# FINITE ELEMENT ANALYSIS ON THE MACINTOSH

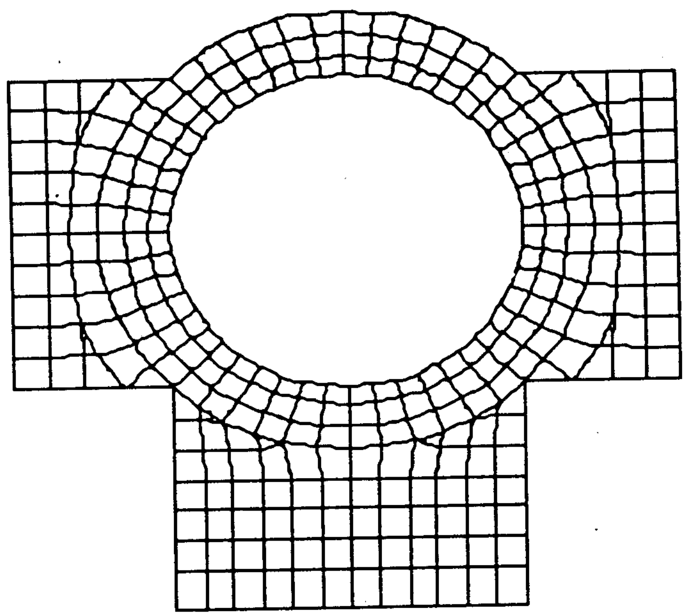
The MacNeal Schwendler Corporation has recently introduced a new version of the MSC/pal finite element system for the Apple Macintosh. This new program is an streamlined version of the IBM PC product which has been available for several years. The program functions best on a Macintosh with hard disk but is easily configured to run on a two diskette system. Two versions of the program are available to support both the 512K and 1 MEG Macintosh machines. The program is executable under either the HFS or MFS Apple file systems. This product is the first full featured FEA program for this machine, and its performance is reviewed in this paper.

The Macintosh version of MSC/pal provides full three dimensional analysis of systems composed of beams, plates, and shells. Computations are performed in double precision, and solution types include static deformation, frequency response, normal modes analysis, and transient response. The program features automatic nodal point resequencing to minimize equation bandwidth. The resulting equations are stored in skyline column format.

Solution output (both text and graphical) may be directed to the screen, printer, or disk. Three dimensional or XY plots of computed results are generated automatically. The structure graphics options include animation of static deflections and normal mode shapes. Stress profile plots provide immediate recovery of the maximum Von Mises, shear, major, and minor principal stresses for each element in the problem. Plots of velocities, accelerations, displacements and forces are produced by the transient and frequency response routines. The figures provided in this paper were generated by the MSC/pal system and transported to the MacWrite word processor using the Apple Clipboard. This process is a standard feature of the Macintosh which provides a document transfer path between screen images and the word processor. There are three analysis problems with solution results in this paper. These problems were chosen primarily to demonstrate the execution speed of the Macintosh. This is a major area of concern with this product since it has no numeric coprocessor. The solution times provided for each example were obtained using a Macintosh Plus equipped with a Data Frame SCSI 20 megabyte hard drive.

## Trunion Bearing Analysis

The figure below illustrates a FEA model of a trunion bearing design for a earth mover. The bearing was modeled using the plane stress quad and triangular plate elements provided in the MSC/pal library. The model contains 288 elements, 346 nodal points, and required 660 equations in the stiffness matrix. This model filled 21% of the available matrix space.

