LapCAD
a MSC/NASTRAN and MSC/pal2™ front end on the Personal Computer

Gert M. Lundgren

LAPCAD Engineering
1568 Diablo Point Court
Chula Vista, California 91911, USA
(619) 421-1236 or (619) 467-1947

ABSTRACT

LAPCAD Engineering has upgraded their Macintosh version of the LapCAD finite element modeler to handle large element and node numbers, with the MSC/NASTRAN user in mind. Also included is an IGES interface, and enhanced solids meshing.

Two versions of LapCAD are available, one for the Macintosh Plus, Portable, SE, Classic, LC and Si. The other version supports the Macintosh II, IIx, IIcx, IIci and IIfx. Similar version are under development for the IBM compatible personal computers. The IBM-compatible versions will utilize a graphical user interface that is similar to the Macintosh version, including pull-down menus, tool windows and descriptive icons.

The following example illustrates how LapCAD was used for the creation of a finite element model of an automotive suspension part. This model was output in the MSC/NASTRAN format, the primary file format of LapCAD. The model was transferred to a VAX mainframe, and the resulting F06 file was taken back to the Macintosh and postprocessed in LapCAD. The model solution could have also been obtained via MSC/pal 2, on the Macintosh, since the model has fewer than 2000 nodes. LapCAD allows creation of models up to 10,000 nodes or 60,000 degrees of freedom.

INTRODUCTION

LapCAD creates an entire model, including the mesh, element and material properties, and loads and constraints. For MSC/NASTRAN it also creates the Executive and Case Control Sections, as well as the Bulk Data. The Executive and Case Control Sections from other MSC/NASTRAN files can also be intercepted and merged.

In addition to LapCAD's own modeling features, it can also utilize geometry from CAD programs in the DXF and IGES formats. LapCAD also computes the weight and c.g. of an object, as well as properties and stresses for arbitrary cross sections.

In the example below, LapCAD's modeling capability is used to create the model geometry for an automotive suspension part, that consists of line and surface elements as well as solids. Next the boundary constraints and external forces are applied.

Finally the F06 MSC/NASTRAN output file is utilized for postprocessing, including the display of shaded stress contours and animation of the deformed shape. In addition an automatic search of the output file is done, by simply clicking with the mouse on selected nodes and elements. This feature allows the analyst to display in a separate window all the output relating to the selected model object. The content of this query window, as well as the model itself, can be copied and pasted into a word processor.

LapCAD's Graphical User Interface, shown in Figure 1.

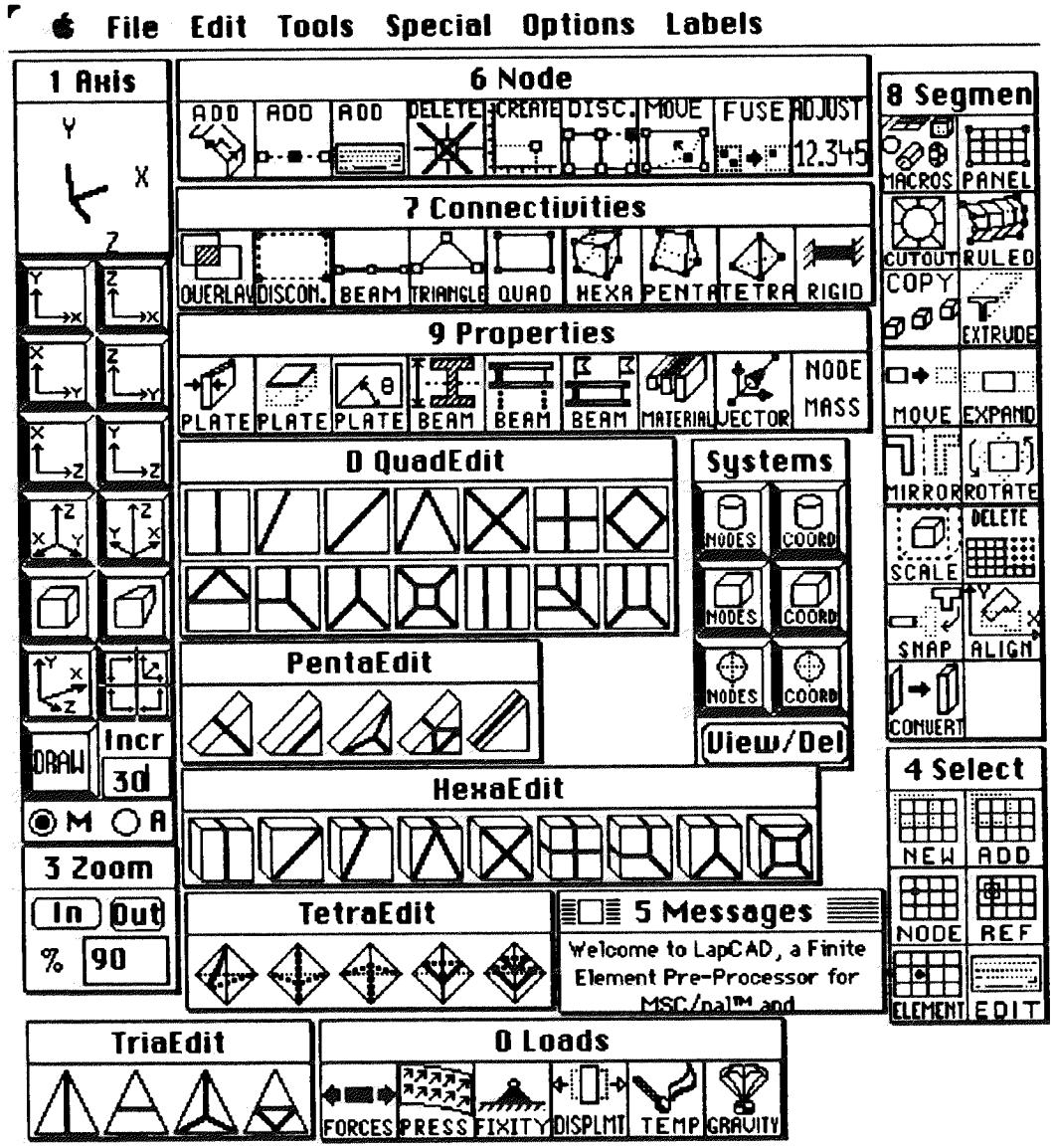
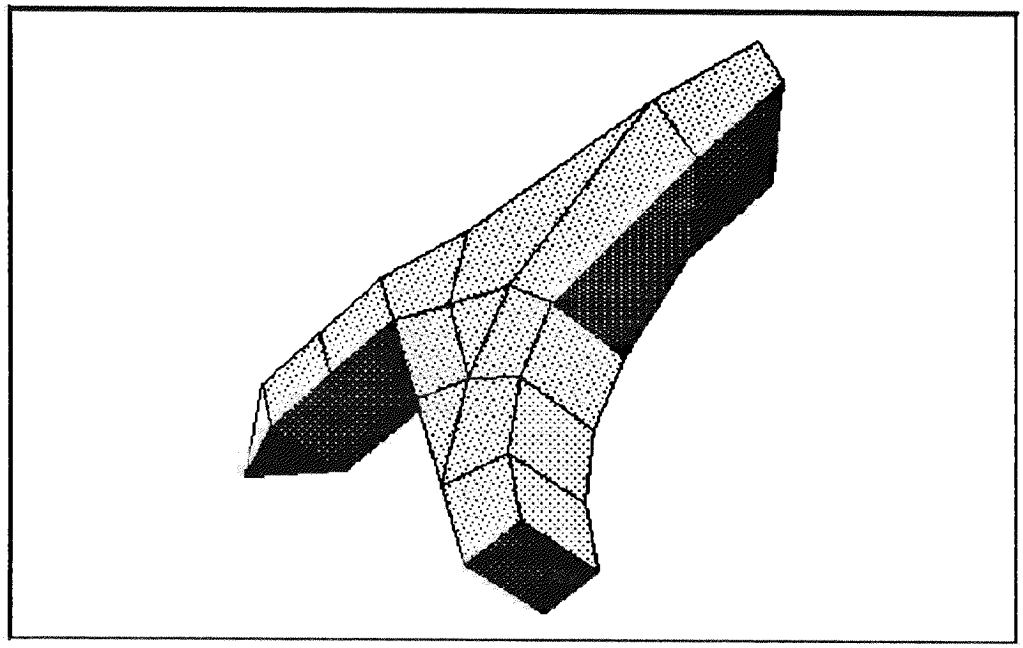


Figure 1 - The Graphics User Interface.

