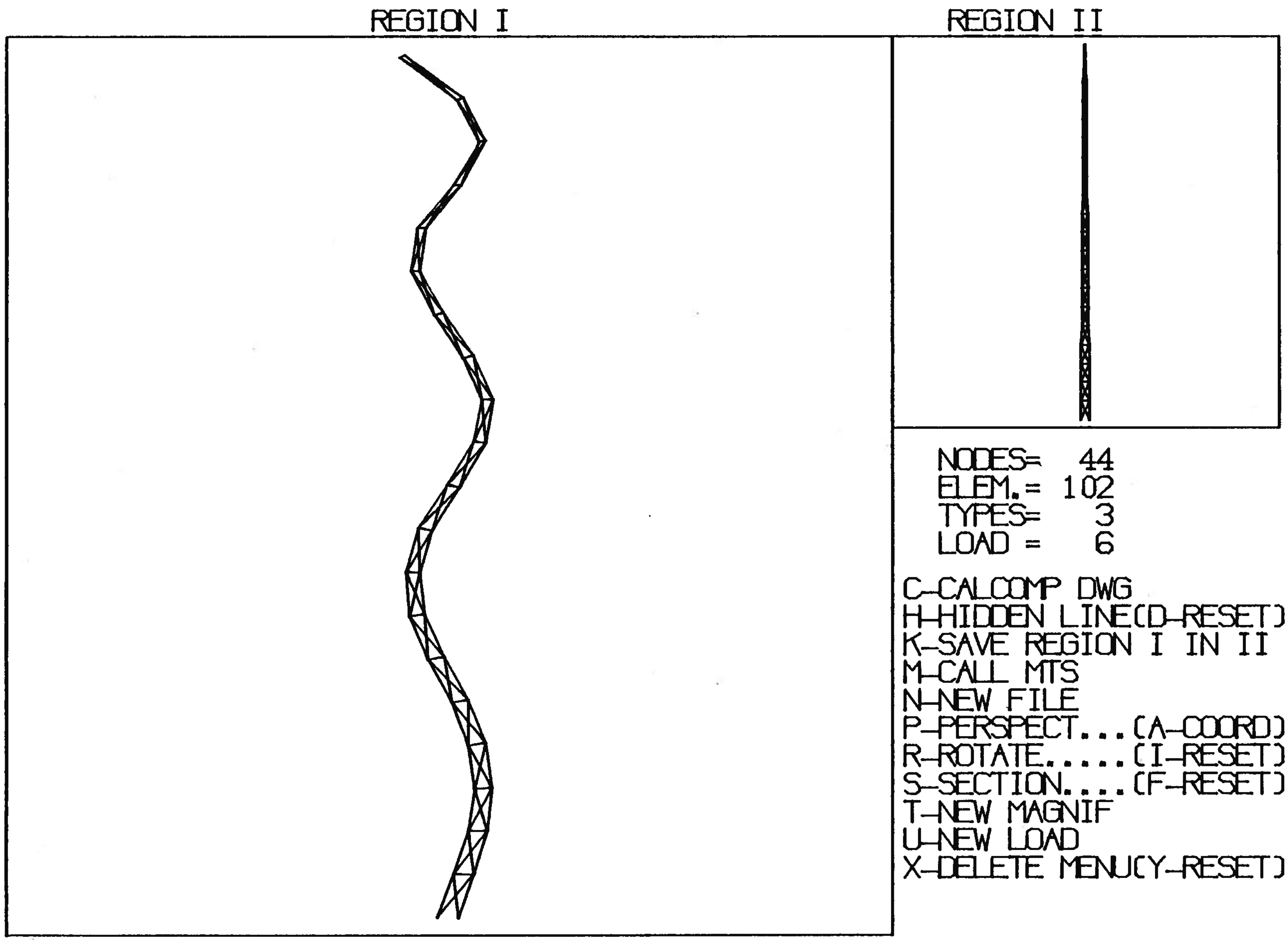


FIGURE 3. Tower Mode Shapes

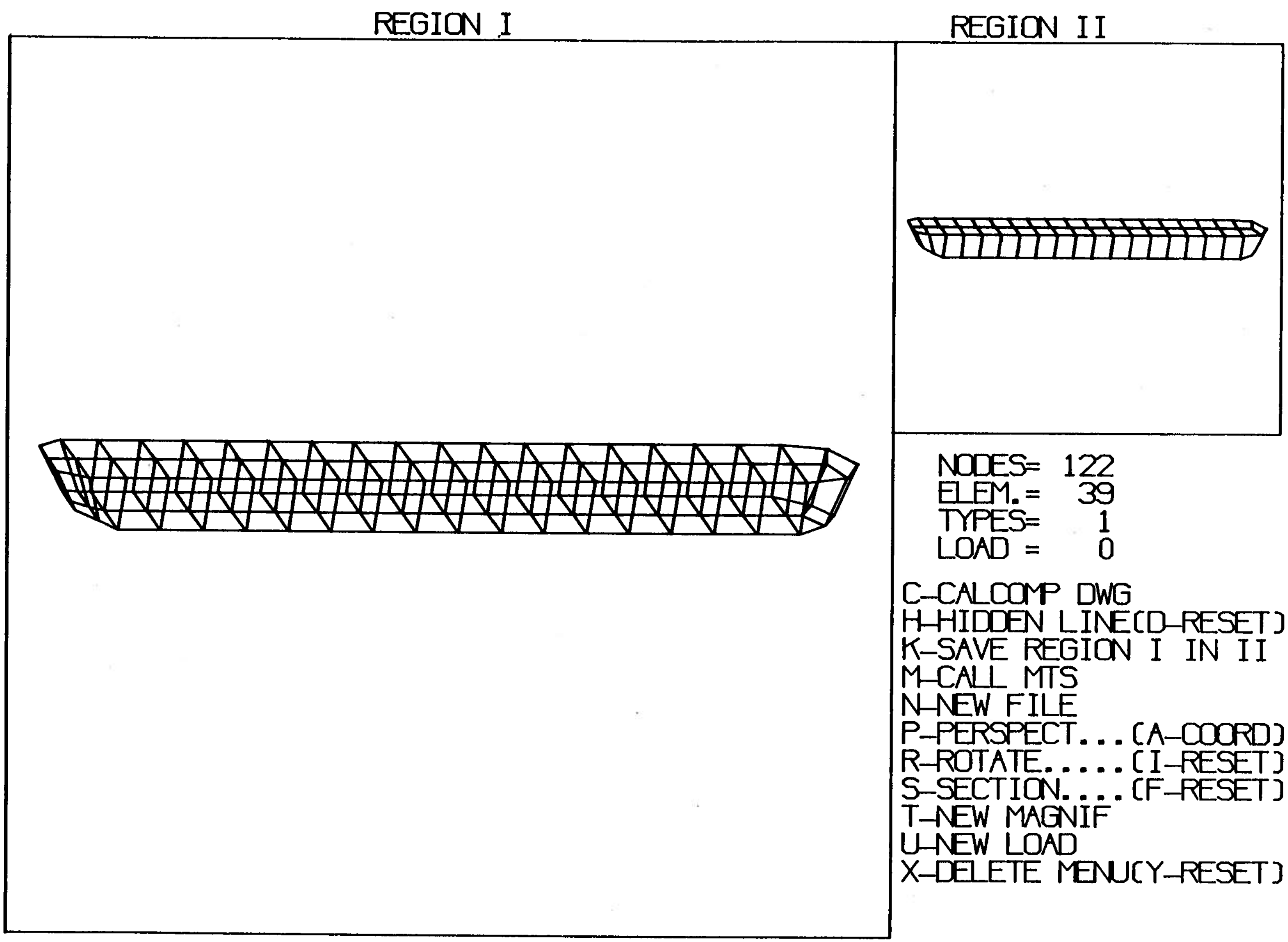


