

Interfacing CAD to MSC Programs

Deepak Brar, Robert Lentz, Jenny Fang, and Ken Blakely
The MacNeal-Schwendler Corporation

This paper describes interfacing CAD (Computer Aided Design) programs to MSC's structural finite element programs MSC/NASTRAN, MSC/XL, MSC/pal 2, and MSC/mod. Interfacing is done by reading a CAD geometry text file and writing an MSC/NASTRAN or MSC/pal 2 model file (containing elements and grid points), an MSC/XL input file (with geometry commands), or an MSC/mod element file (with structural elements or plot-only elements).

In particular, this paper shows how to interface AutoCAD geometry to MSC products via the microcomputer programs MSC/mod and CAD2MSC. Discussion is presented about which CAD entities can be successfully transmitted to MSC's programs. Several examples conclude the paper.

Introduction

CAD (Computer Aided Design) programs use geometric entities to represent structural designs. These geometric entities describe 0-D (points), 1-D (curves), 2-D (surfaces), and 3-D (volumes) geometry. Some CAD programs even have meshers, which break down an entity into multiple entities of the same or lower order (a surface broken into multiple, smaller surfaces or into a series of lines, for example).

The geometric description inherent in CAD programs suggest that they can be used as starting points in finite element modeling. The geometry can be translated from CAD programs to finite element modelers such as MSC/XL and MSC/mod, which are then used to create the element mesh, specify element and material properties, and apply loads and constraints. The CAD programs that contain some type of meshing can also be used to directly create elements used in FEA (finite element analysis) programs such as MSC/NASTRAN and MSC/pal 2.

Figure 1 illustrates two ways to interface AutoCAD and CADKEY, two popular microcomputer CAD programs, to MSC/NASTRAN, MSC/pal 2, and MSC/XL. One way is to use MSC/mod, a finite element preprocessor that runs on IBM PCs. Another way is to use CAD2MSC, a utility program that runs on IBM PCs and Apple Macintoshes. These programs are illustrated in the examples that follow.

Translation Methods

There are two ways to interface CAD to FEA: direct translation and indirect translation. Each method has advantages and disadvantages. These methods are illustrated in the examples that follow.

With direct translation, the CAD geometry is translated directly to finite element entities—grid points and elements. This method assumes that there is some sort of mesher in the CAD program, or that there is a one-to-one correspondence between each CAD entity and finite element. The advantage of direct translation is that no finite element preprocessor is required; the disadvantage is that additional user intervention is required to edit the FEA file to add element and material properties, loads, and constraints.

With indirect translation, the CAD geometry is not translated to finite elements but, rather, to geometry of a form that can be used by a finite element preprocessor such as MSC/XL or MSC/mod. The preprocessor, then, performs the meshing, specification of element and material properties, and application of loads and constraints. The advantage of this method is that CAD geometry is translated to geometry, rather than to elements, providing a more accurate translation. A further advantage is that the preprocessor can be used to generate geometry of a higher order than that supplied by the CAD program (for instance, MSC/XL can create a volume from two CAD-supplied surfaces). The disadvantage with this method is that a preprocessor is required.

Table 1 illustrates the various types of entities: CAD geometry, MSC/NASTRAN elements, and MSC/XL geometry. Note that most microcomputer-based CAD programs do not have true 3-D geometric entities; those are contained in solids modelers.

Table 1. CAD and Finite Element Entities

<u>Entity order</u>	<u>CAD entity</u>	<u>MSC/NASTRAN entity</u>	<u>MSC/XL entity</u>
0-D	POINT	GRID	Point
1-D	LINE	BAR	Curve
	3DLINE	BAR	Curve
	ARC CIRCLE	Multiple BARs Multiple BARs	Mult. Curves Mult. Curves
	POLYLINE	Multiple BARs	Mult. Curves
2-D	TRACE	QUAD4 or TRIA3	Surface
	SOLID	QUAD4 or TRIA3	Surface
	3DFACE	QUAD4 or TRIA3	Surface

Example 1: Lifting Lug Model

The structure in this example is a lug comprised of a hollow metal tube connected to a planar portion. The geometry consisted of 3DFACEs, which were created in part by using AutoCAD's "mesher." The geometry file was then written using AutoCAD's DXF format.

CAD2MSC was used to read the DXF file and translate directly to MSC/pal 2 elements (quadrilateral and triangular plates). The MSC/pal 2 file created by CAD2MSC consisted of the grid points and plate elements, which was then edited in MSC/pal 2 to add element and material properties, applied loads, and constraints. After that, MSC/pal 2 was run for static analysis.