PROBLEM DEFINITION



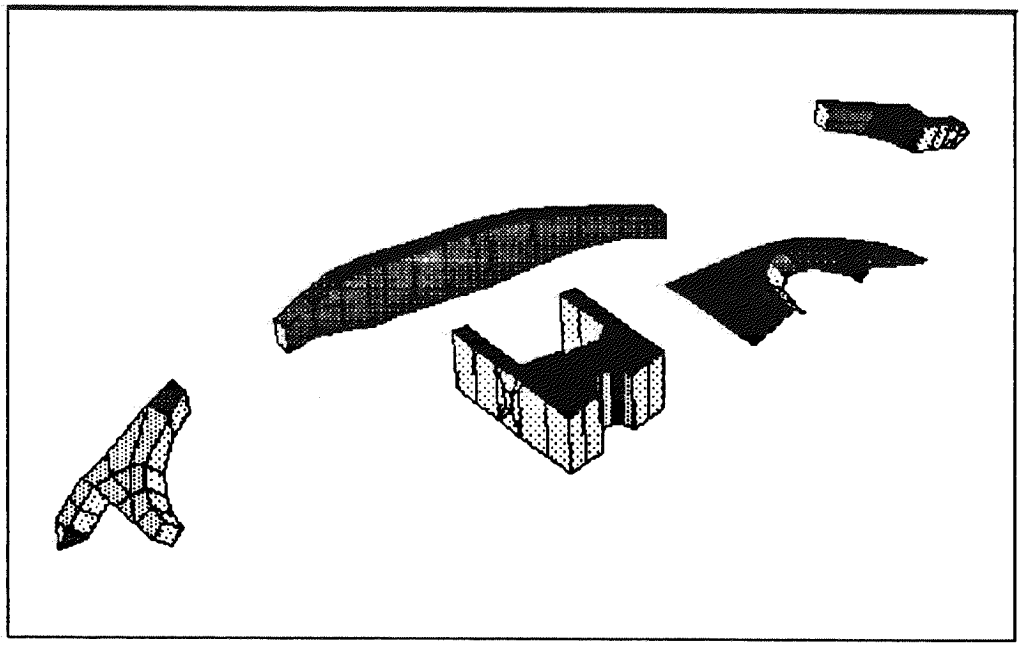


Figure 2 - Creation of Model Segments.

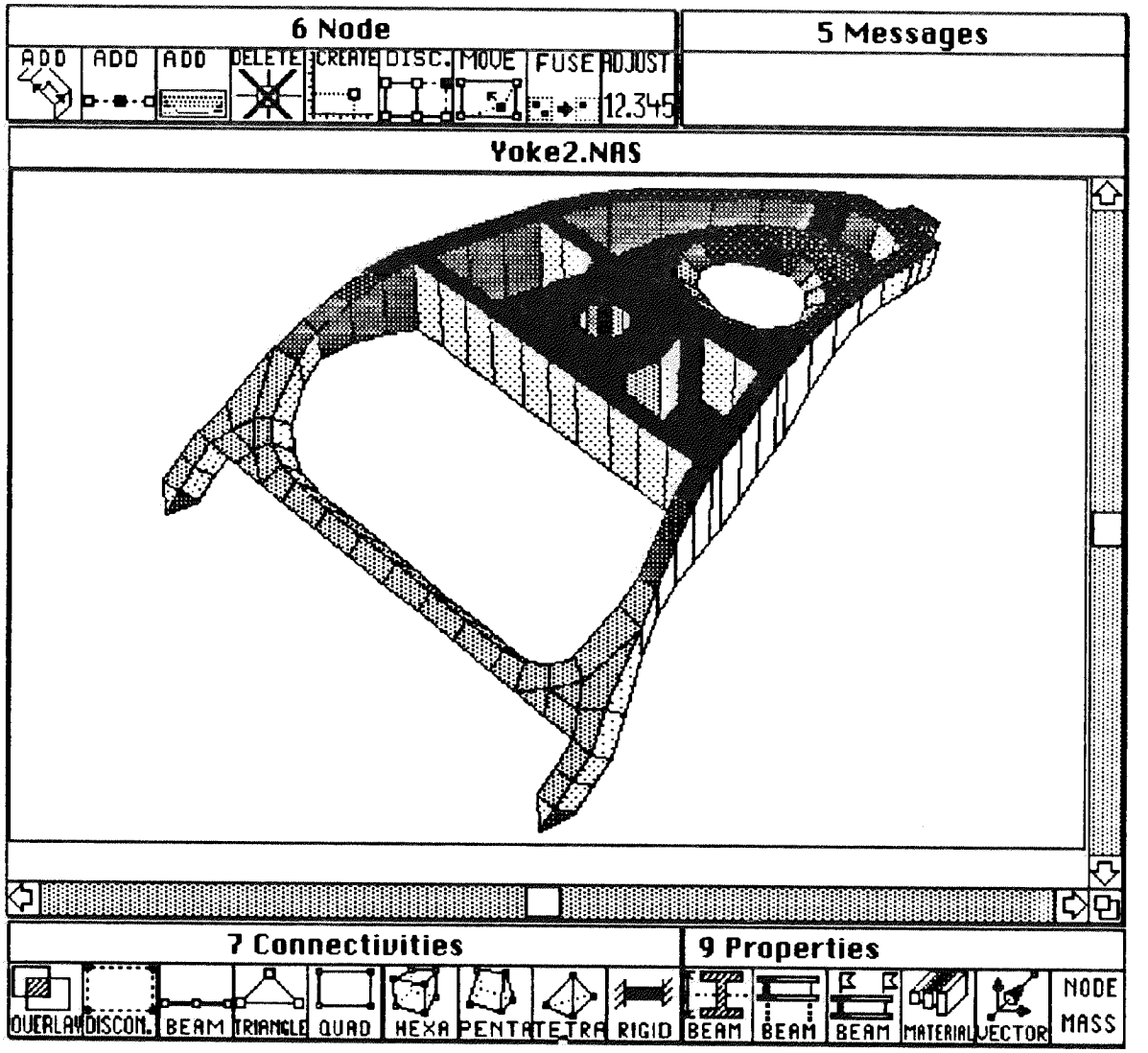


Figure 3 - The Completed Model Geometry.

The smaller model segments were merged together using LapCAD's Snap feature, resulting in the model shown in Figure 3.