FIGURE 4. Tanker Model - Isometric

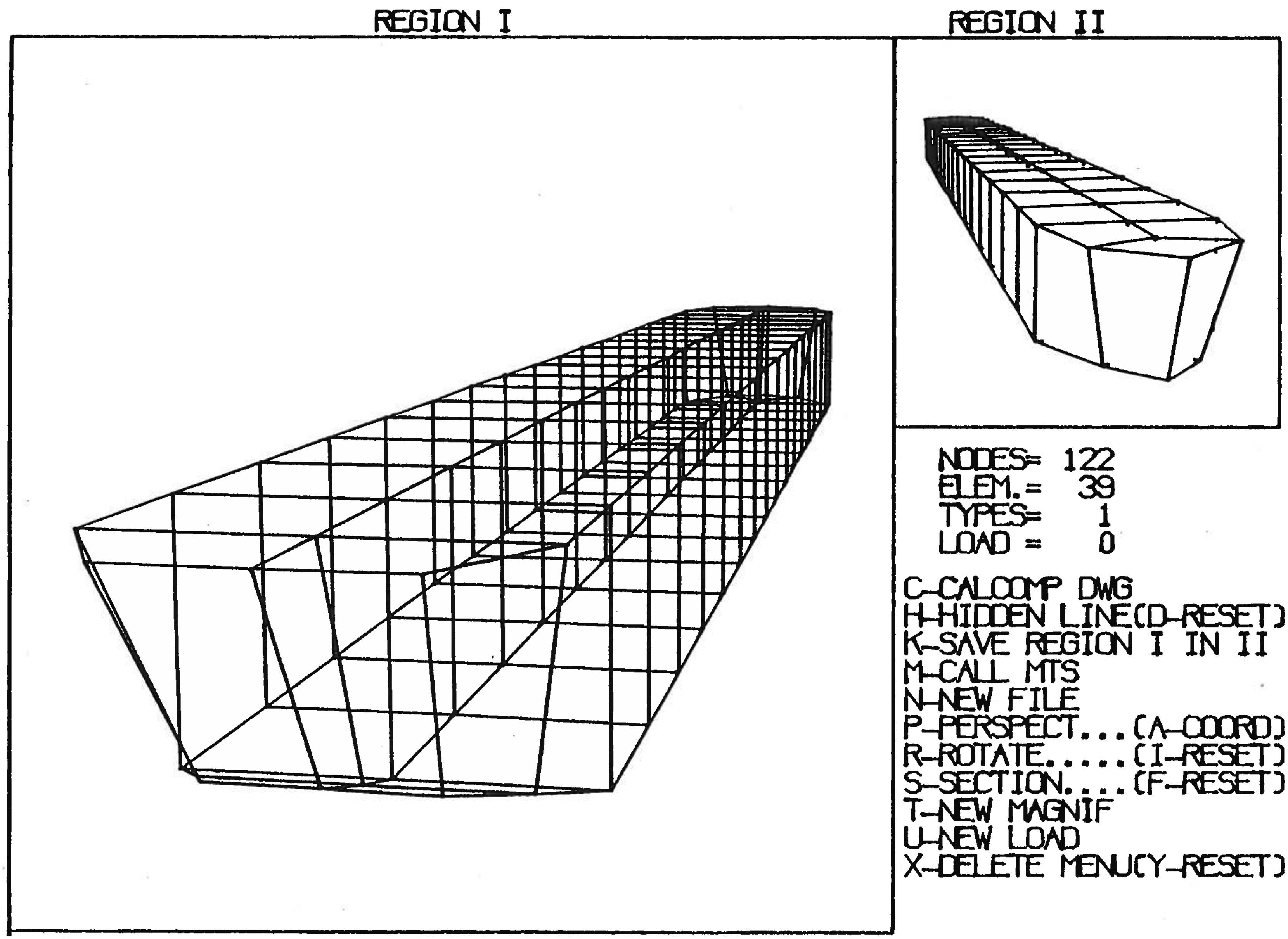


FIGURE 5. Tanker Transverse Web Frame