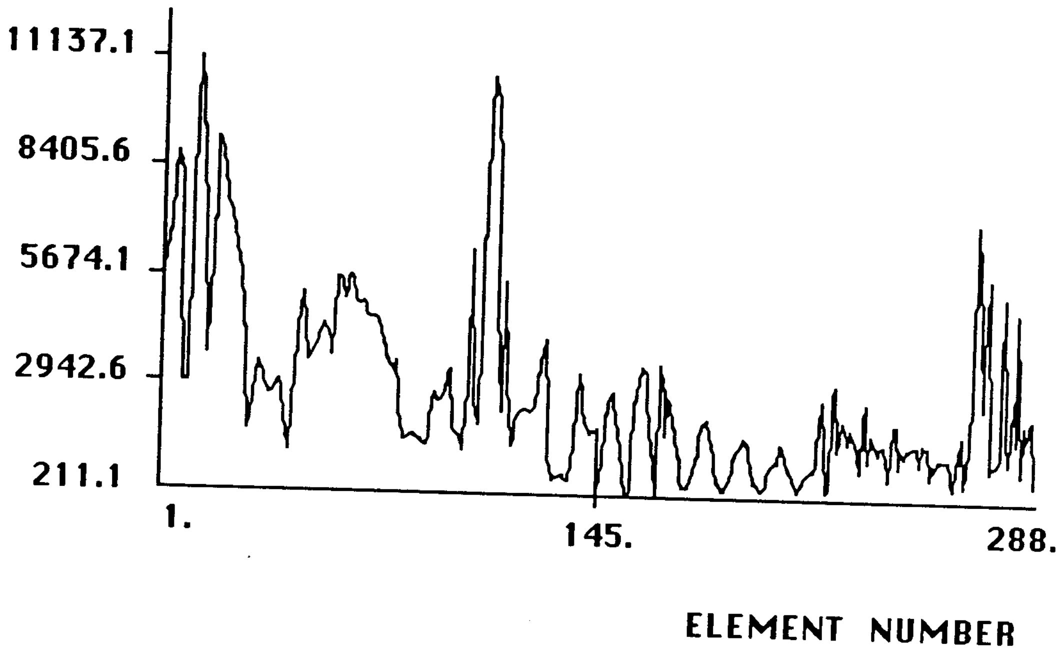


## Trunion Bearing Model

All MSC/pal systems operate in restart mode as a standard feature. That is, once the equations are generated, a solution of any type may be obtained using the current equation set. Multiple solutions may be obtained without regeneration of the model equations. Thus, I am providing computation times for both equation generation and solution.

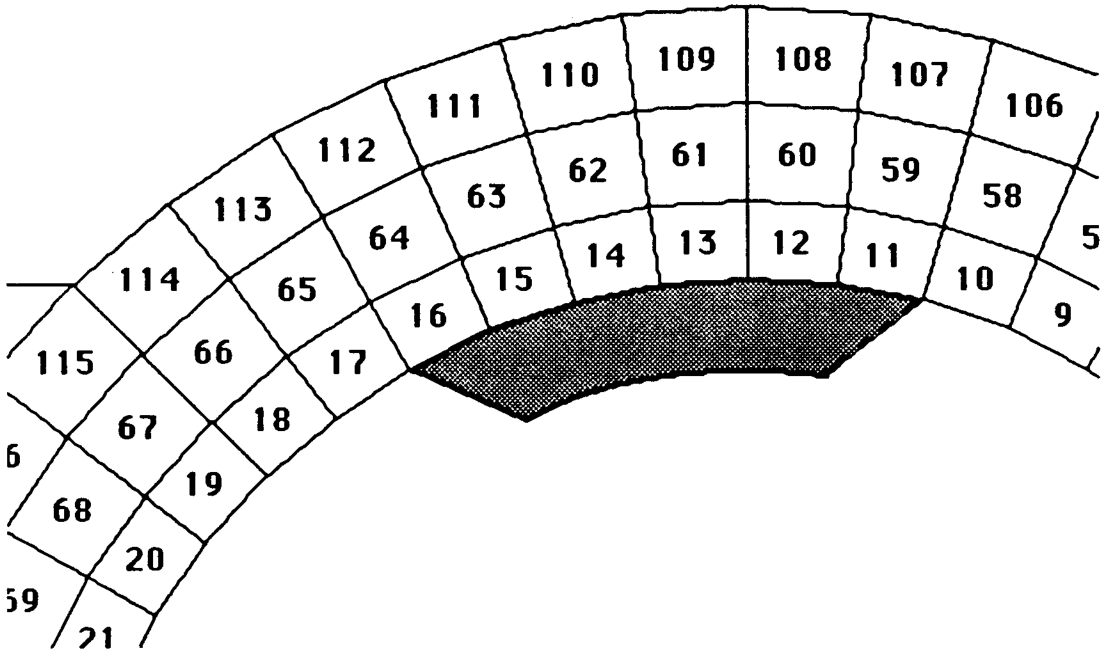
Equation generation for this problem required 8 minutes, 50 seconds and static stress and deflection results were obtained in 6 minutes, 50 seconds.

This version of MSC/pal provides plots of maximum element stresses versus element number. These plots are available for beam, plate, and shell element results.




