IFEMS, AN INTERACTIVE FINITE ELEMENT MODELING SYSTEM USING A CAD/CAM SYSTEM

Spencer McKellip, Todd Schuman and Spencer Lauer
Sikorsky Aircraft
Division of United Technologies

SUMMARY

General purpose 3D finite element mesh generators, require detailed geometric description of the component to be modeled. For complex shapes, this process can be very time consuming. On the other hand, most general purpose CAD/CAM systems have the capability of defining geometry in an efficient, interactive environment. This paper describes one method of coupling a CAD/CAM system with a general purpose finite element mesh generator.

The problem is one of generality in application. Most interactive mesh generators are either tailor made to specific applications or are restricted to classes of geometry such as bodies of revolution, dragged shapes, 2D geometry or digitized input. What is needed is a completely general purpose, 3D interactive tool that can handle any conceivable geometry without digitizing or any other restriction.

A three dimensional interactive graphic system for defining geometry can be an expensive proposition for just finite element analysis. However such capabilities have been in existence for some time in the form of computer aided design and manufacturing systems, commonly known as CAD/CAM systems. These systems are logical candidates for front ends to a general purpose, 3D finite element mesh generator.

The system described in this paper consists of three programs:

1) CAIDS, A Sikorsky developed CAD system.

2) IFEMS, A Sikorsky developed interactive processor to couple a CAD system with a mesh generator.

3) PWAMESH, A Pratt & Whitney developed mesh generator.

INTRODUCTION

Since the introduction of finite element analysis in the 1960's and thru the 1970's it has become obvious that there was a clear need for powerful mesh generation techniques.

What evolved were two different approaches to mesh generation. One was to construct a model starting with building block elements and some computer aid to place and replicate these elements. The result was to build a model of the desired component or design. The second approach was to start with the component or design and break it first into topologically simple regions which in turn could be automatically broken into elements.

In parallel with this program CAD systems were evolving. These systems provided powerful means for defining complex geometry and storing it in computer readable form.

By 1978 we at Sikorsky were becoming more and more heavily involved in finite element modeling principally with manual methods. The burden of this was unacceptable. In our survey of available mesh generation systems we found no commercially available system which we felt fully exploited the available technology.

It was our opinion that finite element modeling should naturally start from defined design geometry and follow the breakdown method of mesh generation. Further we felt that the power of interactive graphics CAD systems should be fully exploited to obtain the geometry.

We had available to us some elements of the desired system. We had available an in-house developed CAD system called CAIDS (Computer Aided Interactive Design System) and a mesh generator developed at Pratt & Whitney Commercial Products Division which met most of our requirements. With these elements we were well positioned to couple them into an effective system.

IFEMS CONFIGURATION & USE

The total interactive modeling system is actually made up of three separate programs, (Figure 1). The first part is a complete computer aided design system, CAIDS, (Computer Aided Interactive Design System) modified to communicate with IFEMS. CAIDS represents a highly sophisticated interactive design system equipped with a large software library of geometry generating routines. CAIDS hardware is the ADAGE (Figure 2) vector generator terminal with real time dynamic rotation, translation and zoom display features. The CAIDS display space is three dimensional, thus allowing for geometry definition of any conceivable component. Future features will include complete three dimensional math routines capable of solving for such things as intersections of general 3D surfaces.

CAIDS uses 3D parametric rational cubic functions to store its geometry. These functions are stored as labeled groups or OBJECTS which are defined by the user. These OBJECTS become the means by which the analyst ultimately defines the finite element regions for the mesh generator and is the key to IFEMS.

The second program is IFEMS. Its purpose is to accept the component geometry from CAIDS and combine it with the user defined model attributes (number of elements, boundary conditions, loads, etc.) and ultimately prepare an input file for the mesh generator.

A functional overview of IFEMS is presented in Figure 3. The analyst begins by defining the actual component geometry using CAIDS. Upon completion of geometry definition, the analyst divides the model into regions. CAIDS then creates a neutral file which contains the model geometry grouped by OBJECTS. This neutral file is independent of the CAIDS data base and is used as a communications media to IFEMS. For large problems, the analyst may create several neutral files, the sum of which represent the complete model.

Mesh generator regions are defined by using the OBJECT feature of CAIDS. That is, a generic name is defined by the user (region number and type, i.e. 25QUAD) to identify the region. The appropriate geometry defining that region is then assigned to the object.

Phase One of IFEMS is a preprocessor which reads the neutral file(s) performing four basic functions: (1) conducts basic data validation tasks on the region data looking for such errors as missing or non-adjacent edges, (2) reformats neutral file data into in-core arrays suitable for the downstream mesh generator (See Figure 4), (3) creates additional data, (from the cubic functions) required by the mesh generator such as node points representing the corners of the regions and (4) merges in previously defined parts of the model. At the end of Phase One, a permanent file, Perm One, is created (or updated in the case of multipart models) and stored on disk. The Perm One file contains all information generated during Phase One (See Figure 5). Phase One is an interactive processor (run on a Tektronix 4014 terminal) and allows minor user intervention and minor corrections during data validation.

Phase Two Part One of IFEMS is the major section. This is where the actual finite element model region attributes are defined: element type, number of elements per region, element spacing, material definition, boundary conditions, applied load, etc. This program begins by reading the region geometry from Phase One, via the Perm One File.