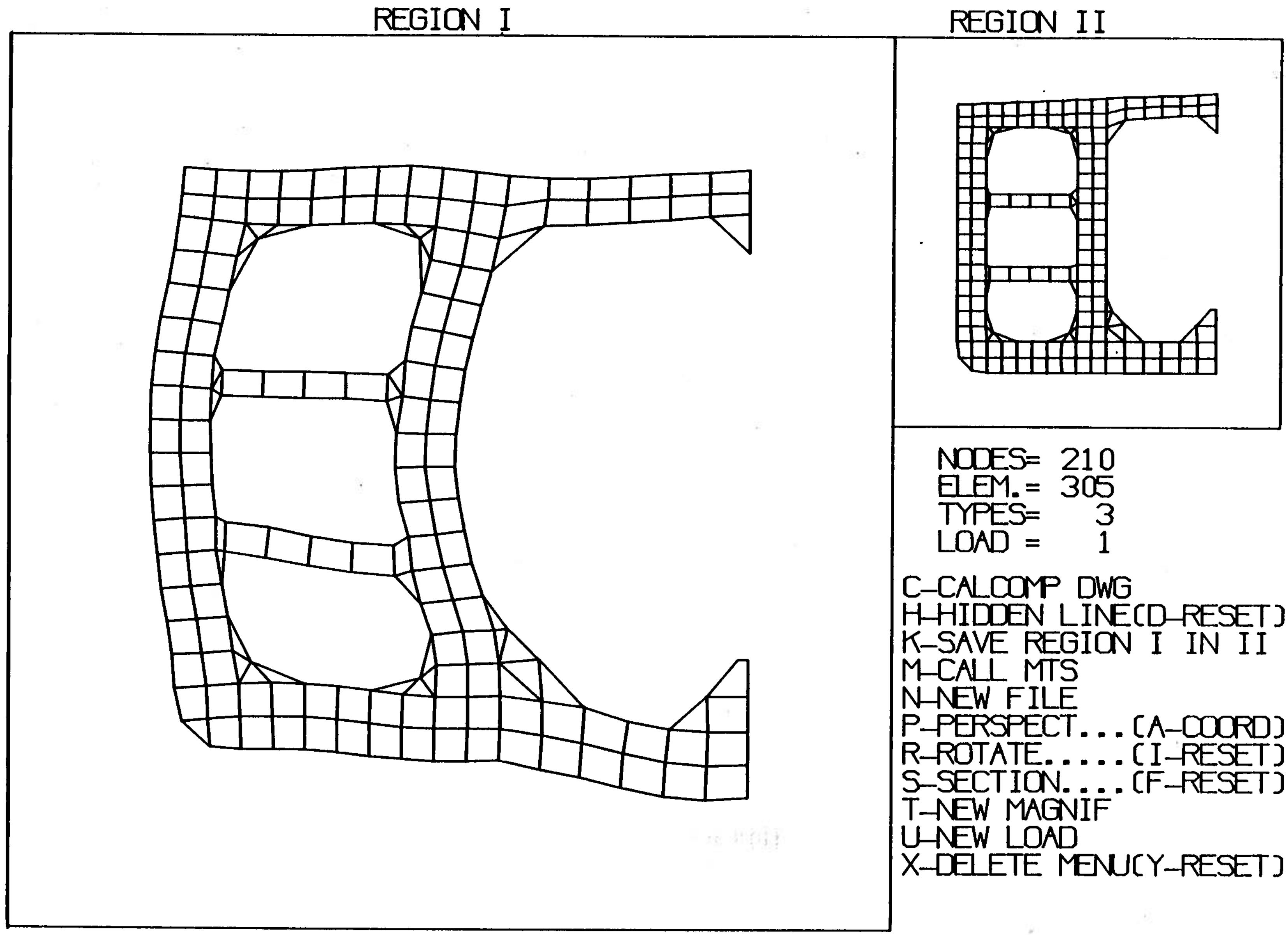


FIGURE 6. Tanker Transverse Web Frame

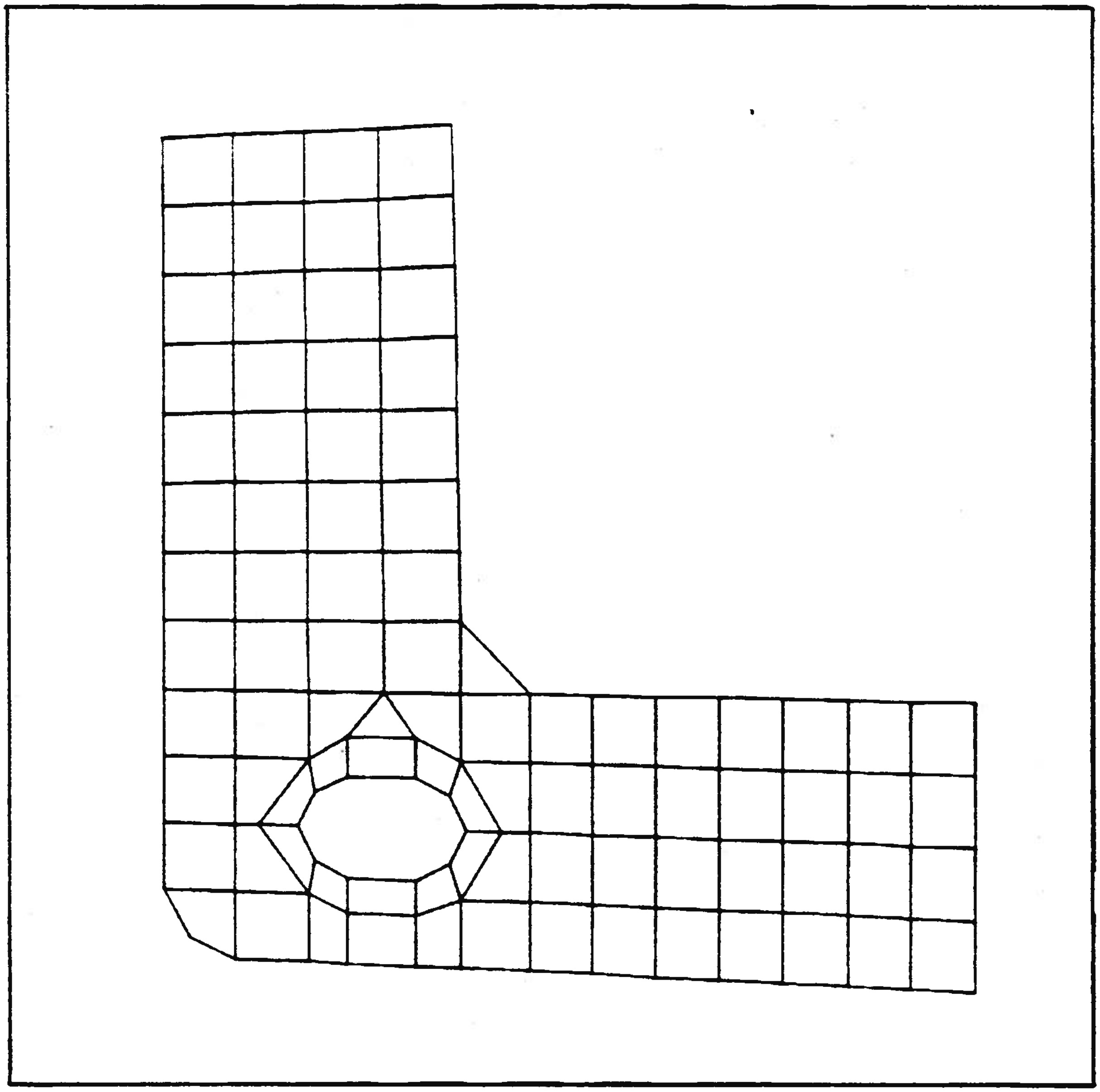