Figure 2 shows the AutoCAD geometry as well as the resulting MSC/pal 2 stress contours. AutoCAD, CAD2MSC, and MSC/pal 2 were all run on an Apple Macintosh.

Example 2: Bracket Model

The structure in this example is a metal bracket, comprised of several thick plates with bolt holes. Because it was desired to construct the MSC/pal 2 model out of solid elements, and AutoCAD does not have solids, the indirect translation method was used. The bracket outline was constructed in AutoCAD by using LINES, CIRCLES, and ARCS. The geometry was then written in DXF format.

MSC/mod was used to read the DXF file, use the model outline for meshing with solids, add the element and material properties, and apply loads and constraints. Then, MSC/pal 2 was run for static analysis.

Figure 3 shows the AutoCAD geometry as well as the resulting MSC/pal 2 stress contours. AutoCAD, MSC/mod, and MSC/pal 2 were all run on an IBM PC.

Example 3: Hemisphere Model

The structure in this example is a hemisphere with a hole at the top. This structure was "meshed" in AutoCAD into 3DFACEs, and a DXF file was written. AutoCAD was run on an Apple Macintosh.

CAD2MSC was also run on the Macintosh to read the DXF file and to write an MSC/XL input (INP) file. This file contains the MSC/XL geometric construction commands, in text format. The commands in this example create a surface from each 3DFACE; these commands first define Points, then Curves, and then Surfaces.

The INP file was written on the Macintosh to a floppy disk in DOS format, and was then read by MSC/XL on an 80386-based IBM PS/2 running 386/ix, a UNIX operating system. The INP file re-created the geometry in MSC/XL, from which the user performed meshing, specified element and material properties, applied loads and constraints, and

wrote the MSC/NASTRAN file. MSC/NASTRAN was run on the 80386 machine, and MSC/XL was then used to display the results.

Figure 4 shows the AutoCAD geometry, the MSC/XL geometry, and the MSC/NASTRAN stress contours.

Observations

CAD programs can provide convenient starting points for finite element modeling, but that is only what they are—starting points. Many CAD drawings will have fillets, notches, and other small features which may not be applicable to your model. Therefore, it is best to have in mind the type of FEA model you want at the time you are creating your CAD drawing.

As another observation, the indirect method is probably best, since: (1) it provides a one-to-one translation (i.e., geometry to geometry); and (2) you can use the preprocessor to add engineering properties (element and material properties, loads, and constraints) and clean up your model (eliminating coincident grid points, for example).

As a final observation, it is wise to practice on small drawings with few entities prior to tackling production-type drawings. This could save unwanted surprises in the long run.

Acknowledgements

We thank Juliette Gutierrez and Wai Ho for their assistance in helping to prepare the models and hardcopy.

MSC and MSC/ are service and trademarks of The MacNeal-Schwendler Corporation. NASTRAN is a registered trademark of the National Aeronautics and Space Administration. MSC/NASTRAN is an enhanced proprietary version developed, marketed, and supported by MSC. MSC/XL is a trademark of MSC. MSC/pal 2 is a registered trademark of MSC. MSC/mod is a trademark of MSC. AutoCAD is a registered trademark of Autodesk, Inc. CADKEY is a registered trademark of CADKEY, Inc. IBM and PS/2 are registered trademarks of International Business Machines Corporation. Macintosh is a trademark of Apple Computer, Inc. DOS is a trademark of Microsoft Corporation. 80386 is a trademark of Intel Corporation. UNIX is a registered trademark of AT&T. 386/ix is a trademark of INTERACTIVE Systems Corporation.