The program uses a system of 40 menus to list the various options used in defining the region attributes (Figure 6). Through these menus the analyst controls the operational flow of the program. Displayed on the menu is the particular region that the analyst is working on.

The menu system is hierarchical in that a given option automatically envokes a predetermined series of lower level menus (Figure 7 & 8). This system was designed to be tutorial. Given an upper level option, such as defining the number of elements along a region edge, the program will effectively guide the analyst by prompting him for the correct input. On the other hand the user can always interrupt the flow to return to the higher level menus.

A Phase Two session would begin with the menu shown on Figure 9. The analyst will normally use Option D to define default region attributes, then Option E is used region by region to change only those attributes which are different from the default values. Changing the region attributes is accomplished by entering the letter code corresponding to the desired change. Some of these changes are made to the current menu and others will automatically involve lower level menus (See Figures 7 & 8).

The user has additional control over the program through the use of the BAIL, CONTINUE, or NEXT options. BAIL nullifies all current input and reverts to the previously defined values. CONTINUE stores all current information and returns to the previous menu for additional processing. NEXT is used to define the information for the next region or edge.

Disk data sets are used to transfer information between programs. The Perm Two file contains all the region attributes required by the mesh generator. This file is updated each time the anlayst completes a region. After all the regions attributes have been defined the program will compress the Perm Two file and apply the default values to those regions which were not explicitly accessed.

Phase Two, Part Two is basically a pre-prpcessor to the mesh generator, PWAMESH. It creates an input file (from the Perm Two file) containing the region attributes which are then read by the mesh generator. PWAMESH then generates and displays the final mesh. The analyst reviews the final mesh and passes back and forth between Part 1 and 2 until he is satisfied. At this stage, PWAMESH is instructed to generate the bulk data required by the finite element program, typically NASTRAN.

The third program, PWAMESH, is a general purpose three dimensional finite element mesh generator, developed by Pratt & Whitney Aircraft, Commercial Products Division. This program is executed from a Tektronix 4014 CRT. PWAMESH includes such features as variable thickness regions, generalized boundary condition definition, uncoupled regions for sliding mechanisms, automatic generation of quadratic temperature and pressure fields, automatic region joining as well as many other features.

PWAMESH is also equipped with an interactive graphic package for reviewing the finite element model prior to running the finite element code. Its interactive review menu (Figure 10) will plot either the input coarse region(s) or the generated fine region(s) of the model. The entire model or particular regions can be displayed at any viewing angle along with the element and node identification numbers. Offline plots can be drawn using the VERSATEC plotting system. In addition, the user can blow-up any particular section by indicating the lower left and upper right boundaries of the model. The elements of the model can also be shrunk to make individual elements more visible (Figure 11). PWAMESH will prepare a complete finite element input deck for either NASTRAN, MARC or an in-house boundary integral program.

IFEMS SPECIAL FEATURES

IFEMS was designed to minimize the amount of user defined input data. This has been achieved by allowing the user to define a set of initial model attributes. These attributes are applied to each region the first time the user displays the region. The user need only change those attributes that are unique for that region. Typically just the number of elements along an edge, or thickness.

Additional features include automatic continuous backup of user input, free field input, automatic cursor positioning and a full compliment of displayable help pages.

CONCLUDING REMARKS

IFEMS, in conjunction with CAIDS and PWAMESH provide the analyst with a completely interactive tool for modeling mechanical component independent of geometric complexity. The usual errors associated with manually preparing mesh generator input are virtually eliminated as the analyst continually 'sees' the result of his work.

Production use of IFEMS at Sikorsky has resulted in tremendous saving in both man-hours and lead time. Models that previously took three weeks are now being generated in two days. Solving complex problems and performing stress evaluation for minimum weight and cost with practical lead times has resulted in the more efficient use of the analyst's time and knowledge.