FIGURE 7 Transverse Frame Corner

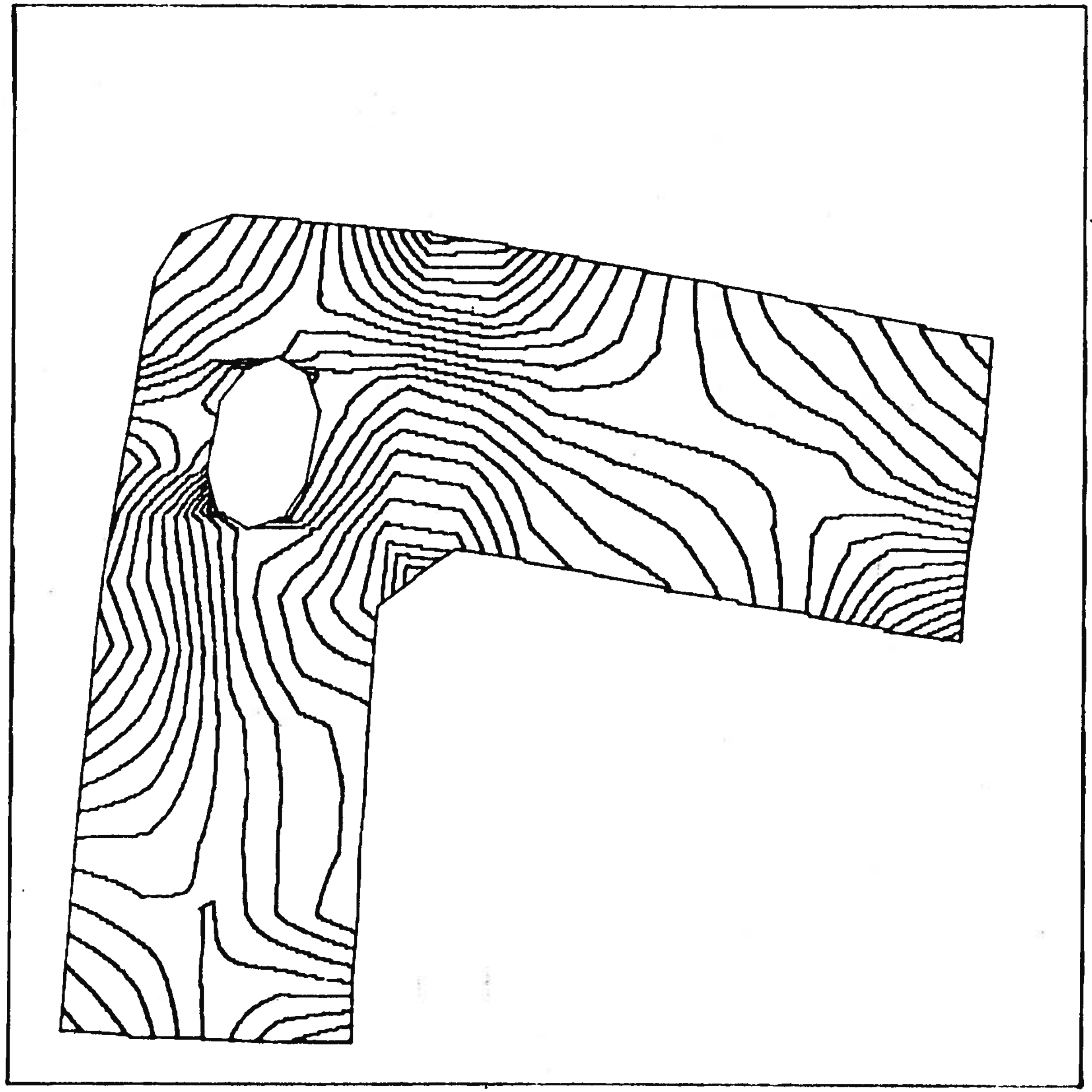


FIGURE 8 Transverse Frame