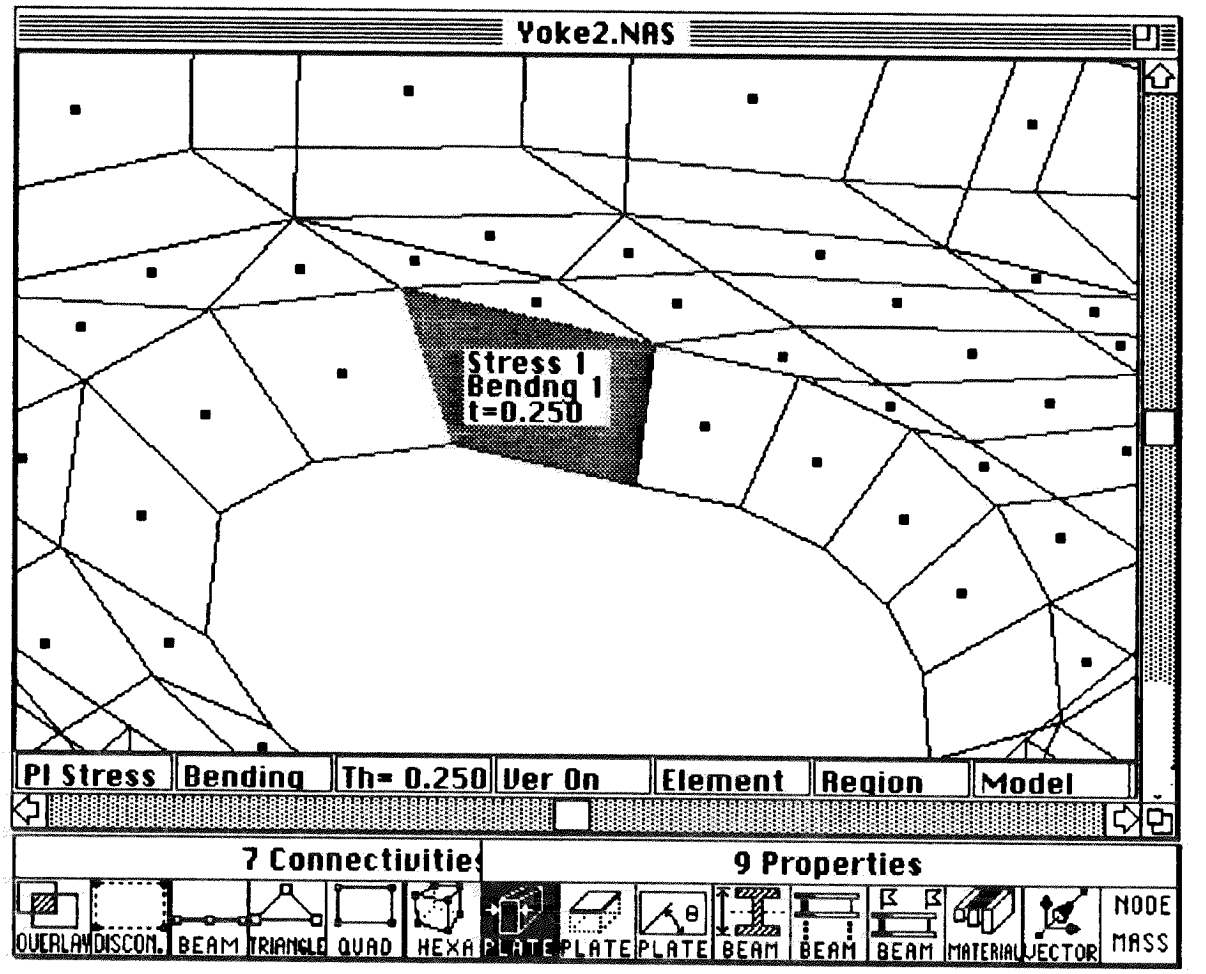


Figure 4 - Application of Physical Properties.

LapCAD automatically adds default values for geometric and material properties. These properties can easily be altered, as shown in Figure 4.

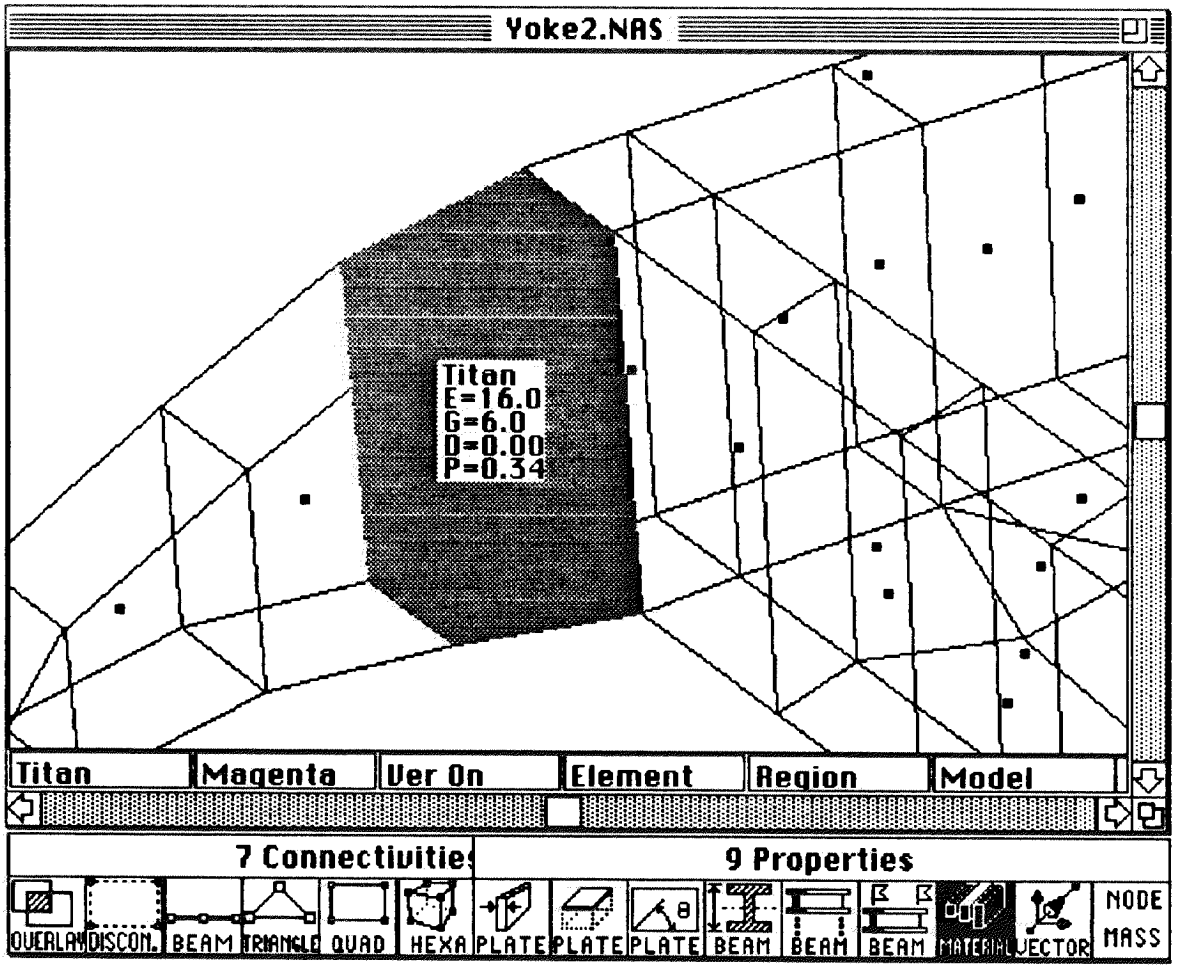


Figure 5 - Application of Material Properties

The user alters the default materials by first selecting the MATERIAL icon, then dials the desired material, and then implements the selection by either clicking on individual elements, groups of elements, or the entire model.

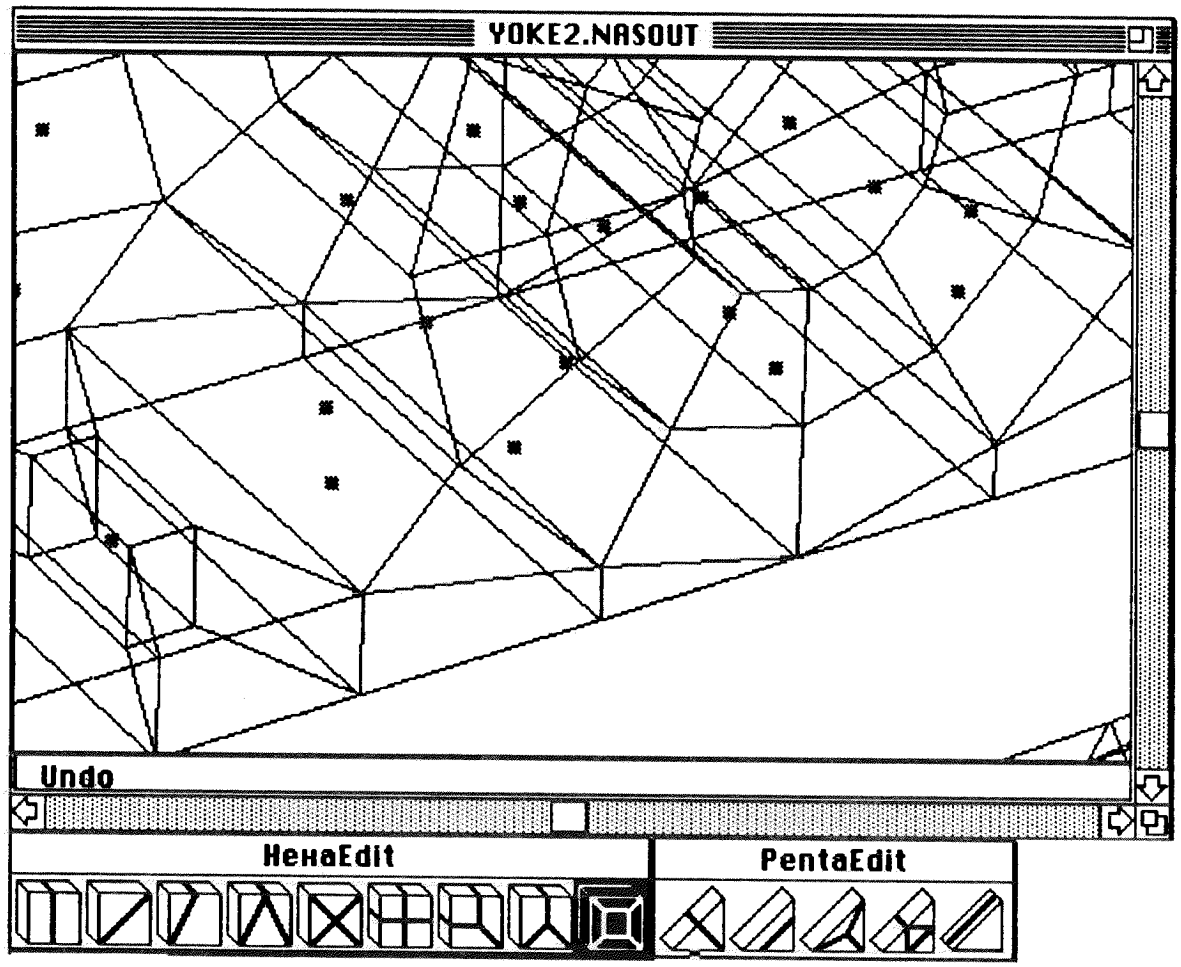


Figure 6 - Editing a Hexa Solid Element.