A TOTAL INTERACTIVE MODELING SYSTEM

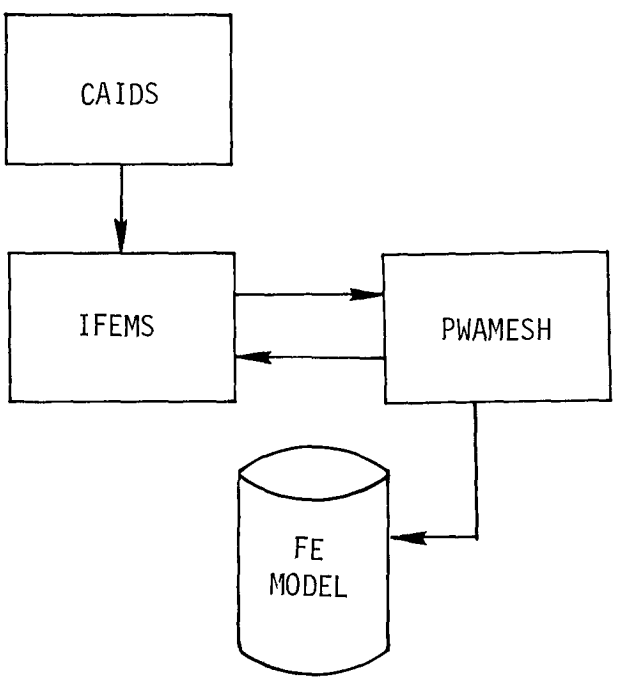


FIGURE 1

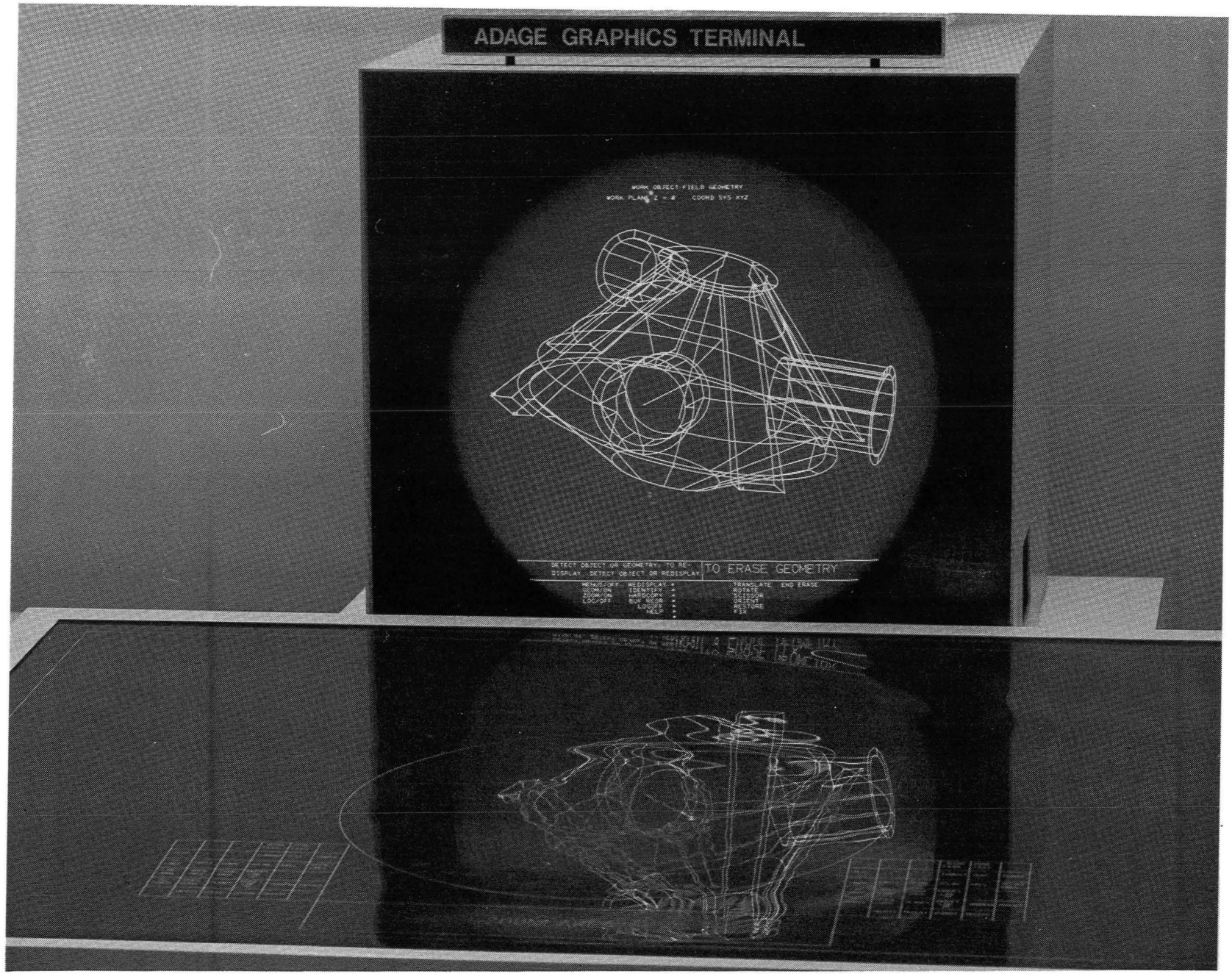


FIGURE 2 - ADAGE Vector Generator Terminal

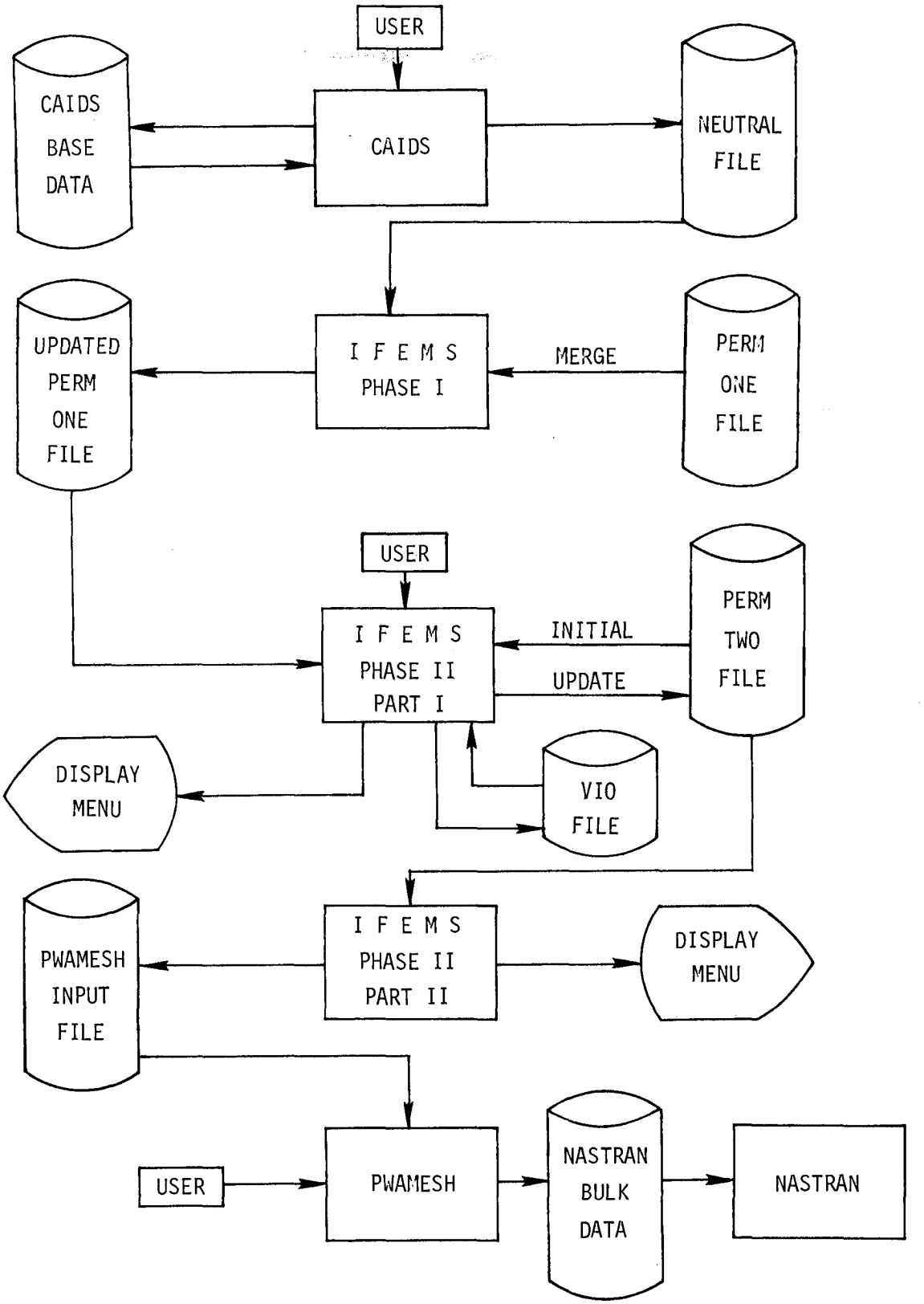

USER
CAIDS BASE DATA
CAIDS
NEUTRAL FILE
UPDATED PERM ONE FILE
I F E M S PHASE I
MERGE
PERM ONE FILE
USER
I F E M S PHASE II PART I
INITIAL
UPDATE
PERM TWO FILE
DISPLAY MENU
VIO FILE
PWAMESH INPUT FILE
I F E M S PHASE II PART II
DISPLAY MENU
USER
PWAMESH
NASTRAN BULK DATA
NASTRAN