Corner with Principal Stress Contours & Displacements

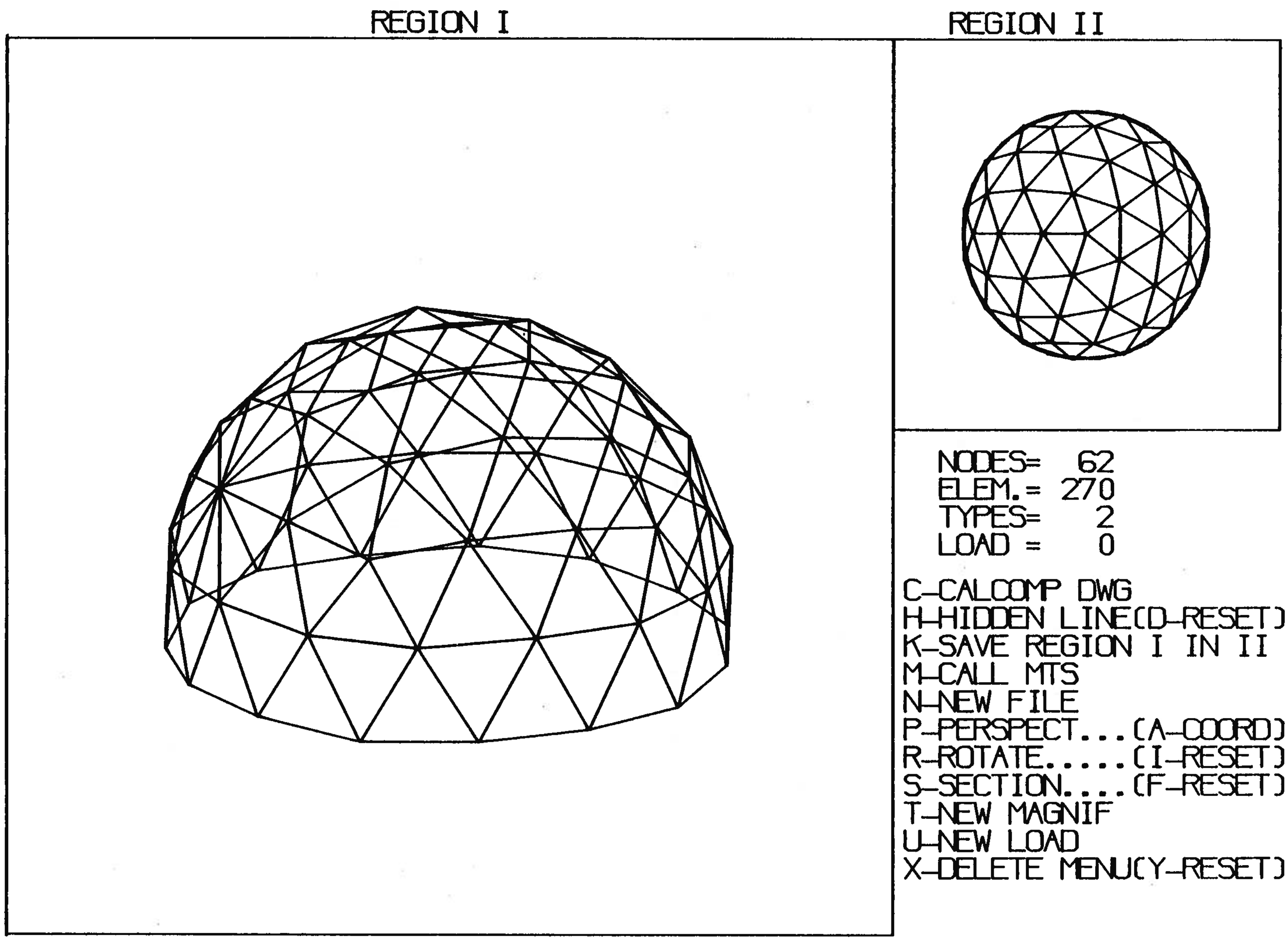


FIGURE 9. Geodesic Dome