Surfaces and solids can be edited via the micro-meshing tools, as utilized in Figure 6.

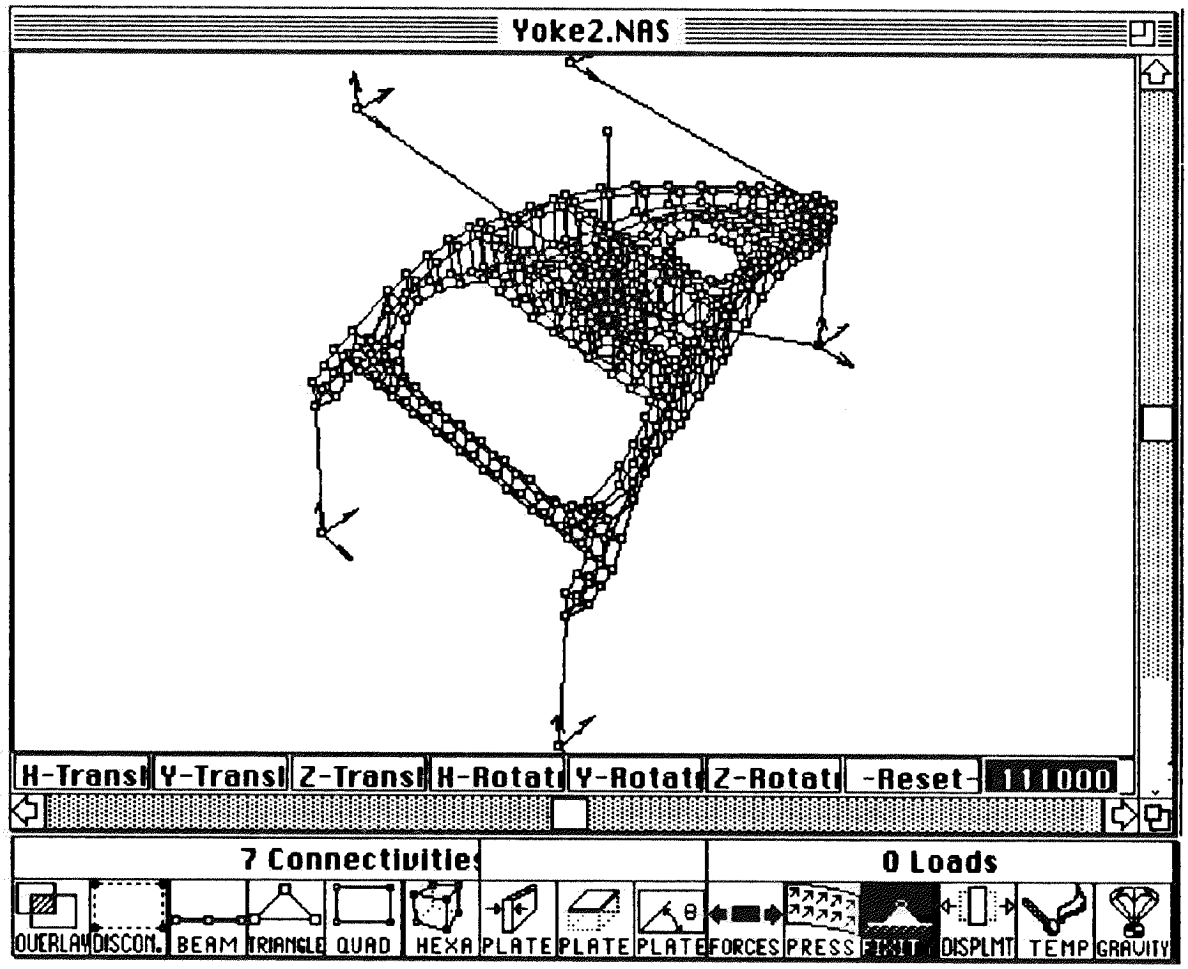


Figure 7 - Boundary Constraints.

The analyst applies six-degree-of-freedom boundary constraints by first selecting the FIXITY icon. He then implements the toggled degrees-of-freedom by clicking on selected nodes, as seen in Figure 7.

```
$ FILE NAME = Yoke2.NAS
$
$ NASTRAN FILE PRODUCED WITH ---- LapCAD ----
$
ID LAPCAD,JOB
DIAG 8,14,20
TIME 10
SOL 24
CEND
TITLE = Created with LapCAD Modeling Software....
SUBTITLE =
ECHO        = BOTH
DISP        = ALL
FORCE       = ALL
STRESS      = ALL
SPCFORCE    = ALL
OLOAD       = ALL
SPC         = 1
SUBCASE 1
LABEL = Single Load Case Created with LapCAD
LOAD  = 1
BEGIN BULK
PARAM,K6ROT,1.0
GRID             1          3.79280-3.61917-0.49684
GRID             2          0.49280 5.65631-0.42262
GRID             3          3.49280-3.61917-0.49684
GRID             4          0.74280 5.65631-0.42262
GRID             5          0.24280 5.65631-0.42262
GRID             6          0.74280 4.75000 0.00E+0
GRID             7          0.44350 5.07944-0.15362
GRID             8          0.00E+0 5.20315-0.21131
GRID             9          1.99280-3.38935-0.30400
GRID            10          2.99E-4 1.00000 0.65000
GRID            11          0.00E+0-1.00000 0.65000
GRID            12          0.00E+0 0.00E+0 0.65000
GRID            13          0.00E+0-0.33333 0.65000
GRID            14          0.00E+0-0.66667 0.65000
GRID            15          0.49980-0.86615 0.65000
GRID            16          0.33320-0.57743 0.65000
GRID            17          0.16660-0.28872 0.65000
GRID            18          1.03360 0.74970 0.65000
GRID            19          1.58360 0.74960 0.65000
GRID            20          1.09980-0.75000 0.65000
GRID            21          1.09980-1.00000 0.65000
GRID            22          1.59980-0.75000 0.65000
```

Figure 8 - The MSC/NASTRAN Executive and Case Control Sections, as created by LapCAD.