FIGURE - 3

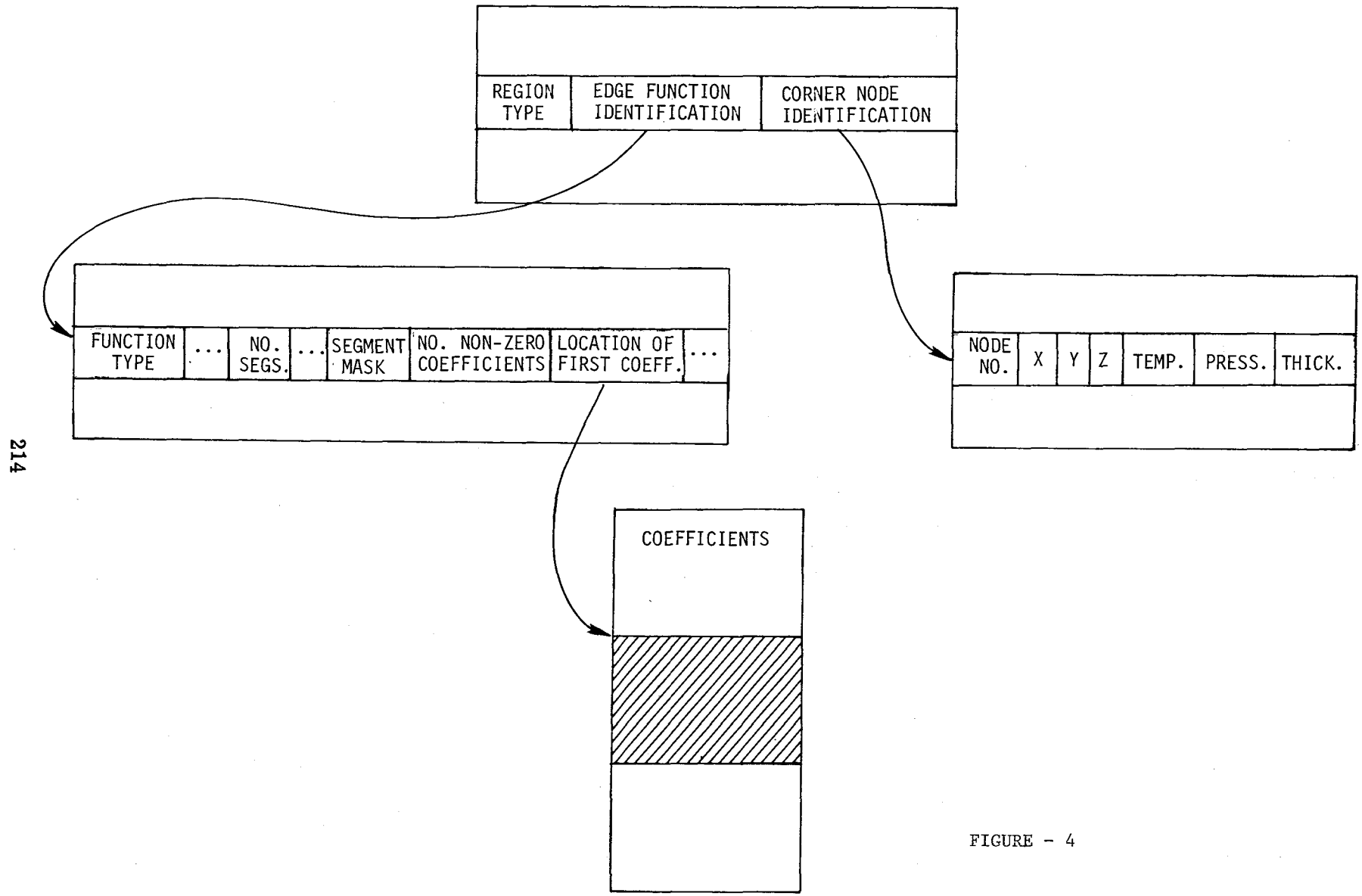

REGION TYPE
EDGE FUNCTION IDENTIFICATION
CORNER NODE IDENTIFICATION
FUNCTION TYPE
...
NO. SEGS.
...
SEGMENT MASK
NO. NON-ZERO COEFFICIENTS
LOCATION OF FIRST COEFF.
...
NODE NO.
X
Y
Z
TEMP.
PRESS.
THICK.
COEFFICIENTS

FIGURE - 4

PERM ONE FILE
STRUCTURE

- REGION DEFINITIONS
- CORNER NODE DEFINITION
- COORDINATE SYSTEM DEFINITIONS
- REGION EDGE FUNCTION DEFINITION & POINTERS
- FUNCTION COEFFICIENTS
- ARRAY PARAMETERS (ie, NO. ENTRIES, ETC.)
- READ/WRITE UNIT NUMBERS

FIGURE - 5

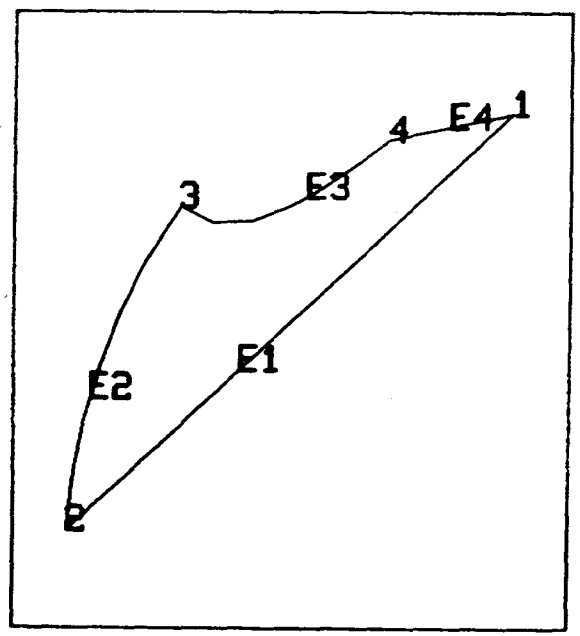
DEFINE ATTRIBUTES FOR REGION 1 (ENTER REGION ID THE FIRST TIME)

SELECT ATTRIBUTE

	INPUT/CURRENT
A-NO. ELS DIRECTION 1	/ 6
D-NO. ELS DIRECTION 2	/ 6
E-NO. ELS DIRECTION 3	/ 6
F-ELEMENT TYPE	CQUAD4
G-ELEMENT SPACING	EQUAL
J-EDGE INTERPOLATION	----------
K-MATERIAL ID	/ 1
L-THICKNESS	1.000

	INPUT/CURRENT
M-TEMPERATURE	0.0
P-PRESSURE	0.0
Q-FACE B.C.(TRANS)	---------
S-FACE B.C.(ROT)	---------
T-SPECIAL FUNCTIONS	---------
U-IGNORE ELEMENTS	---------
W-REGION B.C.	---------
X-OUTPUT DISP SYS	/ 0
Y-PERM CONSTAINTS	/ 0