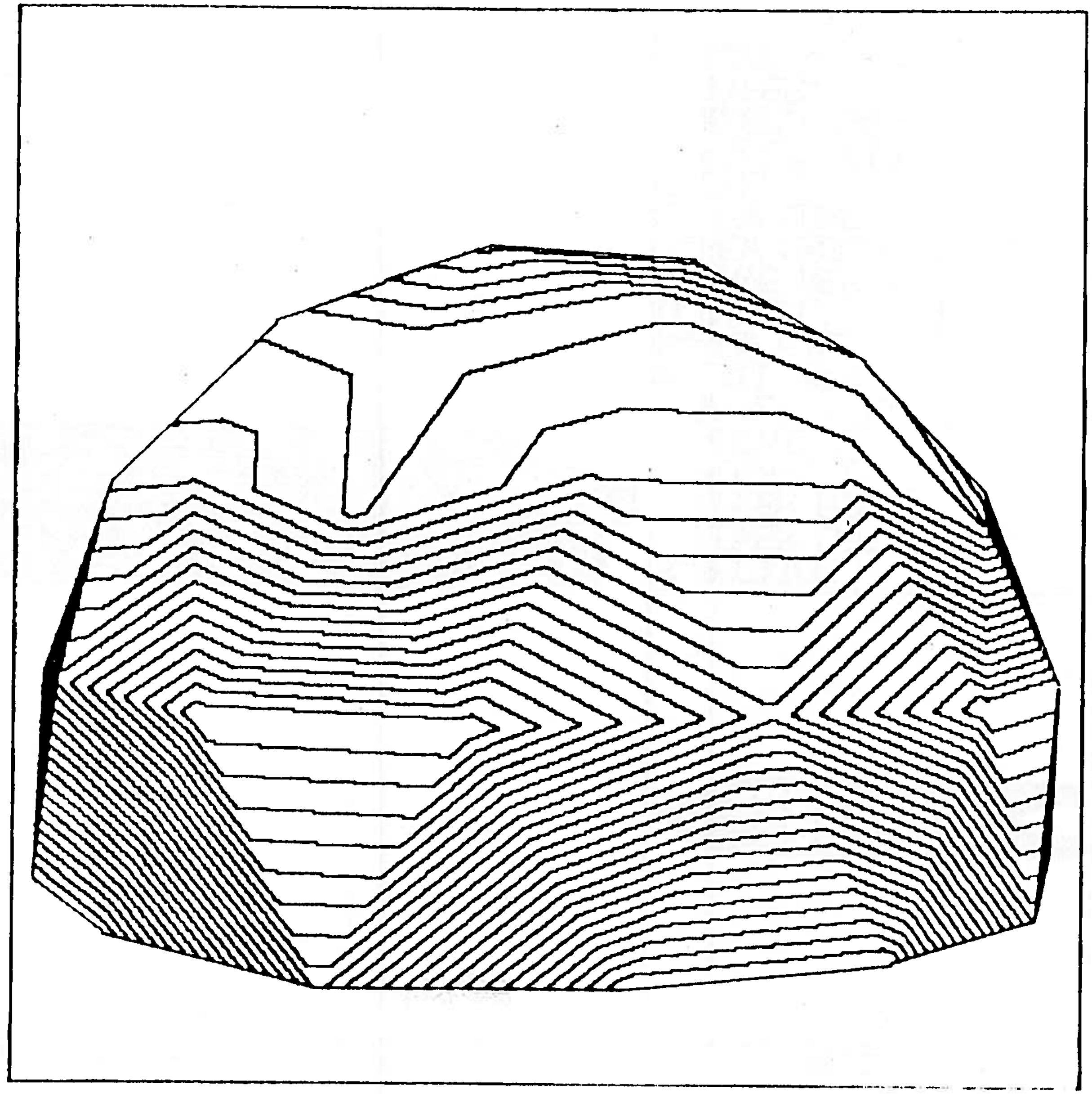


FIGURE 10. Geodesic Dome with refined stress contours

