ABSTRACT:

An interactive graphics model generation system, SDRC SUPERTAB, will be discussed to demonstrate its usefulness in cost-effective preparation of finite element models for MSC/NASTRAN analyses.

A specific application will be outlined pertaining to work performed by Structural Dynamics Research Corporation an a space shuttle component.

INTRODUCTION:

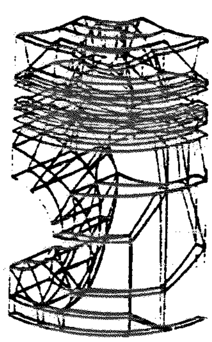
During the past decade and a half, a large number of general purpose computer programs for performing finite element analysis have become available to structural engineers, including such familiar codes as MSC/NASTRAN, ANSYS, and SDRC SUPERB. While these programs concentrated on solving the appropriate stiffness equations, more recent software has been written to remove the burden from the user of preparing input data and interpreting calculated results.

Until recently, actual computer processing represented only about 20% of the total cost and time involved in carrying out a finite element analysis. The remaining 80% was consumed in tedious data handling tasks.

To address this problem, SDRC has developed and enhanced the SDRC SUPERTAB program as a means of automating the data generation procedure, thereby improving the productivity of modeling personnel. Documented examples, such as those shown below, have proven that engineers can more than double their productivity when using SUPERTAB to prepare finite element models.

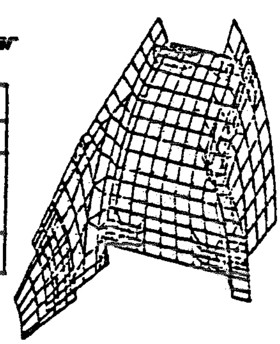
Diesel Piston — *1052 nodes . . . 151 parabolic solids*

Operation	Man Days	
	By Hand	SUPERTAB
Generate Model	10	8
Model Checking	4	—
Total	14	8



Industrial Fan — *466 nodes . . . 451 linear shells*

Operation	Man Days	
	By Hand	SUPERTAB
Generate Model	7	4
Model Checking	3	—
Total	10	4



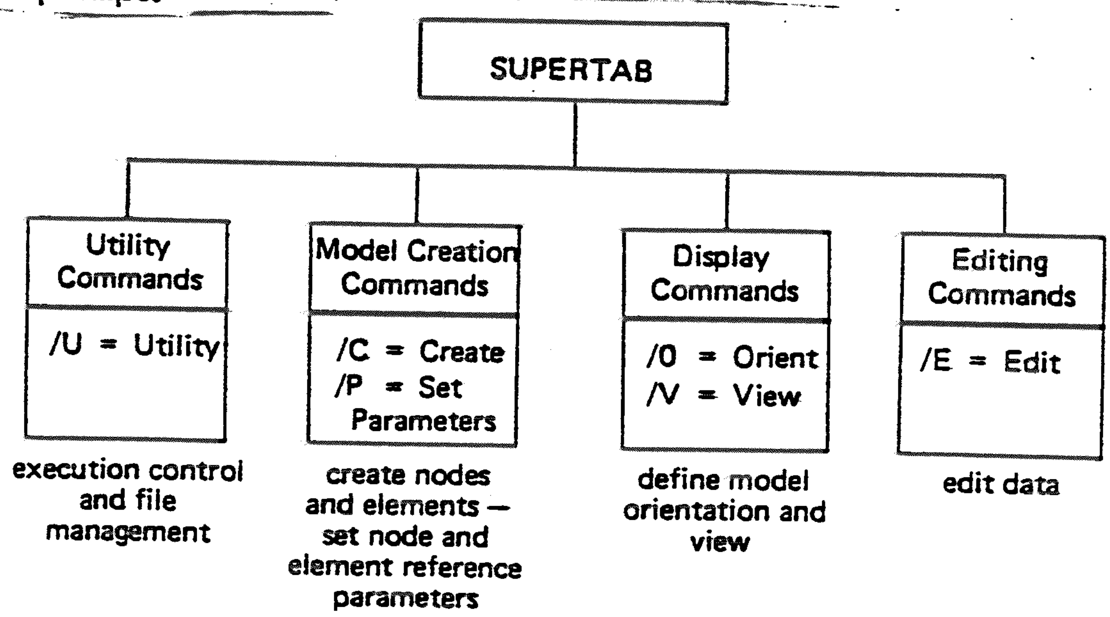
This productivity improvement can be applied to virtually all finite element codes, whether or not the codes themselves have data generation schemes inherent in them. By combining the digitizing, editing, and display capabilities of SUPERTAB, maximum state-of-the-art productivity can be achieved.