V-VIEW CHANGE
N/C = NOT CONSTANT



B-BAIL C-CONTINUE H-HELP N-NEXT REGION R-REDISPLAY MENU

FIGURE - 6

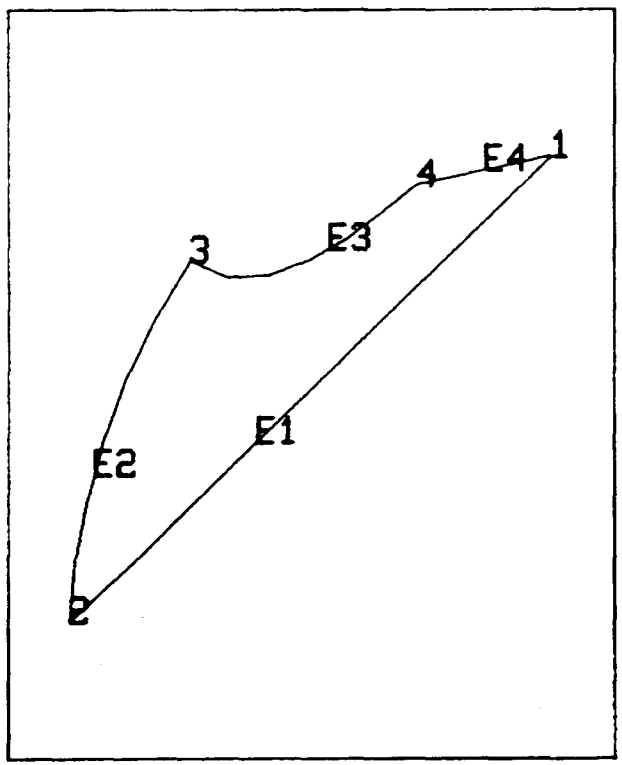
ELEMENT SPACING FOR REGION 1

QUAD REGION - PERMISSIBLE EDGES ARE
1,2,3 OR 4

ENTER EDGE NUMBER

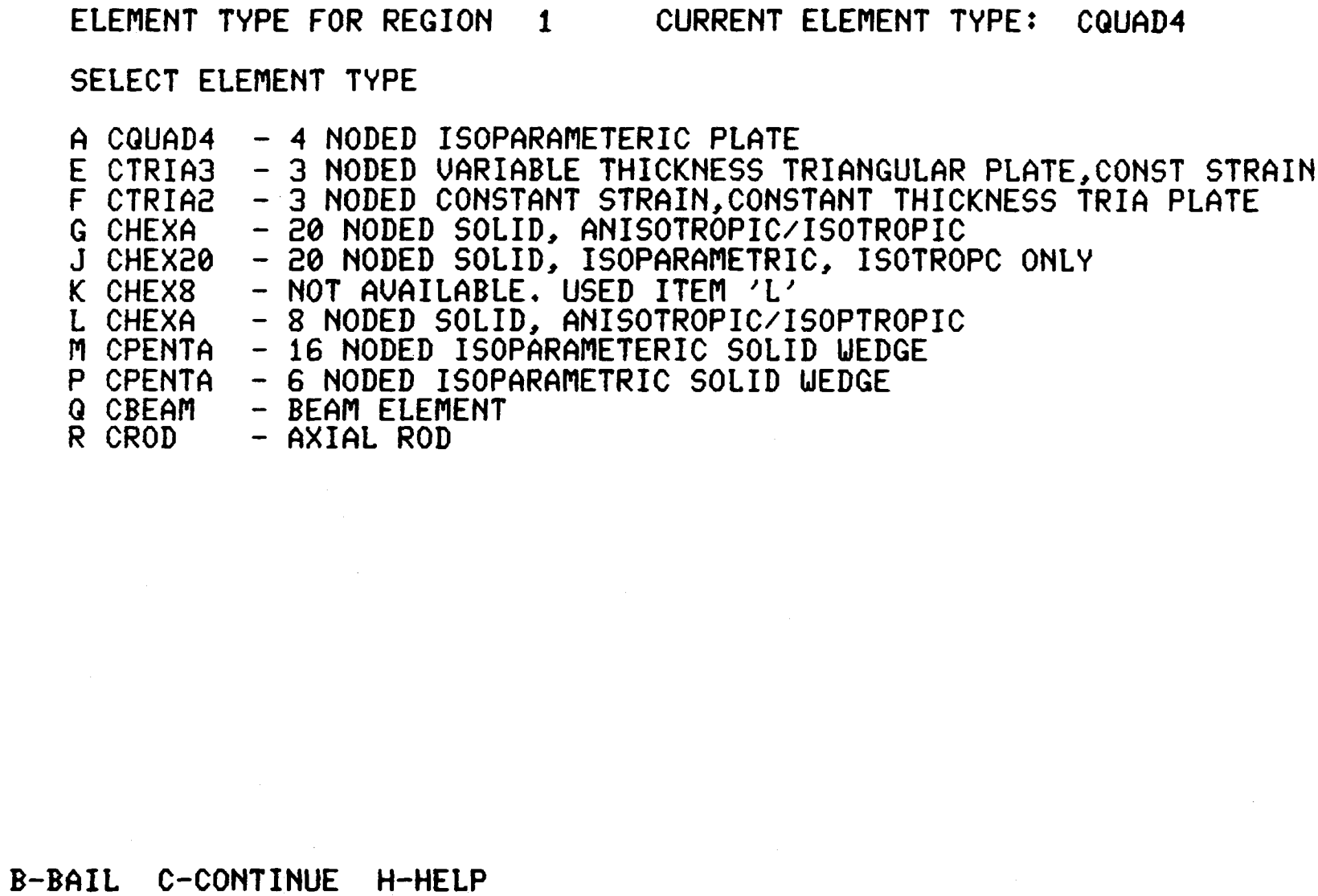
SELECT SPACING OPTION

A UNEQUAL
D GRADIENT
E DISPLAY CURRENT VALUES
F ENTER GRADIENT HERE
G SAME AS EDGE   OF REGION
 (NOTE: BE SURE THIS EDGE EXISTS,
  OTHERWISE PWAMESH WILL FAIL)