**Trunion Bearing Stress Plot: Maximum Von Mises Stress**

As shown by the vertical scale, the maxium stress level for the study is 11137.1 psi. The maximum stresses occur in the two areas between element numbers 10 to 20 and elements 95 to 115. This area of the bearing is shown enlarged in the plot below. The element numbers are included in this plot. The nodal point numbers were excluded for readability. The load for this solution was distributed along the inner race and is shown in grey below.




**Trunnion Bearing Model: Enlarged with element numbers**

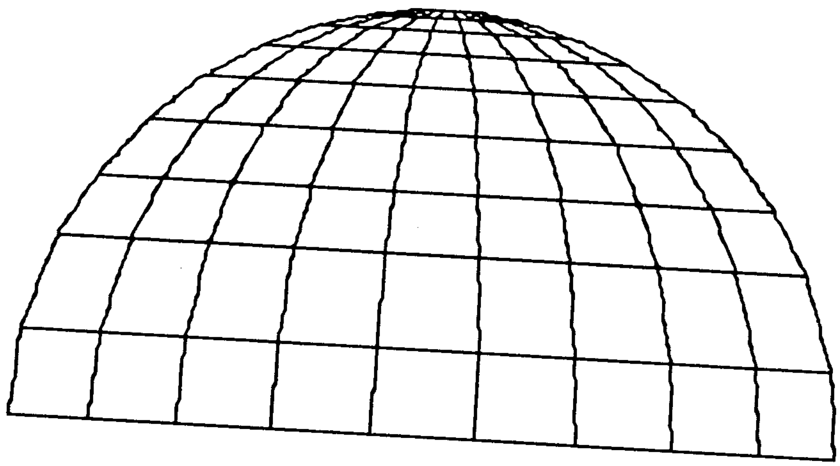
MAXIMUM STRESSES FOR QUADRILATERAL ELEMENT

| ELMT | NODE | MAJOR | MINOR | SHEAR | VON MISES | CONNECTED NODES | | | |
|---|---|---|---|---|---|---|---|---|---|
| 12 | 12 | -3.0144E+03 | -1.1106E+04 | 4.0459E+03 | 9.9475E+03 | 12 | 13 | 62 | 61 |
| 12 | 13 | -2.4769E+03 | -1.1103E+04 | 4.3132E+03 | 1.0095E+04 | 12 | 13 | 62 | 61 |
| 13 | 13 | -2.6188E+03 | -1.2213E+04 | 4.7972E+03 | 1.1137E+04 | 13 | 14 | 63 | 62 |
| 13 | 14 | -2.7462E+03 | -1.2213E+04 | 4.7335E+03 | 1.1098E+04 | 13 | 14 | 63 | 62 |
| 14 | 14 | -2.5473E+03 | -1.1100E+04 | 4.2762E+03 | 1.0071E+04 | 14 | 15 | 64 | 63 |
| 14 | 15 | -2.6492E+03 | -1.1100E+04 | 4.2254E+03 | 1.0041E+04 | 14 | 15 | 64 | 63 |
| 108 | 160 | 9.2390E+03 | -9.5984E+02 | 5.0994E+03 | 9.7544E+03 | 110 | 111 | 160 | 159 |
| 109 | 161 | 1.0312E+04 | -7.2569E+02 | 5.5190E+03 | 1.0694E+04 | 111 | 112 | 161 | 160 |
| 109 | 160 | 1.0312E+04 | -6.2409E+02 | 5.4682E+03 | 1.0638E+04 | 111 | 112 | 161 | 160 |
| 110 | 162 | 9.6663E+03 | -4.0868E+02 | 5.0375E+03 | 9.8770E+03 | 112 | 113 | 162 | 161 |
| 110 | 161 | 9.6650E+03 | -9.1233E+02 | 5.2887E+03 | 1.0152E+04 | 112 | 113 | 162 | 161 |

A detailed printout of the element stress levels can be directed to disk storage for later analysis. This was done for this problem and the printed results are provided above. A "clipping" level of ten thousand psi was set to delete the majority of the printed data. As can be seen, elements 12, 13, 14, 108, 109, and 110 are experiencing the highest stress levels. The listing above also includes the nodal point location for the stress values.