SDRC SUPERTAB:

The SDRC Supertab program is a general computerized graphical tool for preparing bulk data to be used by finite element computer programs for computerized design analysis of complex structures and mechanical systems. The purpose of SUPERTAB is to provide a convenient and low-cost method for automating the data entry cycle of the finite element modeling process. In its present configuration, SUPERTAB can be used to generate bulk data for the SDRC SUPERB, NASTRAN, and ANSYS. SUPERTAB operates in a universal data format which becomes the common reference format from which unique formats are written for each application program. Other application programs can easily be supported by SUPERTAB by simply adding the appropriate formatting modules.

SUPERTAB provides the capability to create, display, and edit finite element models using an interactive CRT display terminal and digitizing tablet. The program is designed to handle both two and three dimensional models using conventional or isoparametric elements. After specifying a physical orientation and view for the model, nodes are entered in a data base by digitizing locations on the surface of the tablet or by typing data directly from the keyboard. Elements can then be defined by selecting nodes on the graphic display screen, the digitizer tablet, or through the user of data generation commands. Flexible display and editing features are available for checking and modifying the model. The data base prepared by SUPERTAB may be written in a universal free-field format or one of several standard formats supported by the program.

The capabilities available in SUPERTAB are segmented into four basic functions as shown below, along with the command abbreviations for each function. Associated with each command is a variety of operations and sub-operations which are easily arranged for quick engineering throughput. Commands are entered directly from the keybroad or selected from a refresh menu prompt.



Utility commands have been included in SUPERTAB to perform basic control and file management operations. Operations used to halt execution of SUPERTAB and restart from a previous execution are provided. Utility operations are also used to Save, Read and Write the two data files used by the program.

Both node and element information are created using Model Creation commands. Node information consists of the node number and three-dimensional location of each node. Nodes can be individually defined or automatically generated using Cartesian, cylindrical, or spherical reference axes. The sourse of node information can be:

1. Digitized locations from drawings placed on the tablet (including "two-view" digitizing),

2. Keyboard entry of coordinates, or

3. A data file previously created.

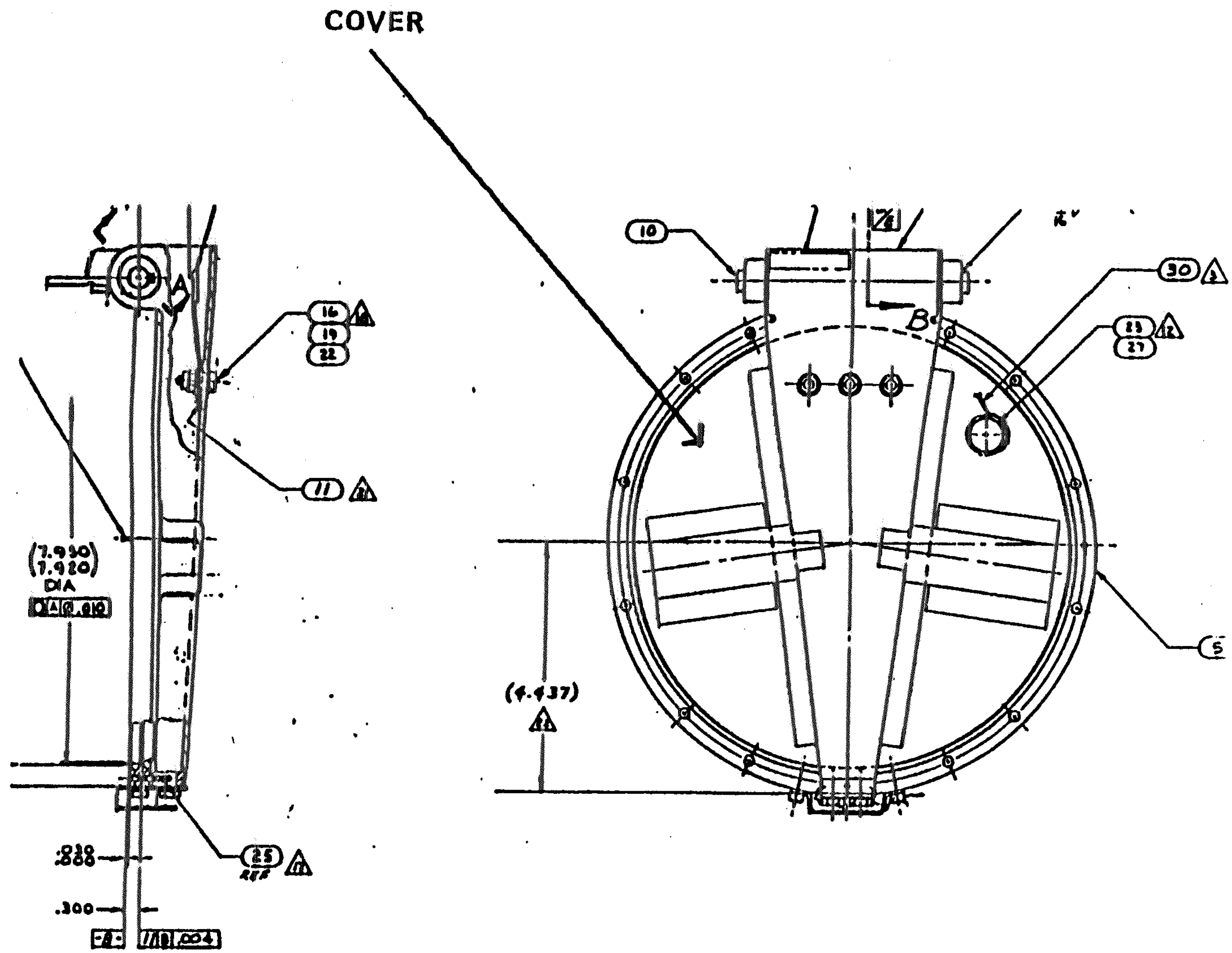
Element information consists of Element Tag, Element Type, Physical Property and Material Property as well as the Node Numbers which comprise the connectively list.