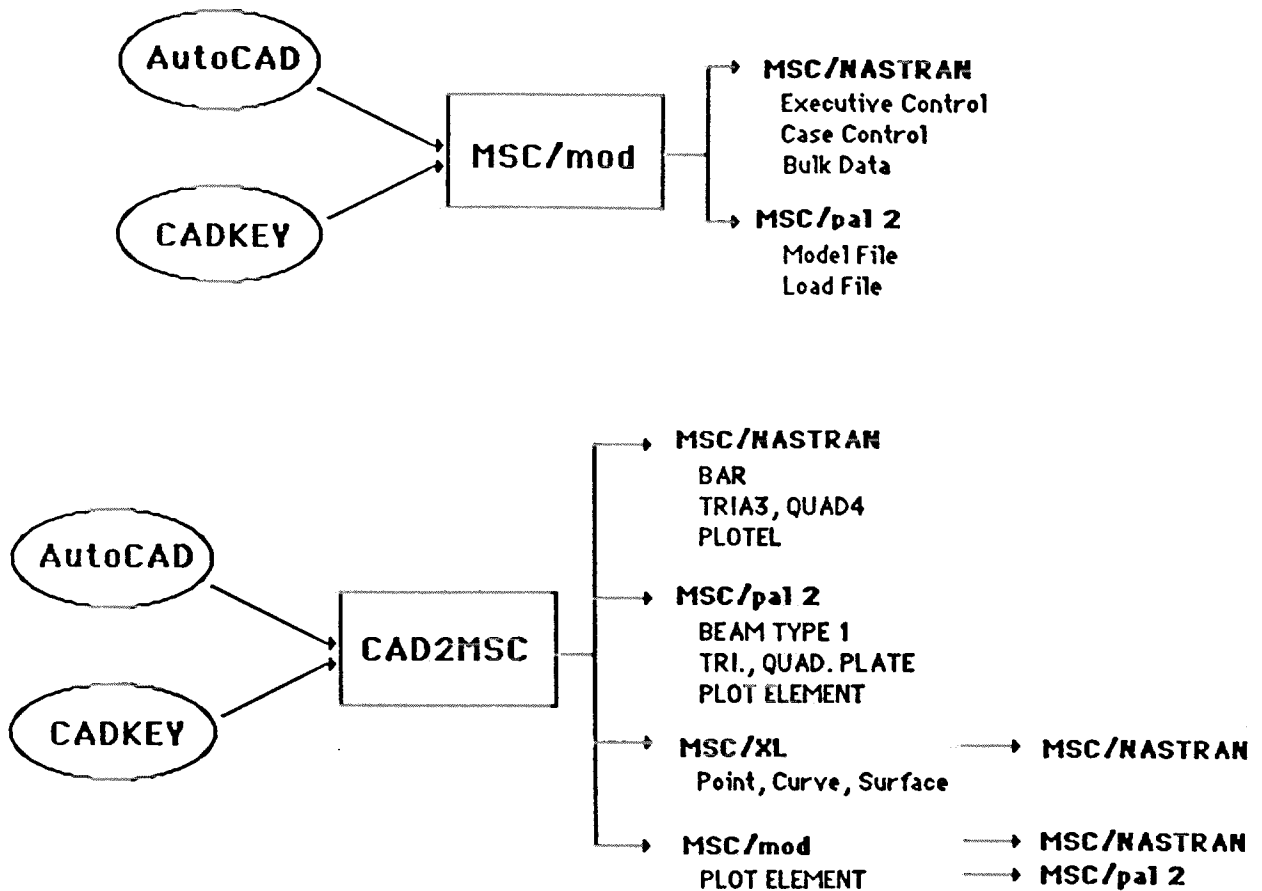


Figure 1. MSC/mod and CAD2MSC interface AutoCAD and CADKEY to MSC's products. MSC/mod performs a direct translation, allowing the user to add element and material properties, loads, and constraints. CAD2MSC creates only elements and grid points (for direct translation) or geometry for MSC/XL and MSC/mod (which are then used to create the MSC/NASTRAN and MSC/pal 2 files).

