FINITE ELEMENT ANALYSIS ON THE MACINTOSH PERSONAL COMPUTER

Ken Ranger
The MacNeal-Schwendler Corporation

ABSTRACT

The MacNeal-Schwendler Corporation (MSC) has upgraded its Macintosh version of the MSC/pal 2 finite element program to include solid elements, thermal stress analysis, improved performance, smaller databases, and a new user interface with more "MAC-like" menus and dialog boxes.

Two versions are now available: a 500-node version for the Macintosh Plus or SE and a 2000-node version for the Macintosh II.

Two examples of analysis are presented to show the improved finite element tools that are available to the Macintosh personal computer owner.

INTRODUCTION

MSC has released the MSC/pal 2 version 3.5 FEA program for use on the Macintosh computer. Macintosh users can now use MSC/pal 2 to perform three-dimensional stress and vibration analysis using an excellent Macintosh graphical user interface, which includes sizable windows, pull-down menus, model editor cutting and pasting, and dialogue boxes. This release is an upgrade from the 1.98 release.

There are two versions available. The 500-node version runs on the Macintosh Plus-SE, and requires at least 1MB of memory. The 2000 node version operates on the Macintosh II-SE/30, and requires at least 2MB of memory and a math co-processor. Both require a hard disk with at least 20 MB.

The 3.5 release includes the following element types:

- straight and curved beams
- quadrilateral and triangular plates
- solid elements (hexa, penta, and tetra)
- axisymmetric quadrilateral and triangular elements
- springs, masses and dampers

The following analyses are supported:

- statics
- thermal stress statics
- normal modes (less than 150 active DOF)
- subspace iteration normal modes (less than 12000 DOF for 2000 node version, 3000 DOF for the 500 node version)
- frequency response, forced and unforced (less than 125 active DOF)
- transient response (less than 150 active DOF).

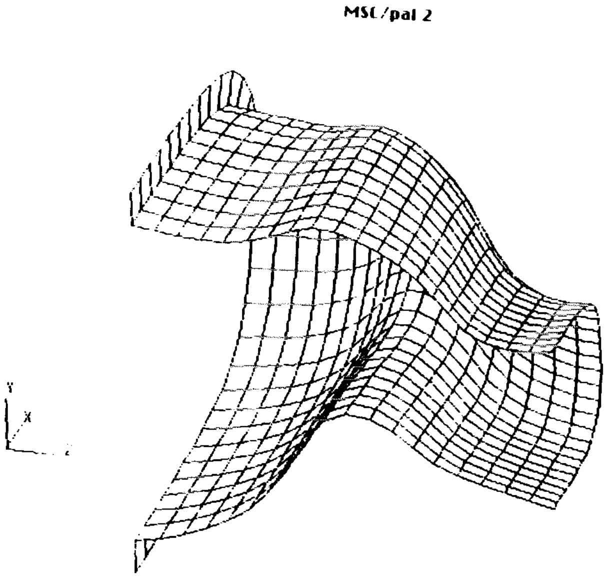
The following types of pre- and postprocessing are available:

- printouts of displacements, element and nodal stresses, applied loads and reaction forces
- undeformed plots showing element and node numbers
- deformed plots of analysis results
- contour plots of nodal stresses, displacements, and temperatures
- XY plots of model element stresses, or nodal displacements, velocities, accelerations and element stresses versus time and frequency.

DISCUSSION

Two examples are provided to illustrate how MSC/pal 2 works. The first shows how MSC/pal 2 interfaces with computer-aided-design (CAD) programs. It is an example of a bent jet exhaust tube (see figure 1 and 2).

This model is a curved, non-circular, non-uniform pressurized tube. Only one half of the tube is modeled because of its symmetrical shape and applied loads. It is loaded with internal pressure and with temperatures. The geometry for the FEA model is made with AUTOCAD, a popular CAD program.



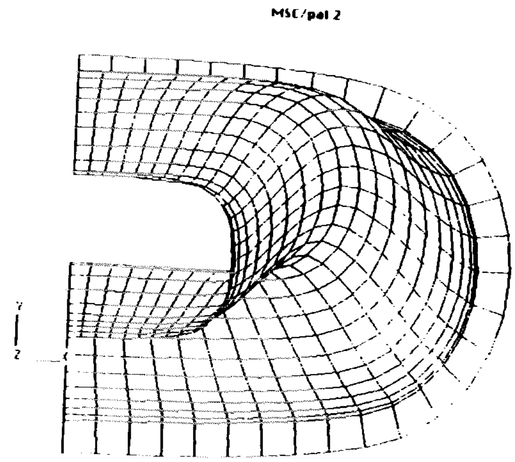


Figure 1 Figure 2

Jet engine exhaust tube

In AUTOCAD, 3D POLYLINES define the edges of the model and the EDGESURF command is used to create the two dimensional shape. This EDGESURF entity is transformed into individual 3DFACE entities using the EXPLODE command. Individual entities are necessary so individual finite elements can be created later on in the translator program. A DXF data file of the model is then written using one of AUTOCAD's output-to-file options.