Display commands are used to specify the spatial orientation of three-dimensional objects on the digitizing tablet and to control the view of the objects as displayed on the graphic display screen. Capabilities are provided for general display control including Shrinking, Scaling, Windowing, Rotating, Translating and Centering of the display. As many as four views may be displayed and operated upon simultaneously.

Editing commands are used to edit the model data file after nodes and elements have been created. Editing operations have been included for displaying selected segments of the physical model, purging extraneous or incorrect data from the model file, modifying data, listing data and labeling selected nodes or elements. All Editing operations reference selected node or element groups called Editgroups. Editgroups are designated by the appropriate label, i.e., "N" for node and "E" for element followed by the node or element Tags to be included in the group. These commands are valuable aids to the user for locating and correcting modeling errors.

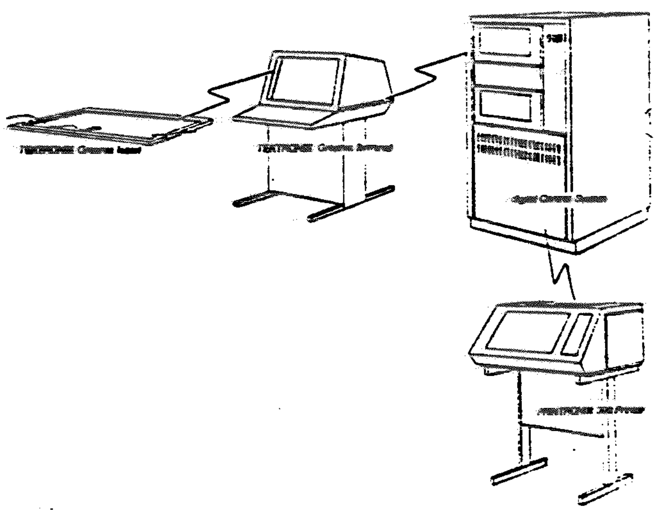
THE PROBLEM:

Structural Dynamics Research Corporation (SDRC*) was contracted to analytically predict the structural integrity of the Space Shuttle component when it is exposed to the specified random accelerations and accoustic pressure. This analysis was undertaken so as to obtain a more complete understanding of the dynamic response of this component than had been obtained from tests and previous analyses. One questions to be resolved was the excitation of modes of the cover that could transmit or complify dynamic loads to adjacent components. Another was the expected fatigue life given the measured and predicted strain cycles. Simultaneously, the effect of the thermal environment was yet to be determined.


COVER
10
16
19
22
11
30
23
27
5
(7.930
7.920)
DIA
(4.437)
25
REF
.030
.000
.300

SUPERTAB APPLICATIONS:

Because of the complexity of the component being analyzed and the time saving capabilities of the SDRC finite element modeler, SUPERTAB was chosen for constructing the model. The program was utilized in conjunction with Tektronix Graphics Terminal hardware and the DEC 11/34 minicomputer at the SDRC facilities in San Diego. The typical system for SUPERTAB usage is depicted below:



SUPERTAB MODELING STEPS:

1. Digitize and generate component nodes
2. Input and generate component elements
3. Perform model checking via:

   - Viewing Commands
   - Labeling Commands
   - Windowing Commands
   - Shrinking Commands

4. Correct nodal or elemental errors via editing commands.
5. Display final viewing before next analysis step.

SUPERTAB RESULTS AND FINALIZATION OF MODEL:

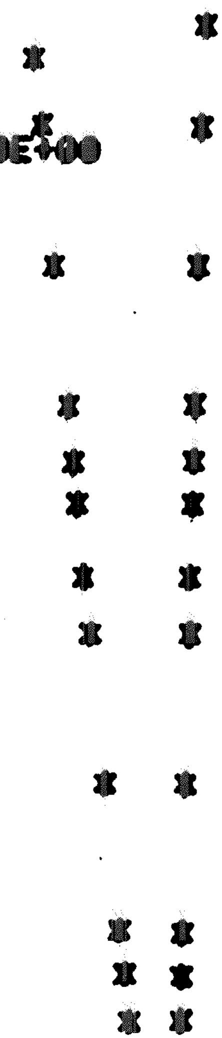
Upon completion of the data generation for nodes and elements in the components, additional peripheral data must be inserted. This includes restraints, material properties, loading, etc. and can easily be input through data set editing or specific pre-processing software.

By using SUPERTAB, SDRC engineers were able to compute a formidible modeling task in the span of only two days. A problem such as this could easily have taken an engineer two weeks to complete without the luxury of this finite element processing package.

DIGITIZING NODES

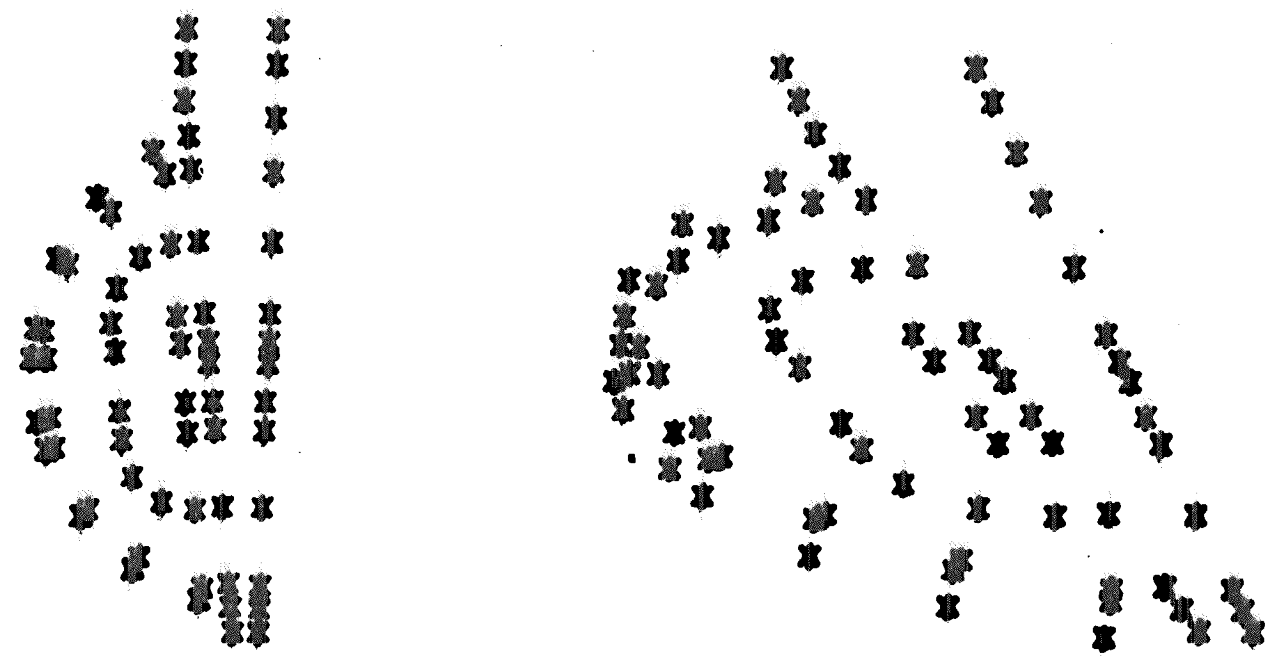
```
/C$/0,PL
ENTER COORD SYSTEM$0
DIG PT  1
DIG PT  2
DIG PT  3
ENTER X,Y,Z-PT  1$0
ENTER X,Y,Z-PT  2$3.96
ENTER X,Y,Z-PT  3$0,3.96
/C$CK
ENTER COORD SYSTEM$0
DIGITIZE CHECK POINT
CHECK POINT X,Y,Z IS-3.955E+00-1.000E-02 0.000E+00
DIGITIZE CHECK POINT
/U$/U,UT
DIG LOWER LEFT
DIG UPPER RIGHT
/U$/C,N,T
SINGLE VIEW
DIG NODE  1
DIG NODE  3
DIG NODE  5
DIG NODE  7
DIG NODE  9
DIG NODE  11
DIG NODE  13
DIG NODE  15
DIG NODE  17
DIG NODE  19
DIG NODE  21
DIG NODE  23
DIG NODE  25
DIG NODE  27
DIG NODE  29
DIG NODE  31
DIG NODE  33
DIG NODE  35
DIG NODE  37
DIG NODE  39
DIG NODE  41
DIG NODE  43
DIG NODE  45
DIG NODE  47
```

```
DIG NODE  49
/C$DR;N,T
```