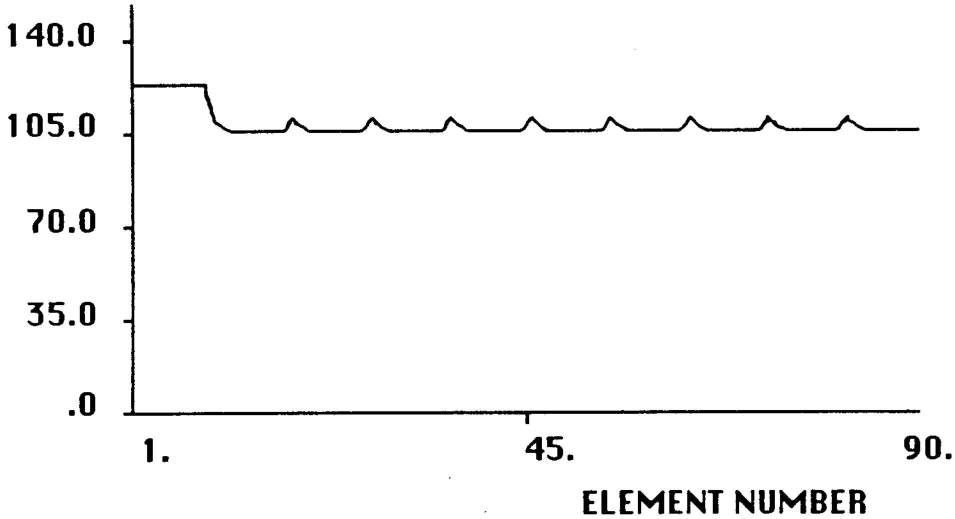
The output listing includes values for the major principal, minor principal, maximum shear, and maximum Von Mises stresses. A more detailed stress report may be requested which provides the membrane and bending components in element coordinates.

## Spherical Shell Analysis



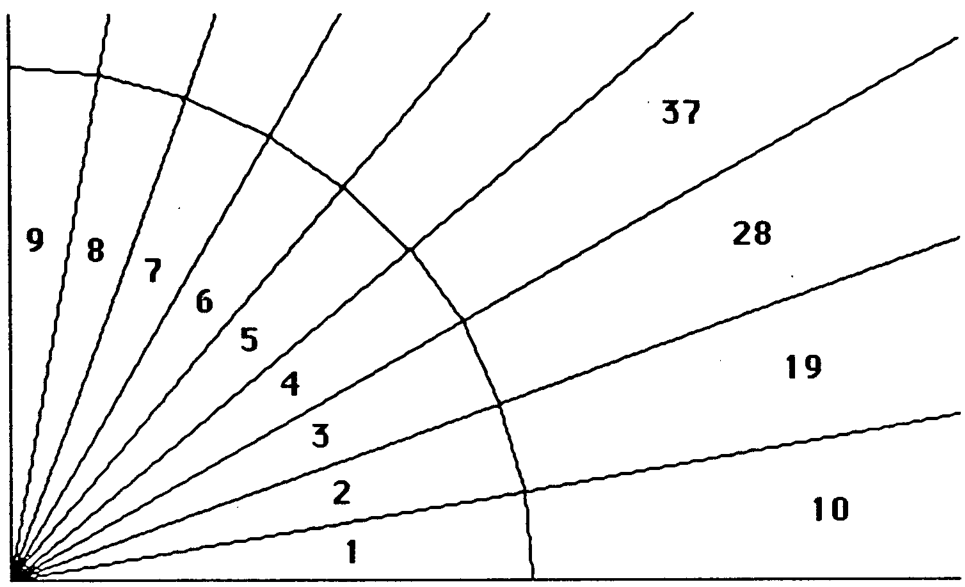
## Spherical Shell Model: 1/8 Section

The spherical shell model was created using quad and triangular shell elements. There are 90 elements and 110 nodal points in the model. The model uses spherical coordinates sweeping 90 degree arcs in two directions. The resulting model is 1/8 of the total surface of the complete sphere being studied. Boundary conditions were applied at the edges to enforce symmetry and a uniform internal pressure was applied to all elements. The model definition required 541 equations and used 35 percent of the matrix coefficient space available. The equation generation time for this model was 11 minutes, 45 seconds, with 16 minutes, 13 seconds being required to compute the deflection and stress levels. This total solution time was 28 minutes. The stress profile results are shown below.