SELECT
B-BAIL C-CONTINUE V-VIEW CHANGE H-HELP N-NEXT EDGE

FIGURE - 7



```
ELEMENT TYPE FOR REGION  1      CURRENT ELEMENT TYPE:  CQUAD4

SELECT ELEMENT TYPE

A CQUAD4  - 4 NODED ISOPARAMETERIC PLATE
E CTRIA3  - 3 NODED VARIABLE THICKNESS TRIANGULAR PLATE,CONST STRAIN
F CTRIA2  - 3 NODED CONSTANT STRAIN,CONSTANT THICKNESS TRIA PLATE
G CHEXA   - 20 NODED SOLID, ANISOTROPIC/ISOTROPIC
J CHEX20  - 20 NODED SOLID, ISOPARAMETRIC, ISOTROPC ONLY
K CHEX8   - NOT AVAILABLE. USED ITEM 'L'
L CHEXA   - 8 NODED SOLID, ANISOTROPIC/ISOPTROPIC
M CPENTA  - 16 NODED ISOPARAMETERIC SOLID WEDGE
P CPENTA  - 6 NODED ISOPARAMETRIC SOLID WEDGE
Q CBEAM   - BEAM ELEMENT
R CROD    - AXIAL ROD

B-BAIL  C-CONTINUE  H-HELP
```