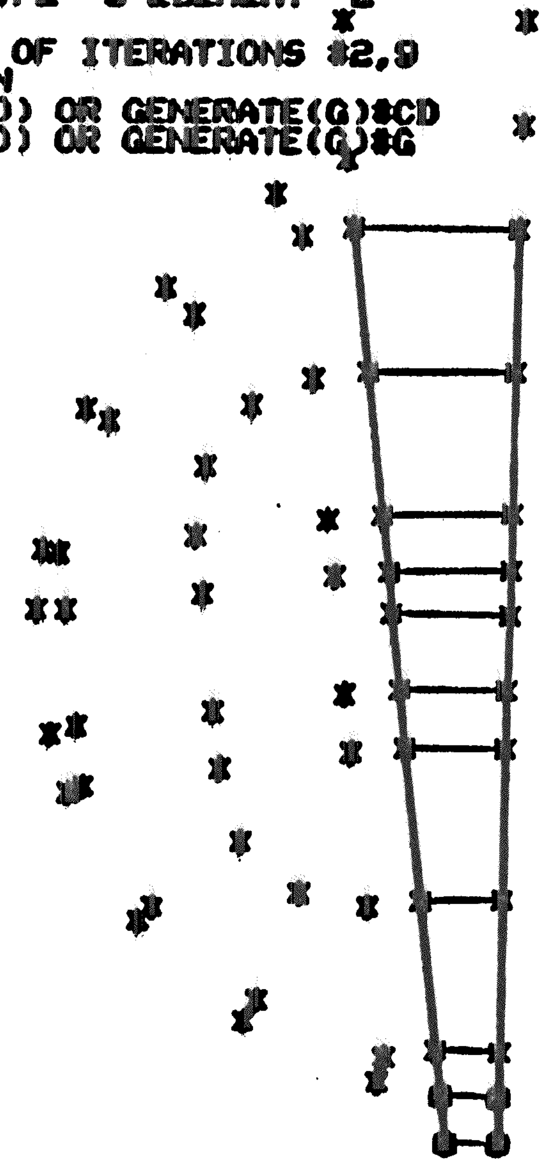
GENERATING NODES

```
YE$/C,G,N,61,70
ENTER NODE NO. INC$100
ENTER ITERATIONS$1
COORD SYS  0 ENTER DELTA X,Y,Z$0,0,-.3
ENTER CHECK DISPLAY(CD) OR GENERATE(G)$CD
ENTER CHECK DISPLAY(CD) OR GENERATE(G)$G
  10 NODES GENERATED
/C$
```