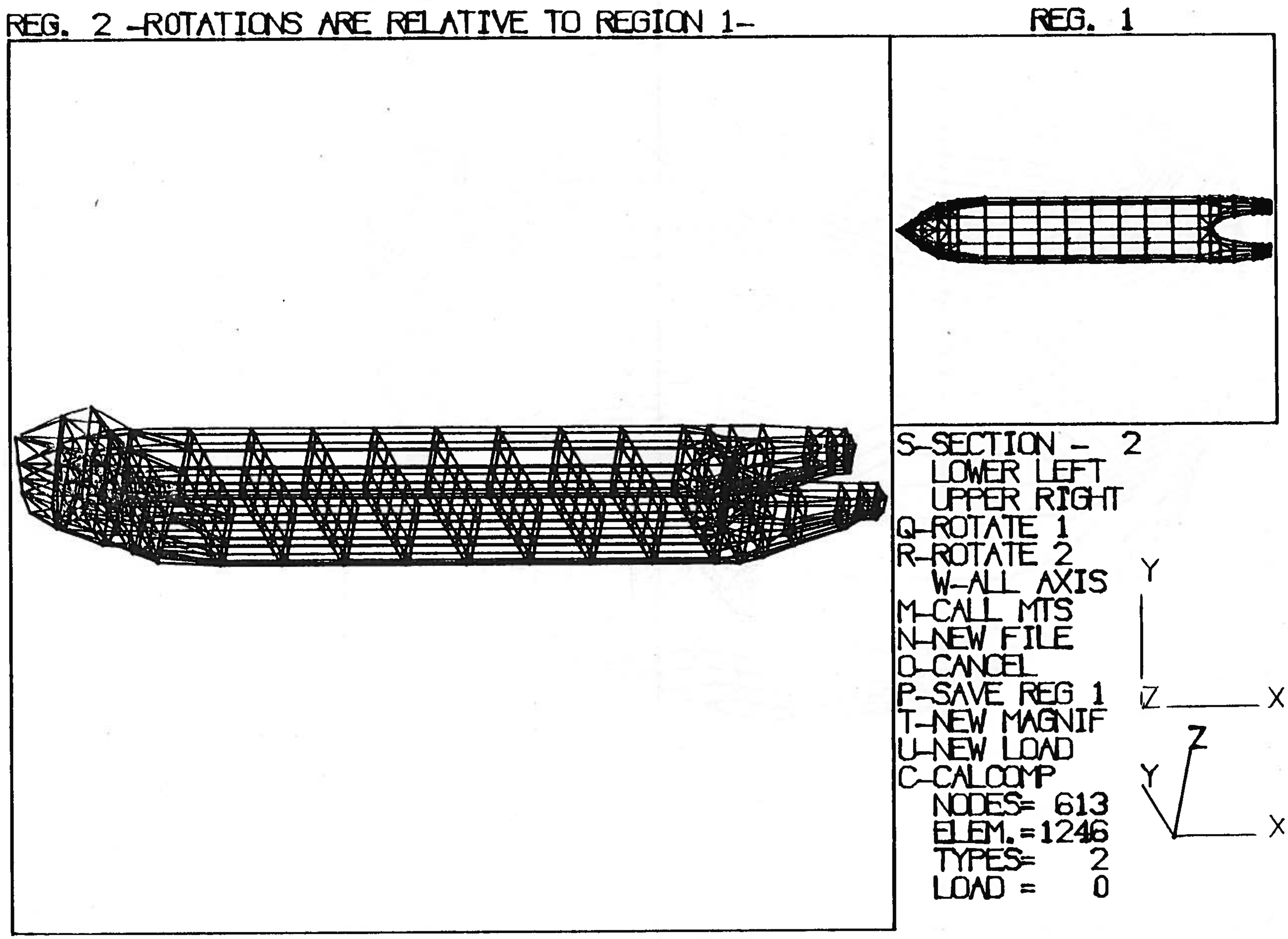
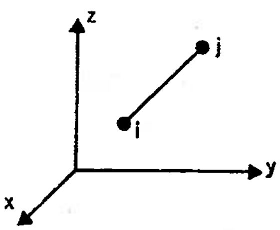
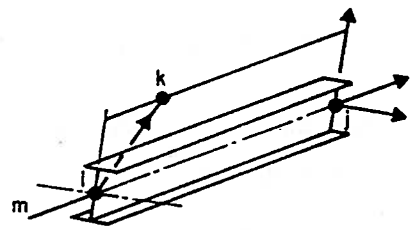