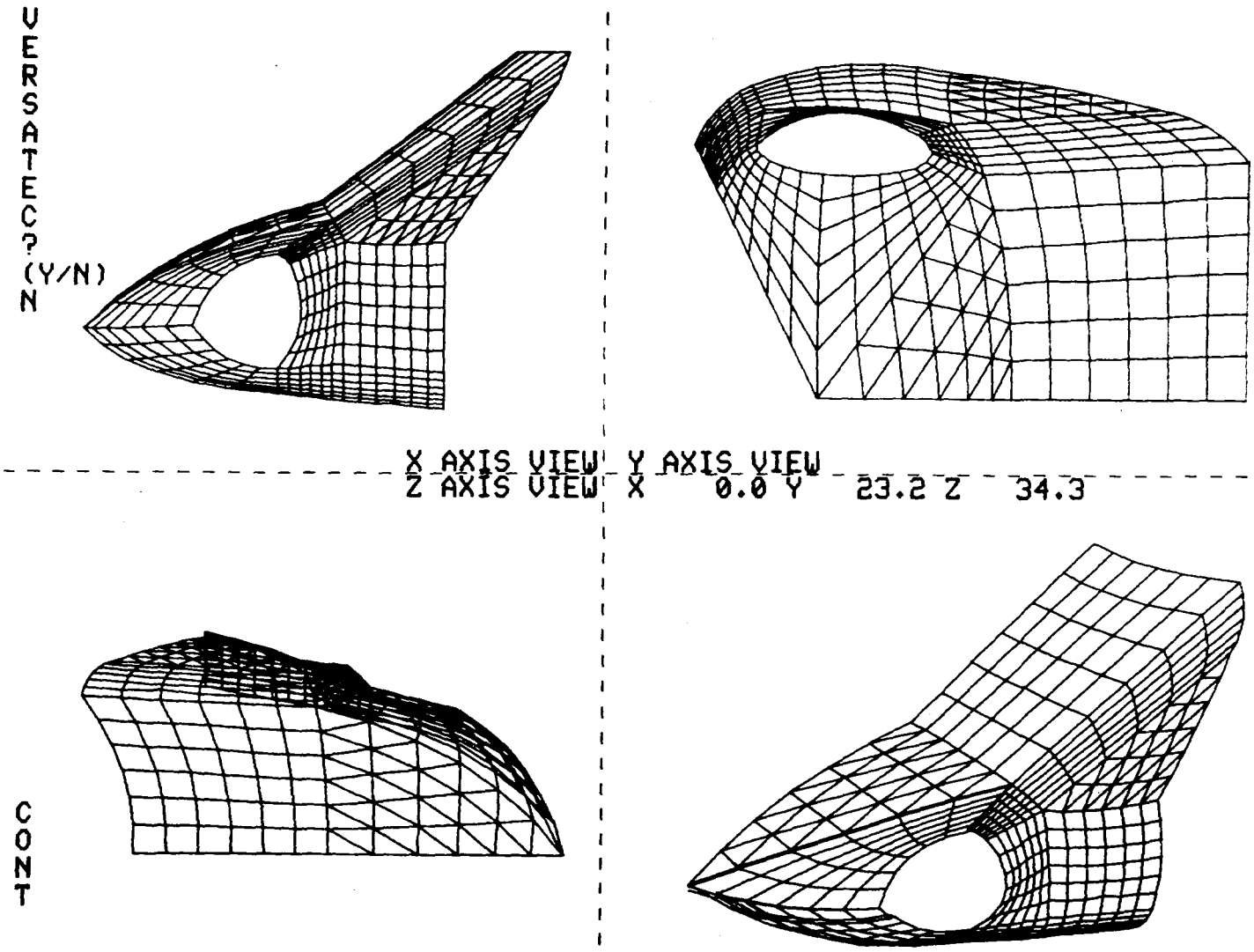
FIGURE - 8

*** I F E M S ***

MAIN MENU

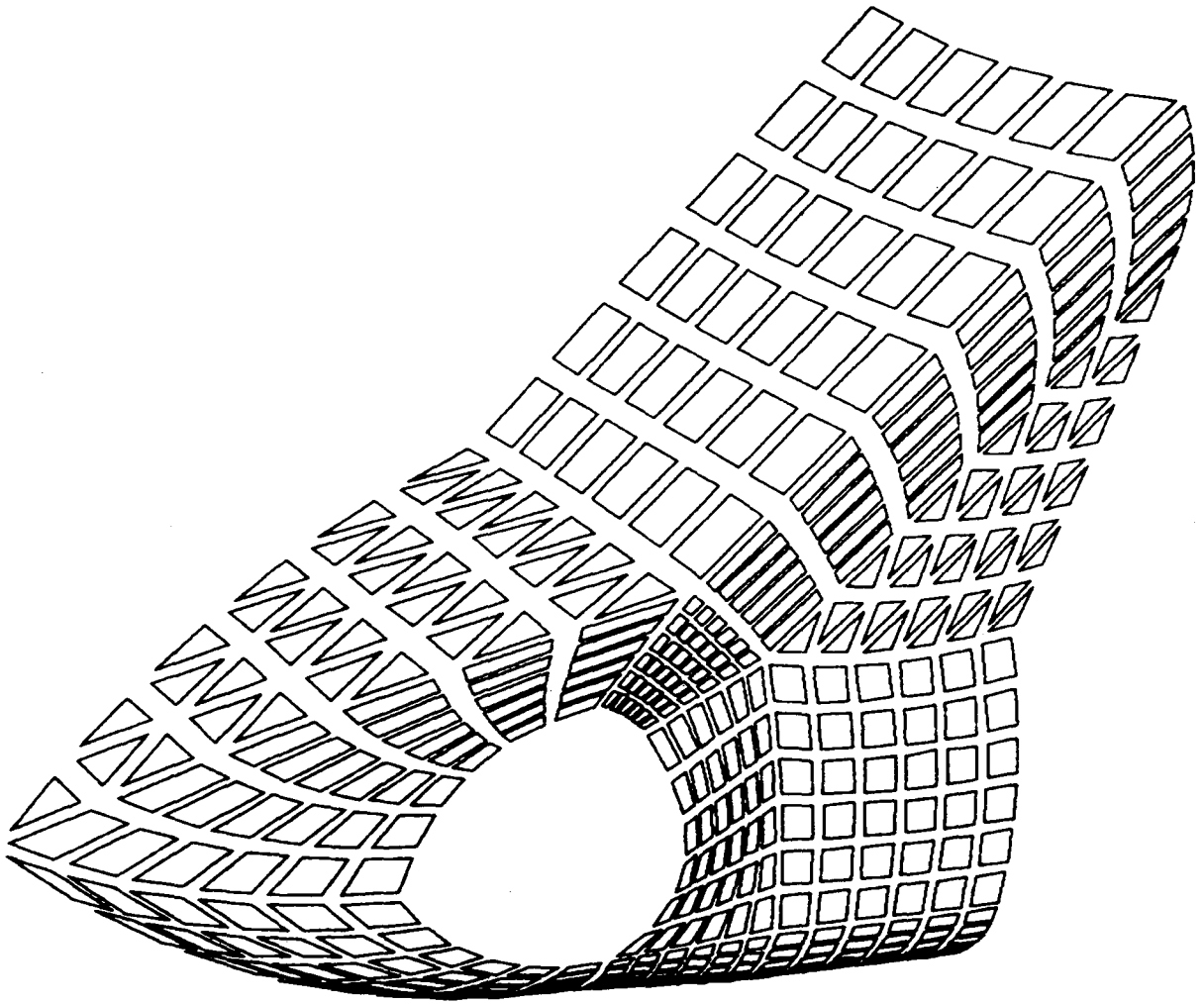
SELECT ITEM		OPTION PREVIOUSLY SELECTED
D	DEFINE REGION DEFAULT VALUES	( )
E	DEFINE INDIVIDUAL REGION ATTRIBUTES	( )
F	DEFINE MATERIAL PROPERTY(S)	( )
G	GENERATE PWAMESH INPUT	( )
J	LIST REGION ATTRIBUTES FOR REGION -	
K	GENERAL OPERATING INSTRUCTIONS	
T	TERMINATE SESSION (UPDATE PERM 2 FILE)	
H	HELP	
X	DEBUG TRACE	OFF
U	ABORT PHASE II	
V	LIST COMPLETED REGIONS	

FIGURE - 9



SELECT:
1 4-VERSATEC
2 12-VERSATEC
3 ADD NODE NO
4 BLOW-UP
5 BLOW-UP + NODE NO
6 BLOW-UP + FIND NODE NO
7 EXPLODED VIEW
8 ADD ELM NO
9 BLOW-UP + ELM NO
10 OPTION 5+9
11 CONTINUE
ENTER-

FIGURE - 10



SELECT:
1 ADD NODE NO
2 BLOW-UP
3 BLOW-UP + NODE NO
4 BLOW-UP + FIND NODE NO
5 ADD ELM NO
6 BLOW-UP + ELM NO
7 OPTION 3+6
8 CONTINUE
ENTER-

X 0.0 Y 23.2 Z 34.3

FIGURE - 11