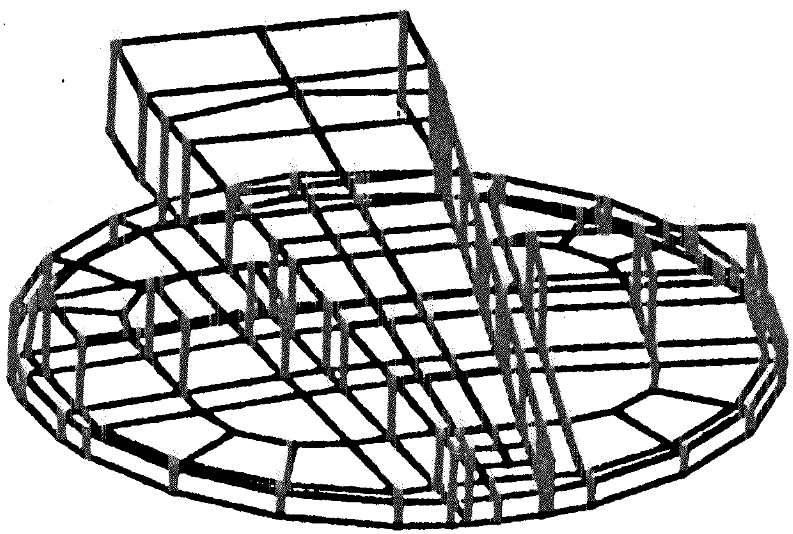


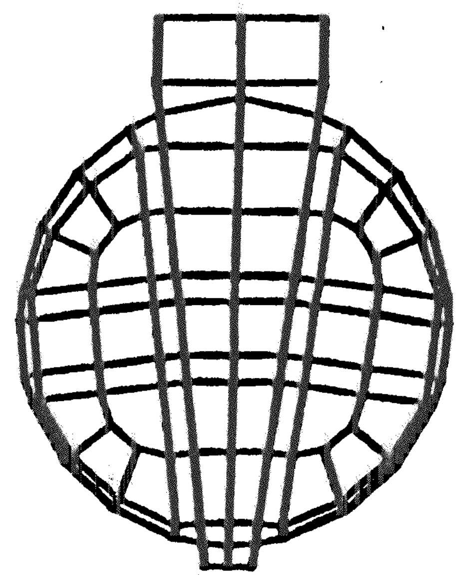
GENERATING ELEMENTS

```
/C#E,C
DETECT   4 NODES FOR TYPE  3 ELEMENT  1
DETECT   4 NODES FOR TYPE  3 ELEMENT  2
/C#,E,1
ENTER NODE INC,NUMBER OF ITERATIONS #2,9
   9 ELEMENTS TO BE GEN
ENTER CHECK DISPLAY(CD) OR GENERATE(G)#CD
ENTER CHECK DISPLAY(CD) OR GENERATE(G)#G
   9 GENERATED
/C#
```