ANALYSIS

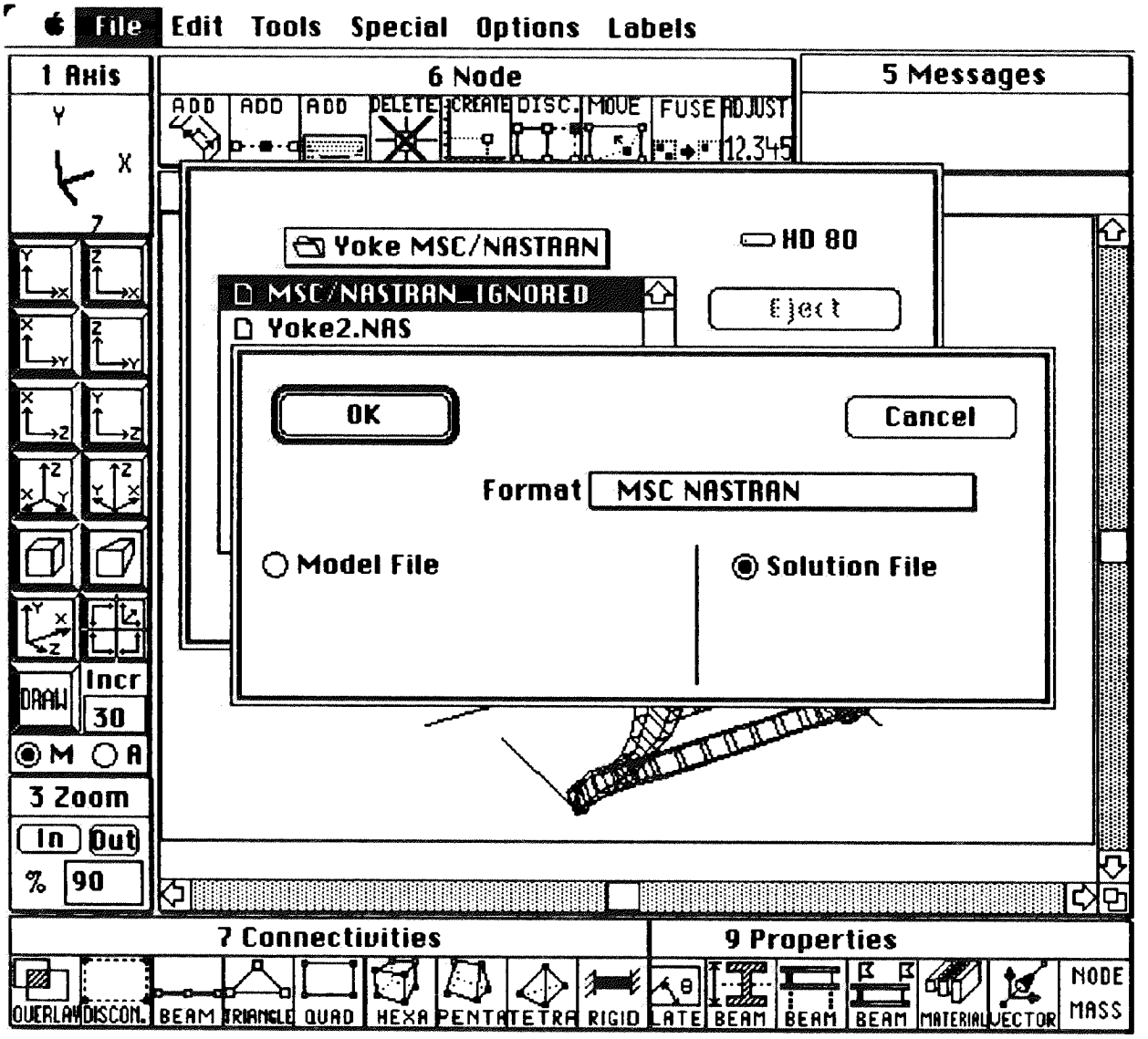


Figure 9 - Opening of the MSC/NASTRAN F06 Output File.

After The MSC/NASTRAN F06 file has been generated, the model is first read back into LapCAD, in order to have a skeleton on which to "hang" the graphical representation of the comprehensive output. The user does this by selecting the Solution File radio button in the dialog box shown in Figure 9.

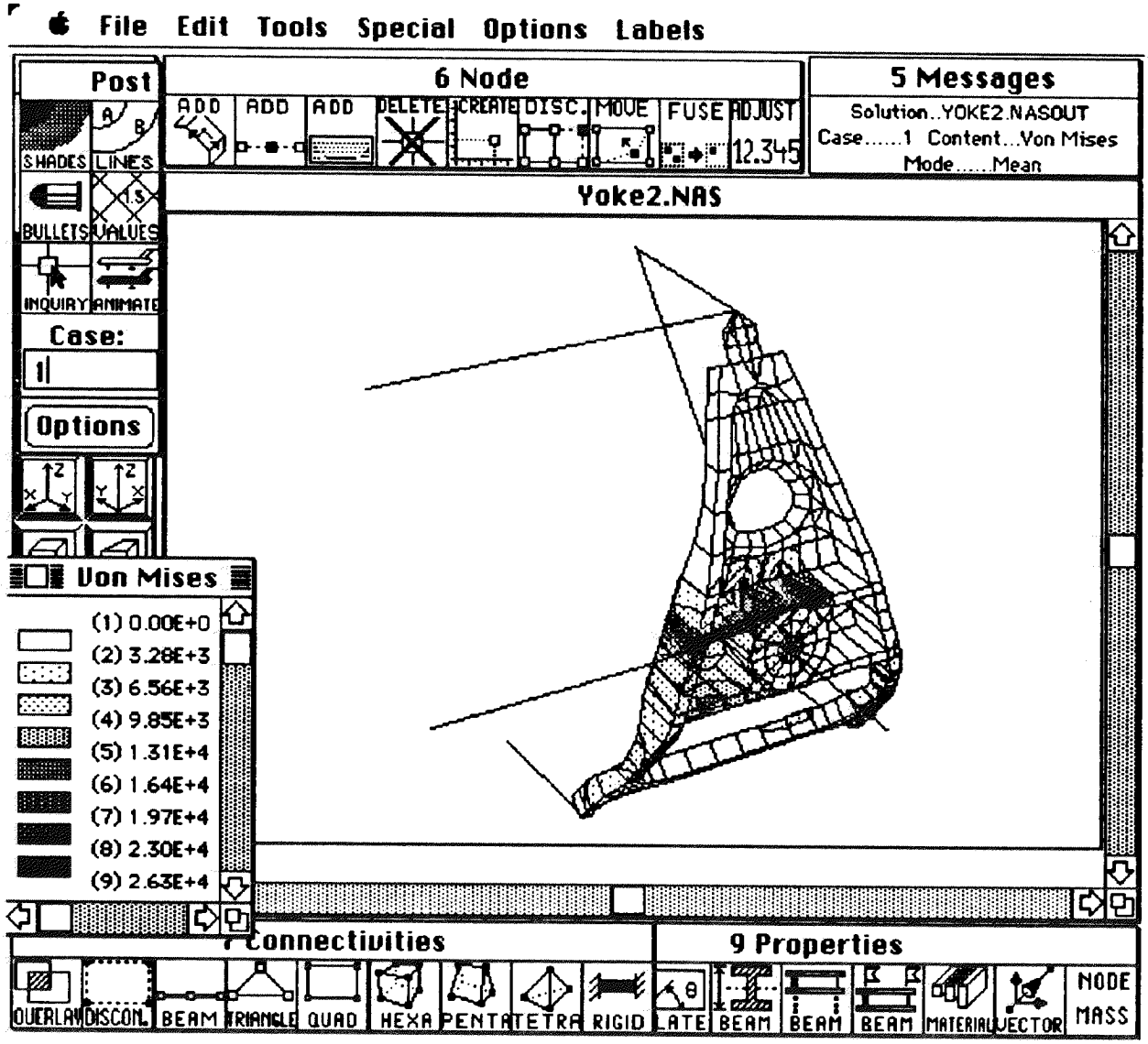


Figure 10 - Shaded Contour of The MSC/NASTRAN Internal Stresses.

A color shaded stress contour is automatically generated by clicking on the SHADES icon in the emerging Postprocessing window, in Figure 10.