MSC's CAD2MSC program is then used to translate the DXF datafile to a MSC/pal 2 file. The MERGE NODES nodes option in CAD2MSC must be active because the 3DFACE entities created in AUTOCAD specify coordinates for four points, not node numbers. When the point coordinates are translated to MSC/pal 2 nodes, points with coincident locations need to be given the same node number. The DXF to MSC/pal 2 MODEL translation option is then used to write the MSC/pal 2 model geometry to a file.

The model contains 851 nodes and 630 quadrilateral plate elements. MSC/pal 2's GRAPHICS/VIEW option can then be used to check the geometry of the model. The model still needs material properties specified in its model file, and loads and constraints specified in a loads file. This is done in MSC/pal 2's editor. Typical Macintosh cutting and pasting tools are available for easy model editing. The material is steel. An internal pressure is applied to all the elements along with a temperature distribution to the nodes. The model is cantilevered on its large side and constraints are applied to account for symmetry.

The FEA model is now completed. To run the analysis, first the MSC/pal 2's ANALYSIS option is used to generate the stiffness matrix. Then the STATICS option is used to solve the problem.

The results of the analysis can be presented in printed form from the STATICS/DATA RECOVERY option or in the GRAPHICS section in plotted form. The deformed/undeformed shapes (figure 3) and the stress contour (figure 4) are generated using the GRAPHICS/VIEW option. Large stresses can be seen on the flange and in the upper side indicating that stiffening is needed in this area.

These views can be quickly rotated to any angle, enlarged, or moved using the viewing options in MSC/pal 2's dropdown menus and popup dialog boxes. Color and line contours are also available. Plots provide a quick method of checking the analysis results, but for more precise work the DATA RECOVERY part of the STATICS option should be used to obtain listings of the detailed element forces and stresses and nodal displacements.

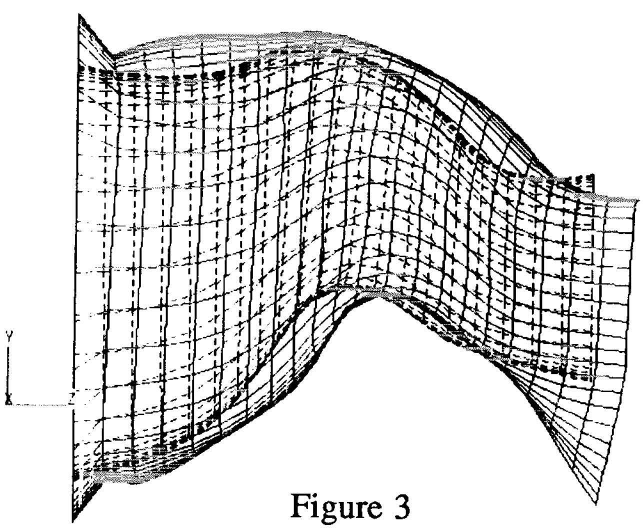


Figure 3
Deformed/undeformed geometry.

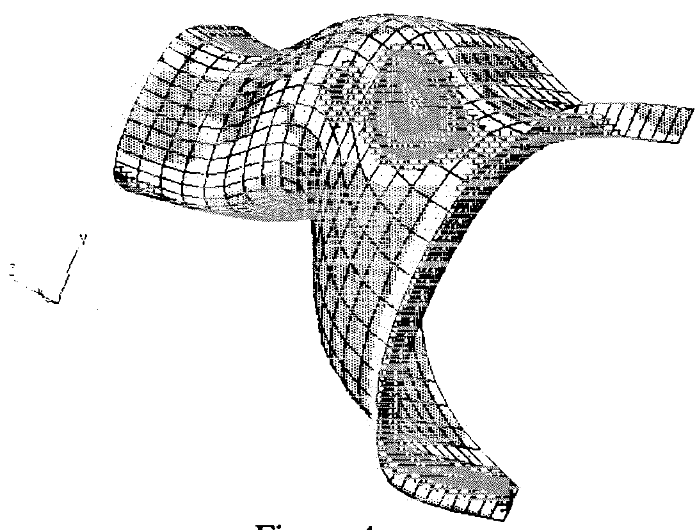


Figure 4
Stress contour plot.

This model contains 4092 equations and takes 51 minutes to complete the static analysis on a Macintosh IIX with 4MB memory.

A normal modes dynamic analysis was also performed. Calculation of the first six modes using subspace iteration takes 85 minutes. The first six modes are at 72, 77, 86, 80, 92, and 98 cycles per second.