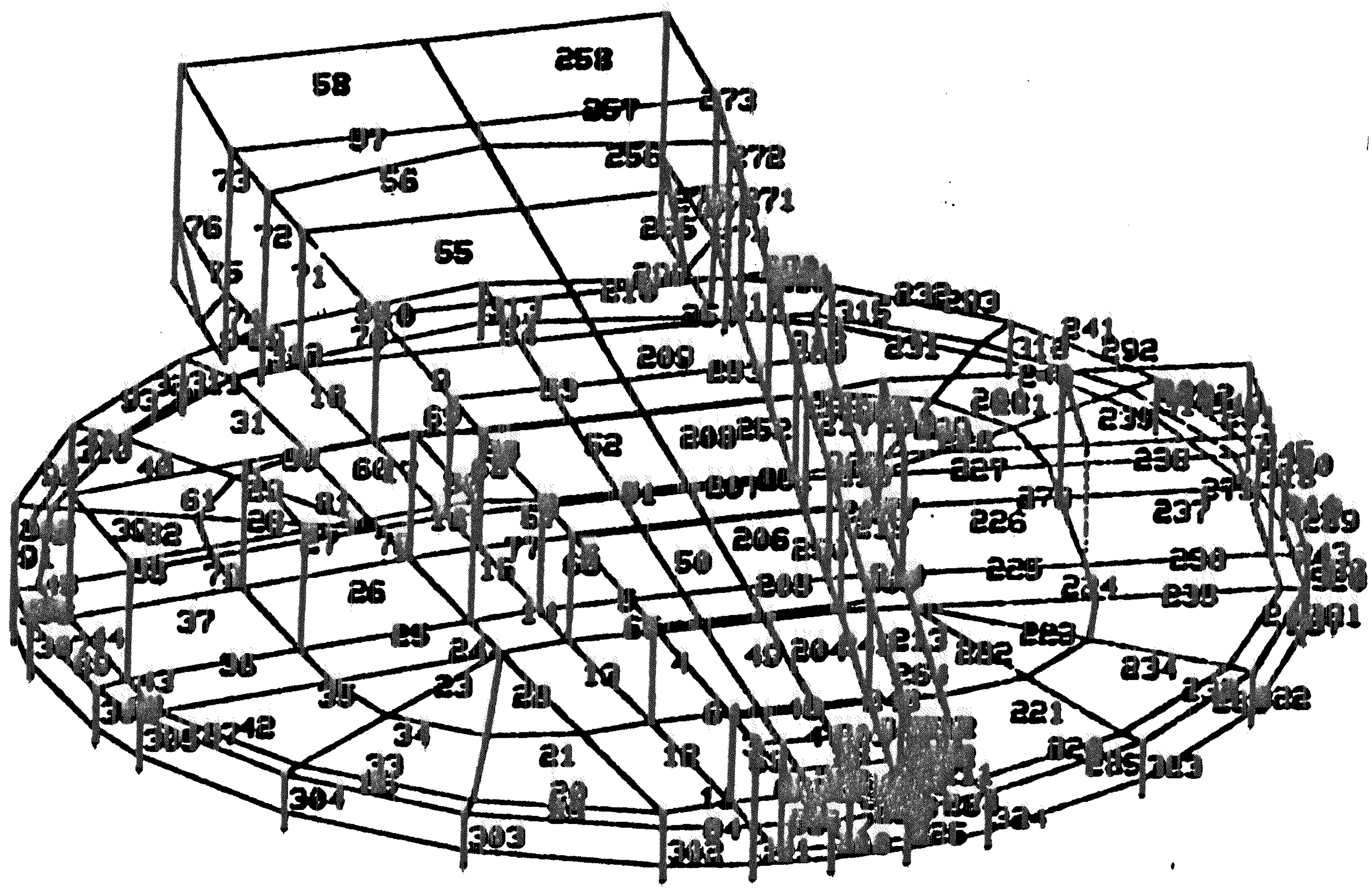
VIEWING





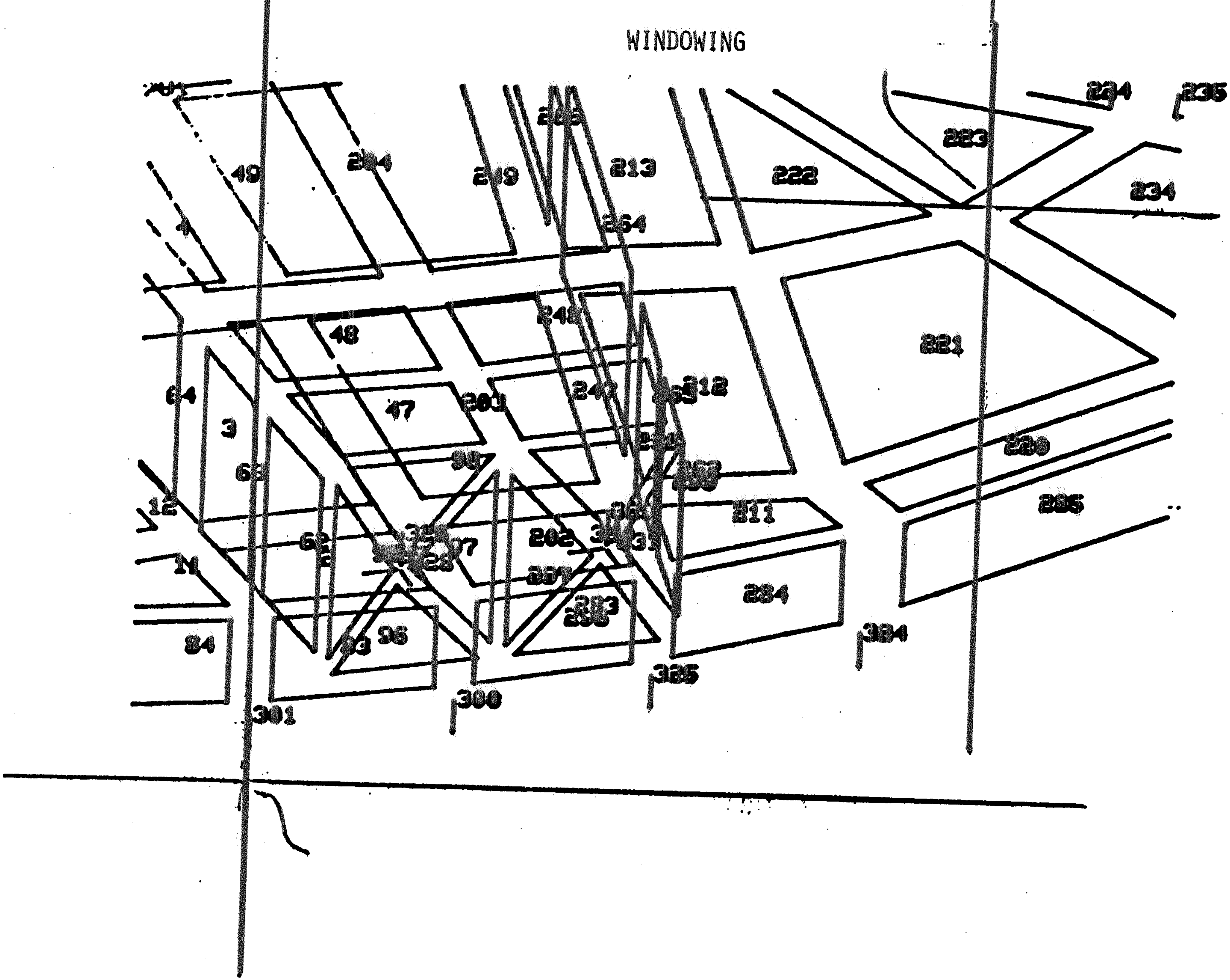
%P1

LABELING



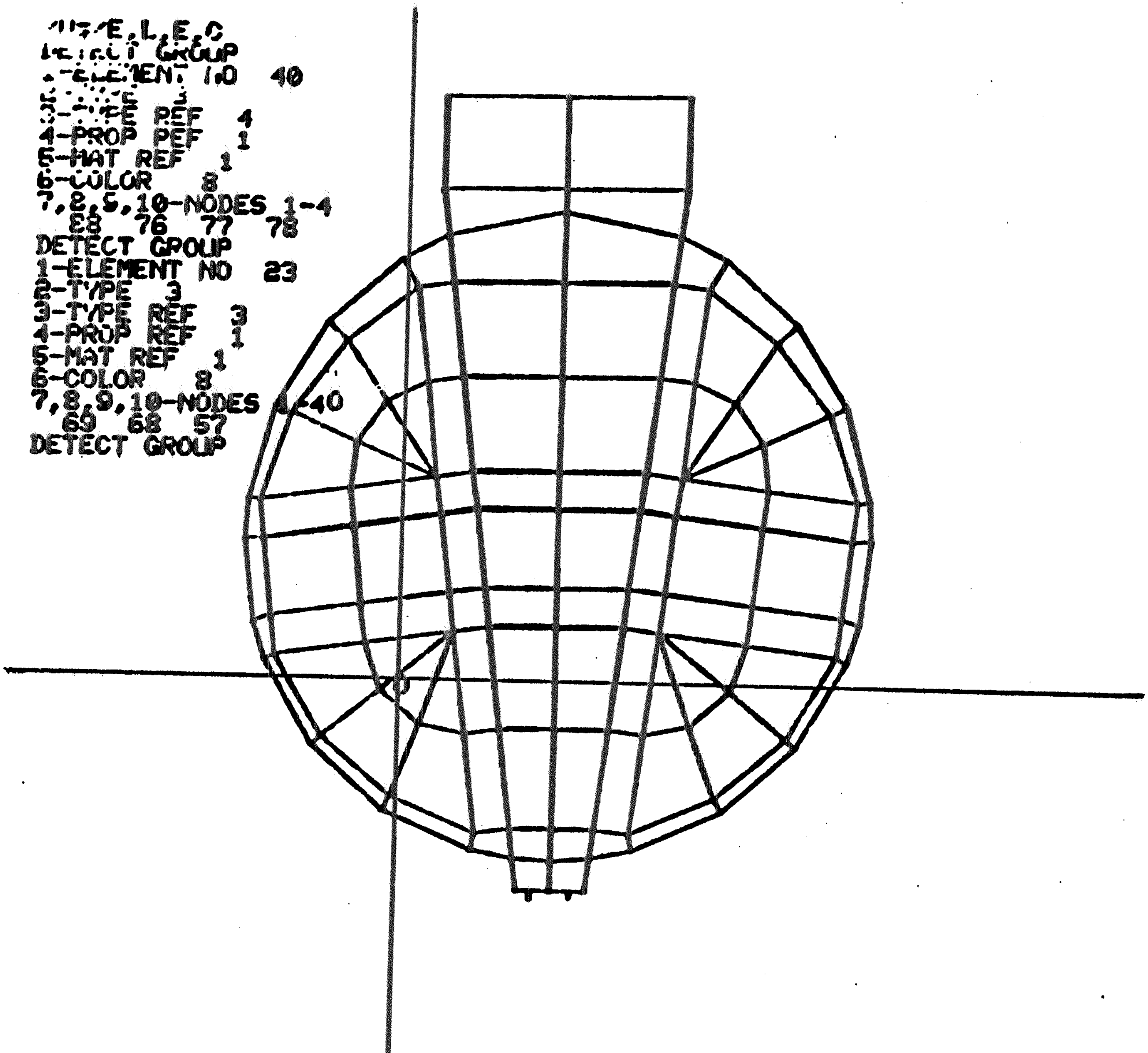
58
258
273
97
272
73
56
256
271
76
72
55
75
71
232
203
315
241
292
231
209
31
60
62
208
227
238
61
237
226
225
290
224
230
26
37
50
206
282
234
221
23
21
42
34
303
304

WINDOWING



49
204
249
213
222
223
234
235
264
248
48
221
64
47
242
312
3
220
63
205
211
12
202
284
62
14
84
96
304
325
300
301

EDITING



E,L,E,C
DETECT GROUP
ELEMENT NO 40
3-TYPE REF 4
4-PROP PEF 1
5-MAT REF 1
6-COLOR 8
7,8,9,10-NODES 1-4
88 76 77 78
DETECT GROUP
1-ELEMENT NO 23
2-TYPE 3
3-TYPE REF 3
4-PROP REF 1
5-MAT REF 1
6-COLOR 8
7,8,9,10-NODES 1-40
69 68 57
DETECT GROUP

FINAL VIEWING