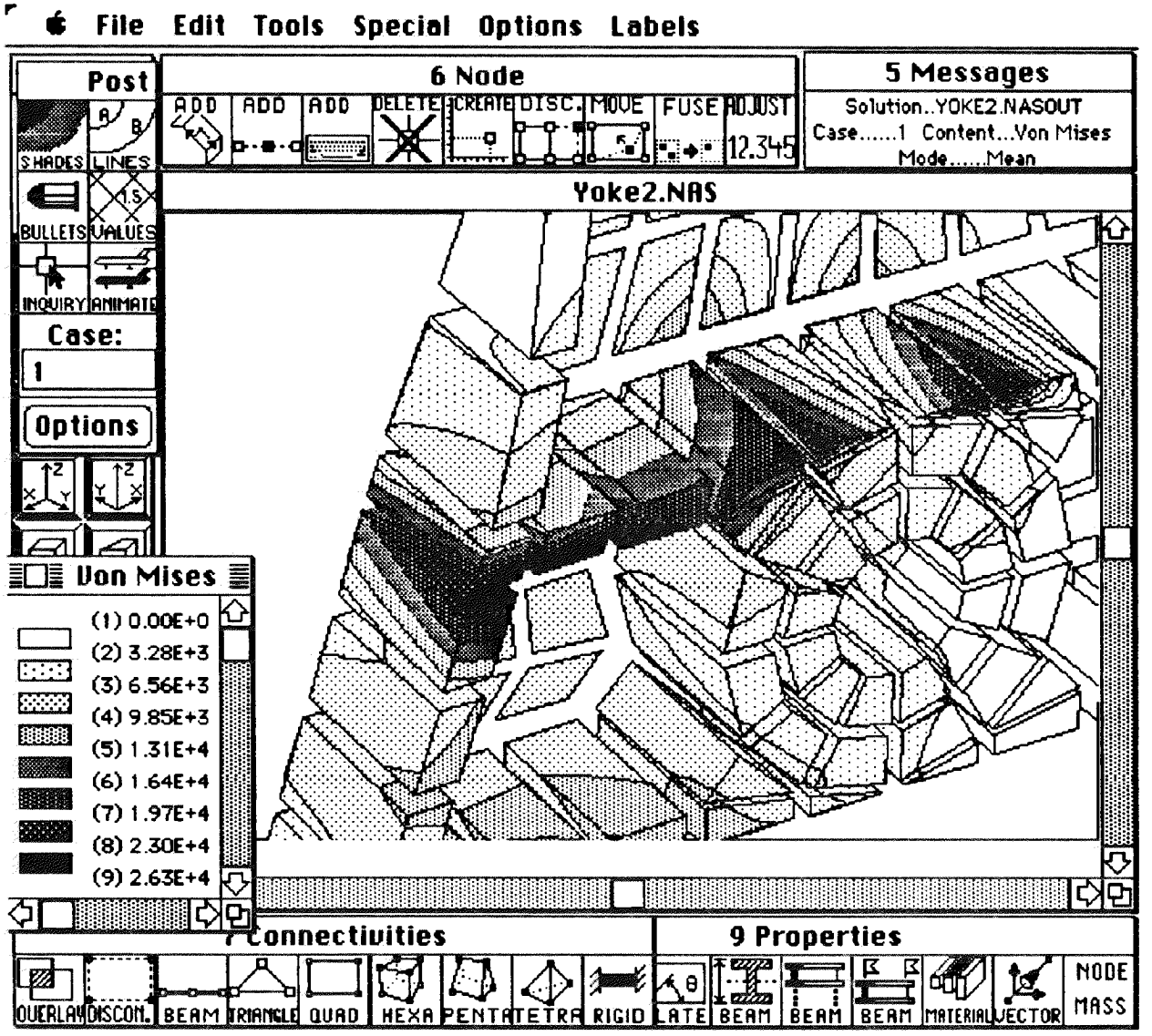


Figure 11 - Shaded Contour of The MSC/NASTRAN Internal Stresses, in Shrink Mode.

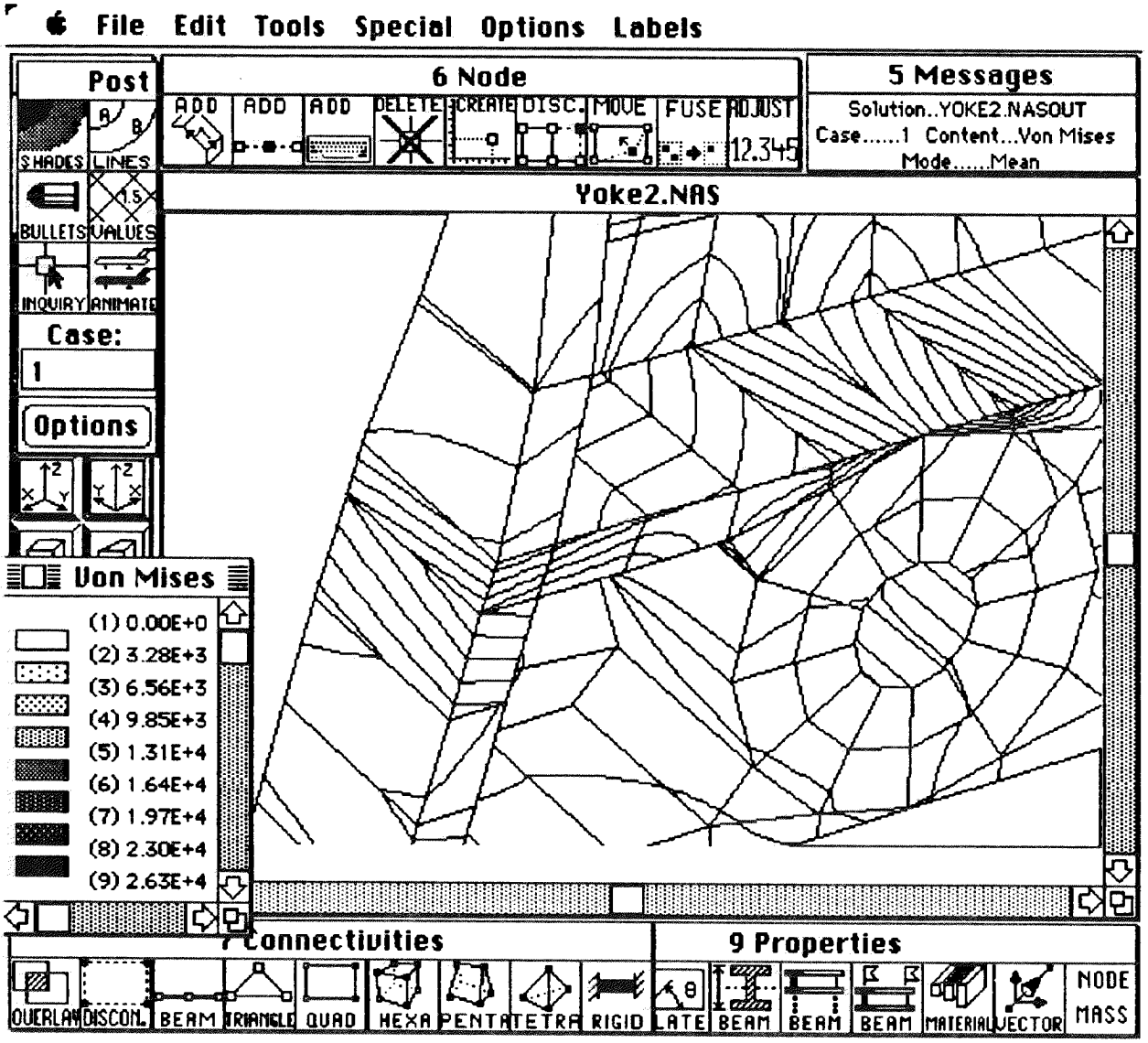


Figure 12 - Line Contour of The MSC/NASTRAN Internal Stresses.

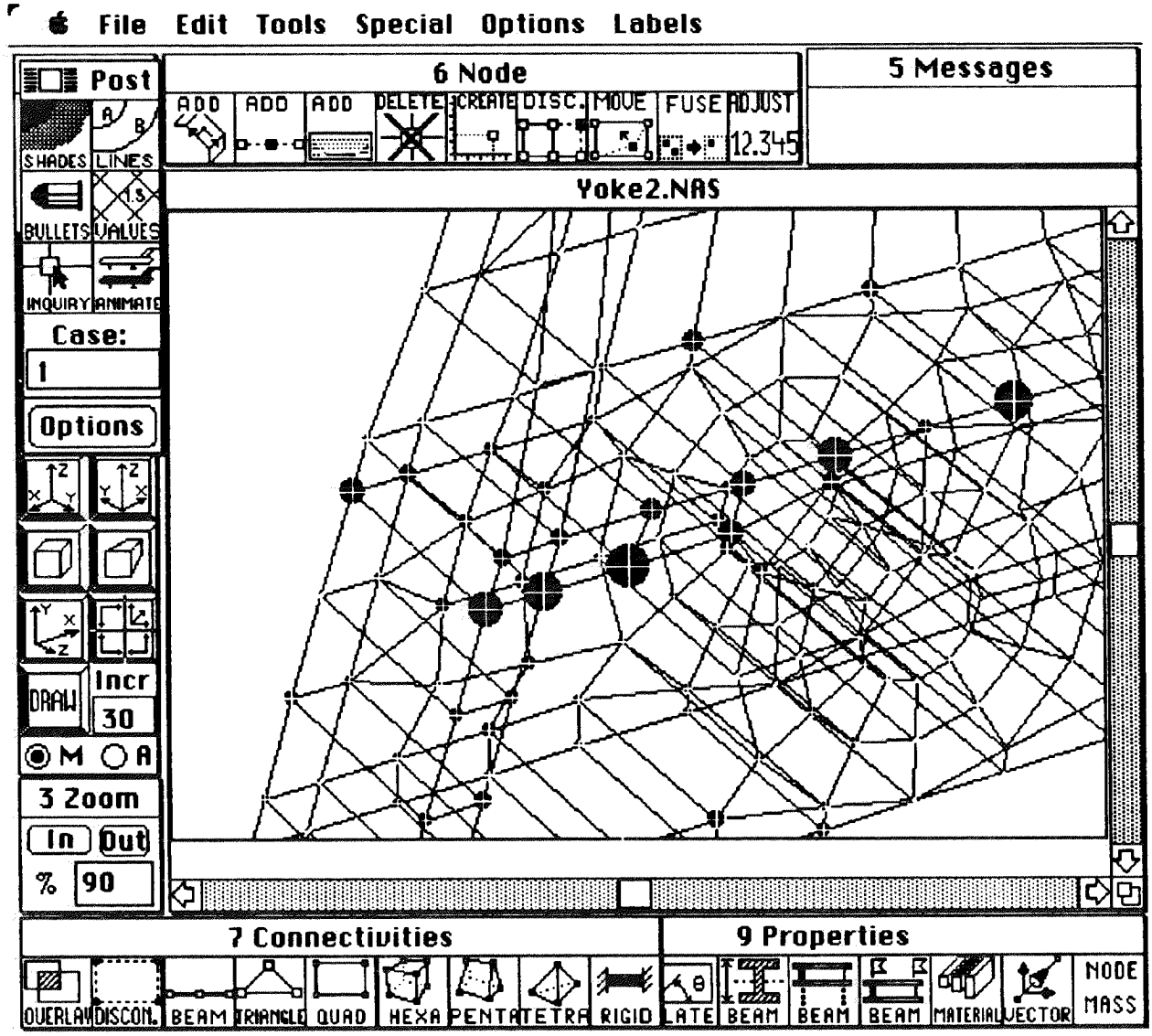


Figure 13 - Bullet Presentation of The MSC/NASTRAN Internal Stresses.