The second example illustrates solid element usage. The model is a pulley loaded by a belt (figure 5). Loads are applied where the belt attaches and are reacted in the inside of the inner hub with balanced forces to obtain the proper load distribution. Again, only one half of the part is modeled due to its symmetrical shape and loading.

Figure 6 shows the deformed/undeformed shape. Figure 7 is a stress contour plot. Figure 8 is a stress versus element number xyplot using the GRAPHICAL/XYPLOT option.

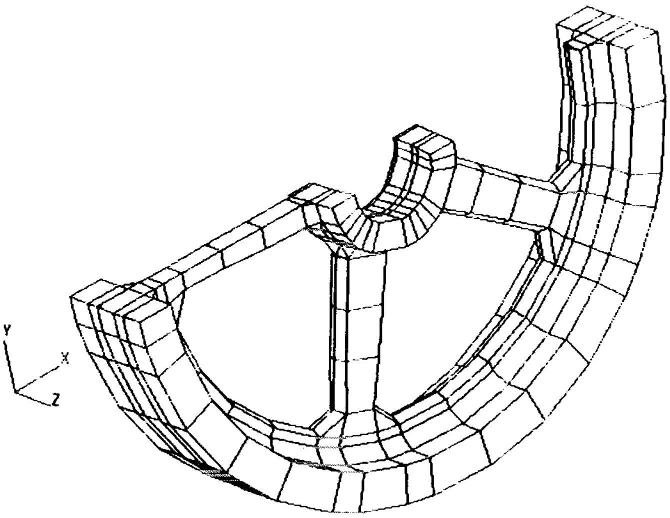


Figure 5
Pulley model geometry.

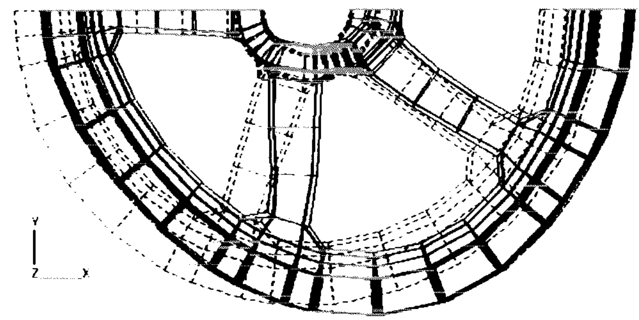


Figure 6
Deformed/undeformed shape

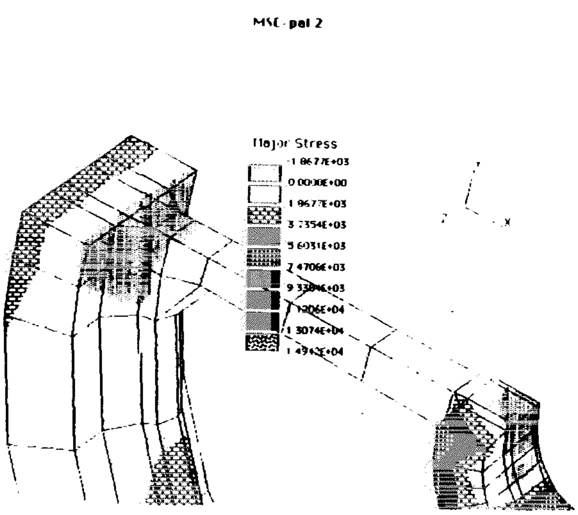


Figure 7
Stress contour.

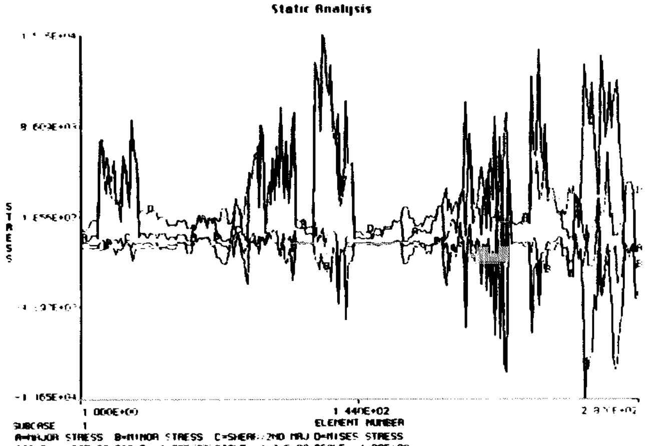


Figure 8
XYPLOT stresses.

The pulley model contains 525 nodes, 1775 equations and 287 solid elements. The static analysis takes 19 minutes to complete.

CONCLUSION

These examples show some of the new FEA features that are available to Macintosh users in the MSC/pal 2 Version 3.5 program. They include solid elements, thermal stress analysis, CAD interface and an easy to use Macintosh menu user interface. In addition, they show that finite element analysis of large models, with significant application to the engineering community, can be performed in an accurate, timely manner with this software.