**Spherical Shell: Von Mises Stress Levels, Pressure Loading**

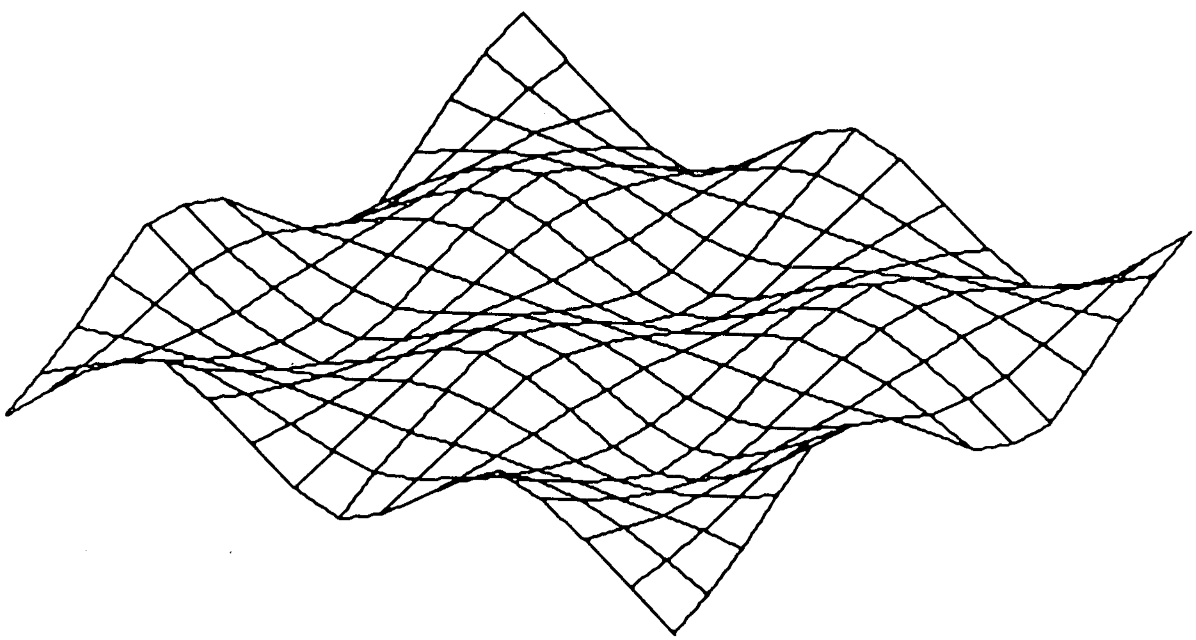
The exact solution was 106 psi which agrees guite well except for the first 9 element results. Most of this inaccuracy was caused by the extreme aspect ratios for the triangular elements used in this model as shown in the figure below.




**Spherical Shell Model: Enlarged View of Apex Elements**

## Free Plate Vibration

The plate vibration mode depicted below was obtained for a 255 node model of a flat rectangular plate. The model has 224 elements and 765 degrees of freedom. The equation generation time was one hour and 42 minutes, most of which was consumed doing the Guyan reduction to 72 active equations. Extraction of the 25 lowest normal modes of vibration for the model required 42 minutes and 30 seconds. The mathematical model occupied approximately 41% of the available matrix space for the program.



**Free Plate Vibration: 25th Mode, 2413 Hertz**

A rerun of the problem above requesting only 20 active equations required 31 minutes, 50 seconds for equation generation with the solution for 20 normal modes requiring 9 minutes, 15 seconds. The second run showed a expected loss of accuracy in frequency prediction relative to the first. The relative percent error was less than 3% for the first eight modes climbing to 10.6% at the 15th mode.

## Summary

Execution times are dependent upon the number of degrees of freedom (dof), the bandwidth or profile size, and the type of solution being performed. The problems of this paper were purposely restricted to plate and shell analysis, excluding the simpler beam, spring, mass, damper type analyses. The bandwidths for all these problems were relatively large due to their square shapes (aspect ratios near one). As such, the results approximate a worst case timing demonstration for the dof involved. The largest problem presented required 41% of the available memory. Thus, the program is capable of solutions considerably larger than those presented here. The execution times are summarized below.

| Problem Type | Generation | Solution | Total |
|---|---|---|---|
| 2D Plane Stress<br>346 nodes, 660 dof<br>Matrix 20% Filled | 9 | 7 | 16 |
| 3D Shell Analysis<br>110 nodes, 541 dof<br>Matrix 35% Filled | 12 | 16 | 28 |
| Plate Bending, 8 modes<br>255 nodes, 765 dof<br>Matrix 41% Filled | 32 | 9 | 41 |
| Plate Bending, 25 modes<br>255 nodes, 765 dof<br>Matrix 41% Filled | 102 | 42 | 144 |

**Execution Times in Minutes**