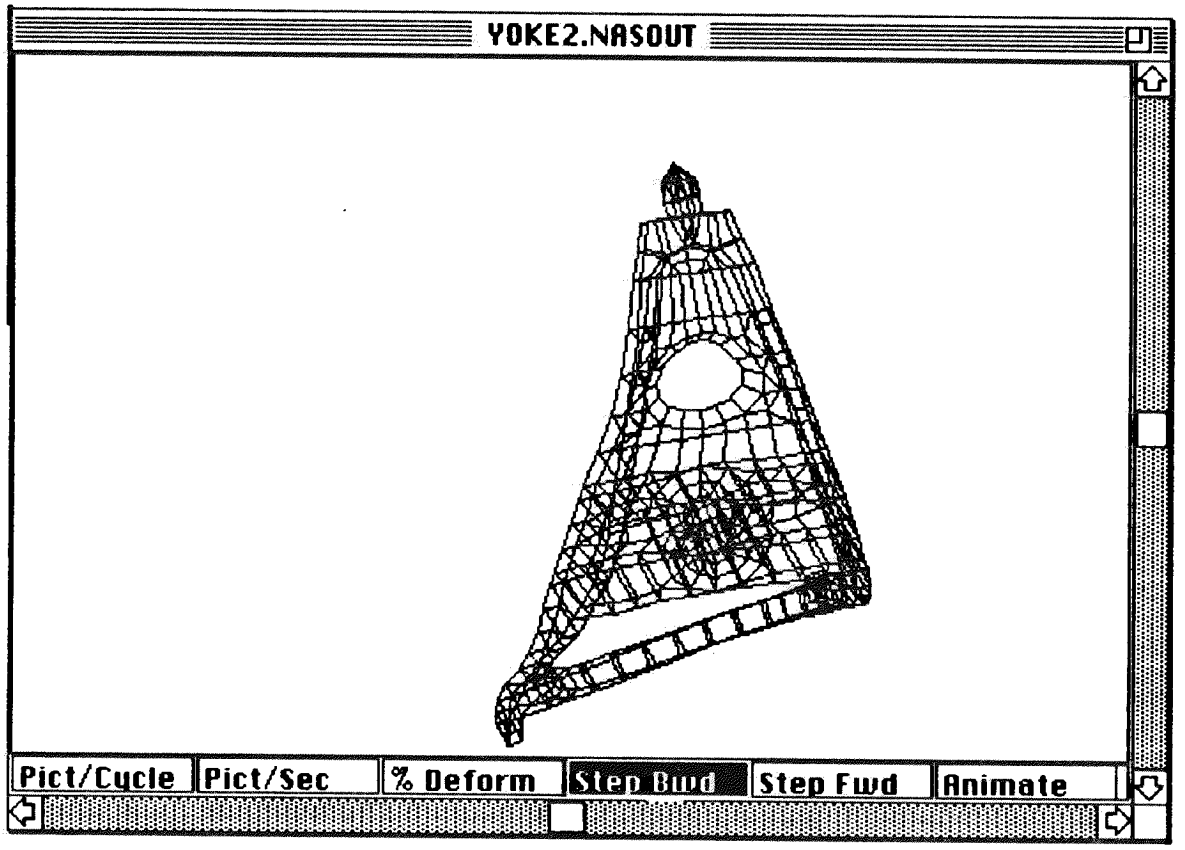


Figure 14 - Animation of The MSC/NASTRAN Displacements.

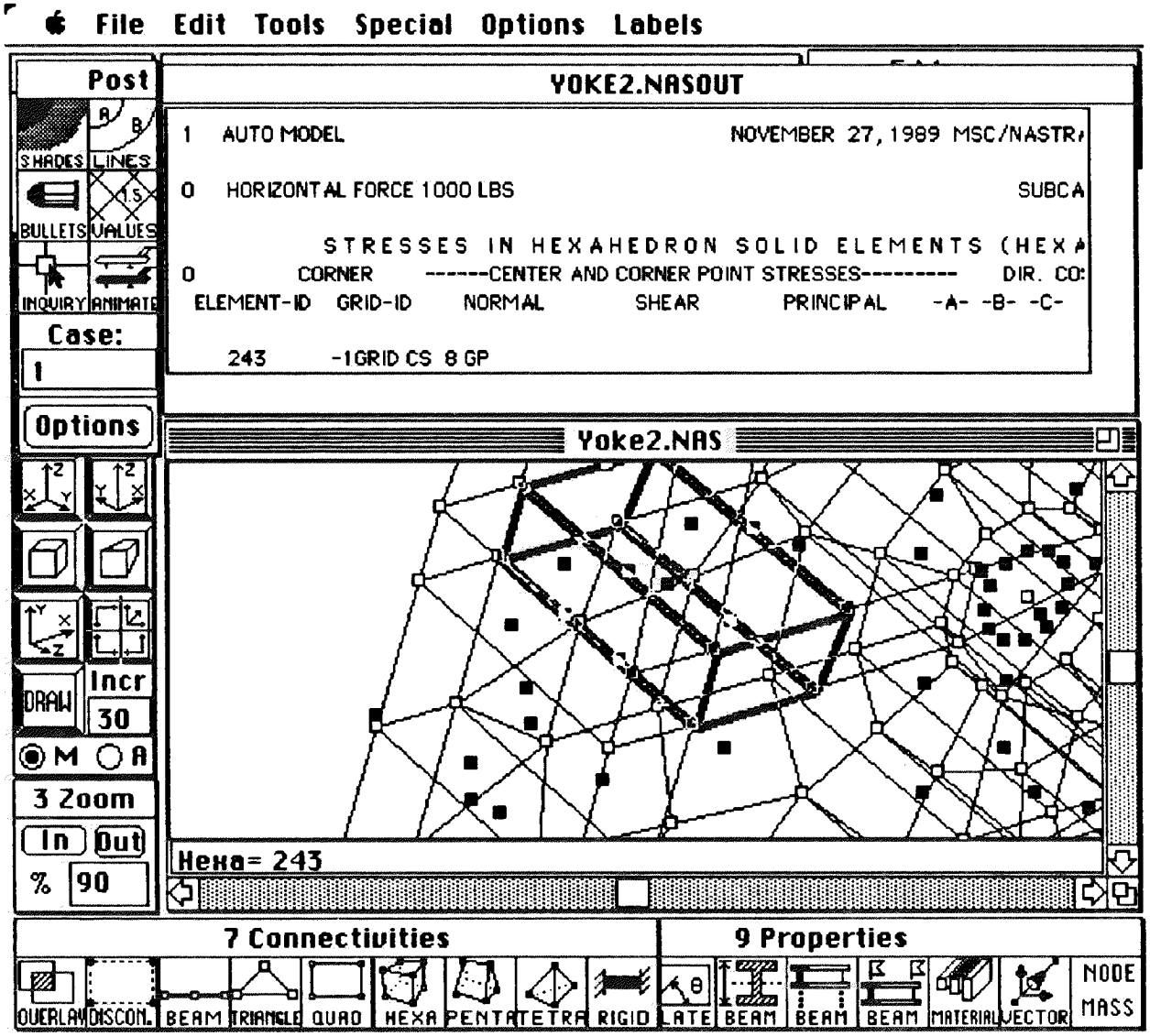


Figure 15 - Query of The MSC/NASTRAN F06 Output File.