FIGURE 11. Tug Barge System

APPENDIX

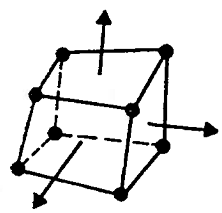


a. TRUSS ELEMENT

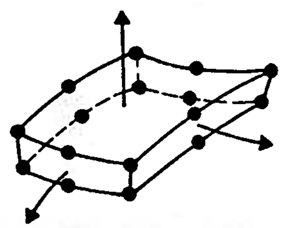


b. THREE-DIMENSIONAL
BEAM ELEMENT

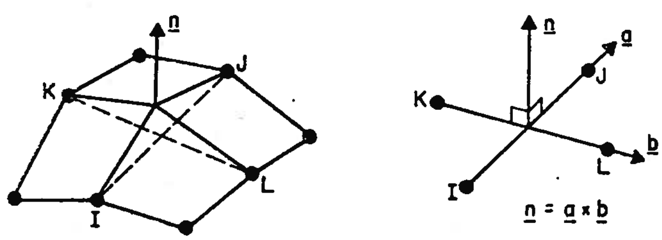
c. PLANE STRESS, PLANE STRAIN AND AXISYMMETRIC ELEMENTS



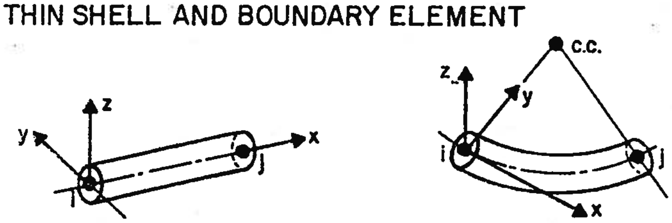
d. THREE-DIMENSIONAL
SOLID



e. VARIABLE-NUMBER-NODES
THICK SHELL AND
THREE-DIMENSIONAL ELEMENT



f. THIN SHELL AND BOUNDARY ELEMENT



TANGENT

BEND

g. PIPE ELEMENT

ELEMENT LIBRARY OF SAP IV

IV. ELEMENT DATA (continued)

TYPE 10- CONTACT (LINK) ELEMENTS

Link elements are identified by the number 9. Axial forces are calculated for each member. The sequence of cards describing the element is given below.

Note: Link elements help calculate:

1. The contact forces between element layers such as a bolt and its nut.
2. The stresses in the structure due to specified crack openings between various element groups such as a wedge driven between a shaft bearing and its supporting structure.

A. Control Card (3I5)

columns 1 - 5 The number 9
6 -10 Total number of link elements
11 -15 Number of different spring stiffness cards

B. Stiffness cards (I5, F10.0)

columns 1 - 5 Stiffness identification number
6 -15 Spring Stiffness (k) (set to 10^{10} if left blank)

There need be as many of the above cards as are necessary to define the stiffness for each element in the structure.

C. Element Data Cards (6I5)

One card per element in increasing numerical order starting with one.

columns 1 - 5 Element number
6 -10 Node number I on first element
11 -15 Node number J on second element (same geometry as node I)
16 -20 Node number K for defining the direction of the spring element - must be on second element or further out, away from point J.
21 -25 Stiffness identification number
26 -30 Optional parameter k used for automatic mesh generation of element data (details of element mesh generation similar to truss).

The last element cannot be mesh generated, must be typed in.

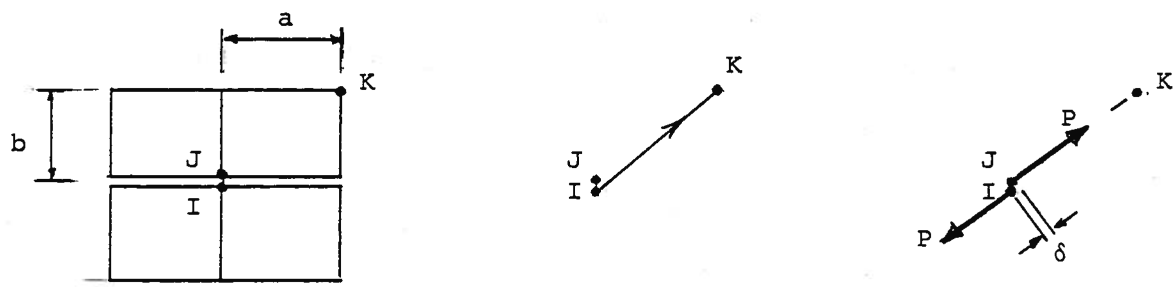
IV. ELEMENT DATA (continued)

NOTES/

(1) Direction of contact (link) element

The direction of the link element I-J is specified by a third model point K which defines the direction of the element from node I to node K.

A positive force in the link element implies that nodes I & J are trying to separate from one another.



(a) Link Element I-J

(b) Element Direction

(c) Prescribed displ.

Note that node K is used only to define the direction of the element and if convenient may be any node used to define other elements. However artificial nodes with all the boundary conditions codes specified as 1 (one), may be created to define directions of link elements.

(2) Prescribed relative displacement

If the link element stiffness (k) is much greater than the stiffness of the ajoining elements, then a small prescribed relative displacement δ between two contact node points can be produced very closely by applying a load $P = k\delta$ at nodes I and J in opposite direction & parallel to I-K as shown in the above figure.