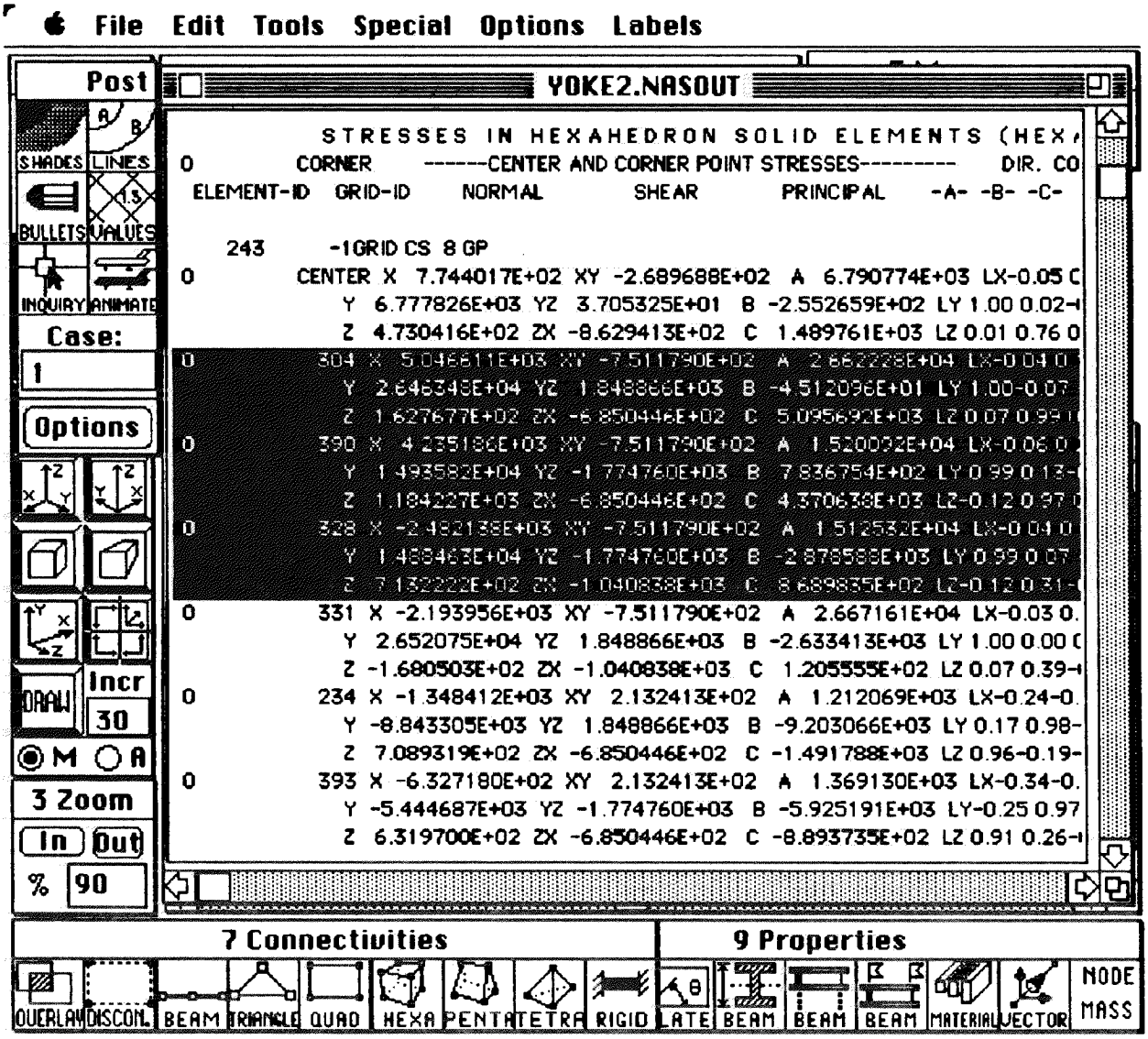


Figure 16 - Copying from The MSC/NASTRAN F06 Output File, and Pasting to a Word Processor.

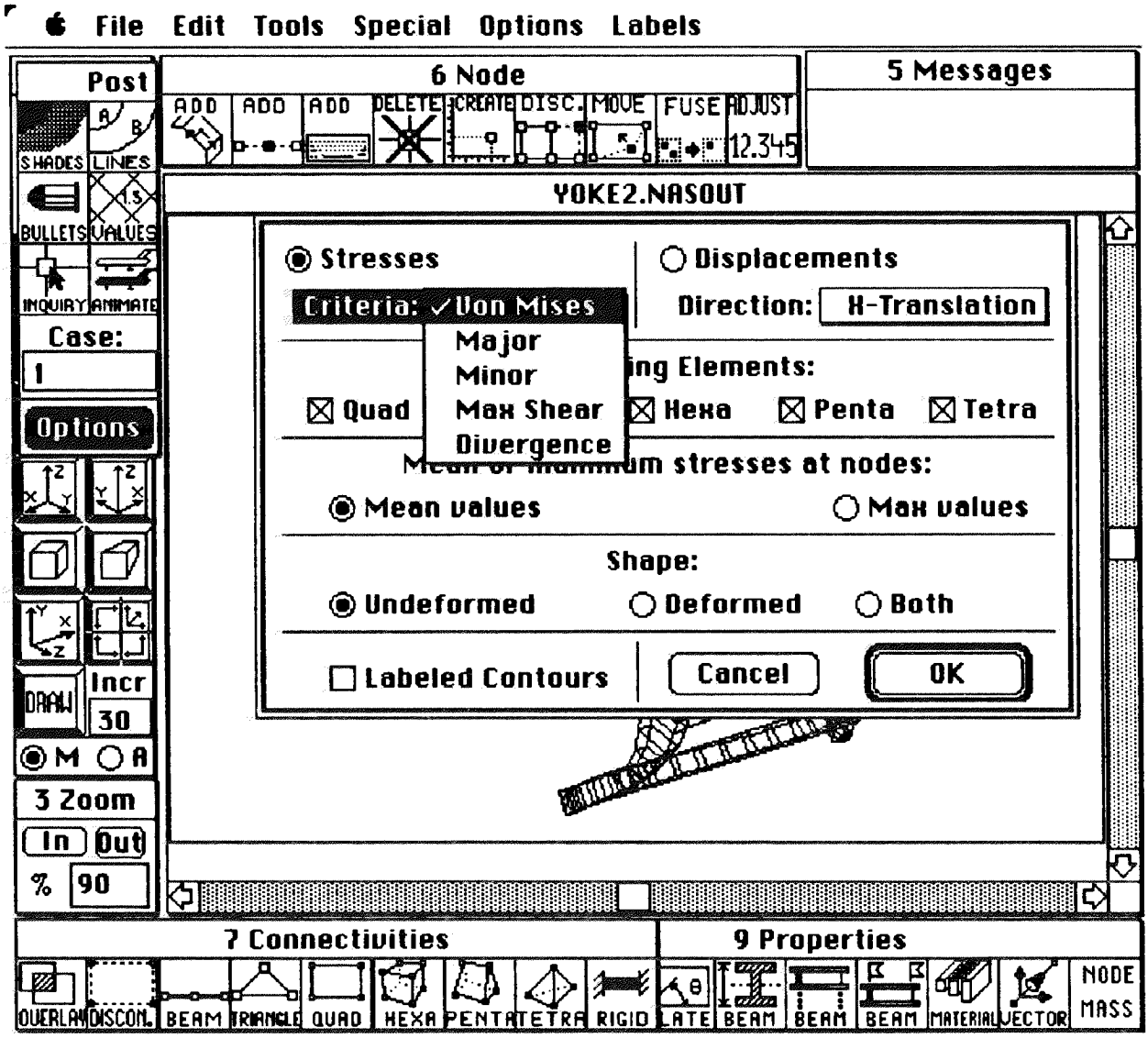


Figure 17 - Options for Postprocessing The MSC/NASTRAN F06 Output File.

DISCUSSION

The example above illustrates how the analyst can take advantage of the easy to use LapCAD modeler, and efficiently create a complex finite element model for The MSC/NASTRAN Solver, including the Executive and Case Control Sections, and the complete Bulkdata.

It also shows how easily The MSC/NASTRAN results can be displayed in LapCAD, by utilizing the F06 Output File.

CONCLUSION

LapCAD is a comprehensive modeling tool, that creates complete finite element models internally, as well as takes advantage of CAD geometry in either the DXF or IGES file formats. It also displays in a clear manner the extensive results that The MSC/NASTRAN Solver produces.

LapCAD is now available on the Macintosh. An IBM version will be available in the near future.