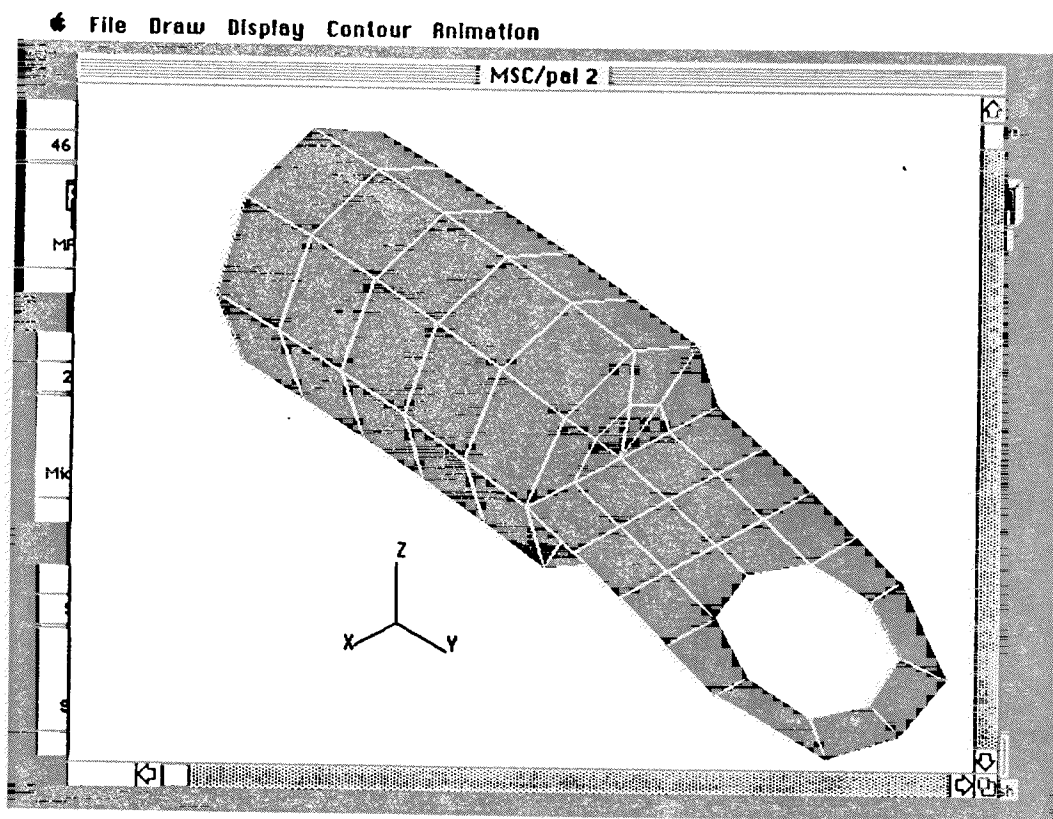
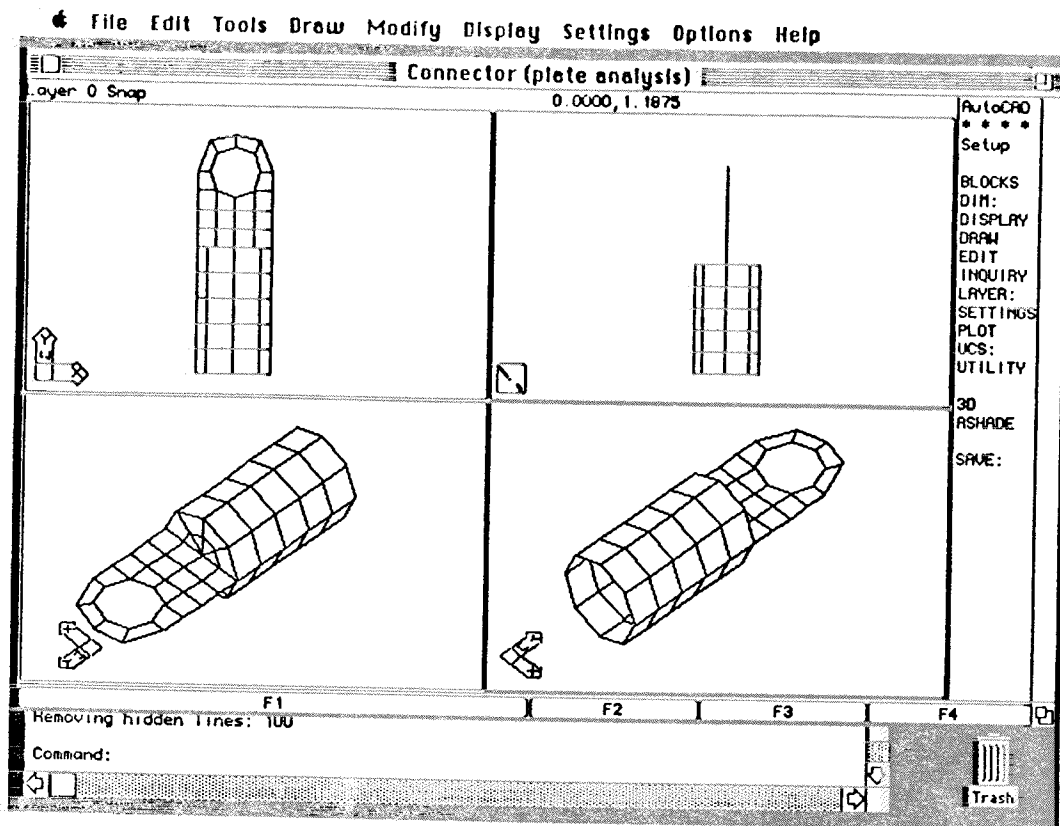


Figure 2. (Top) AutoCAD drawing of the lifting lug.
(Bottom) MSC/pal 2 model of the lifting lug.

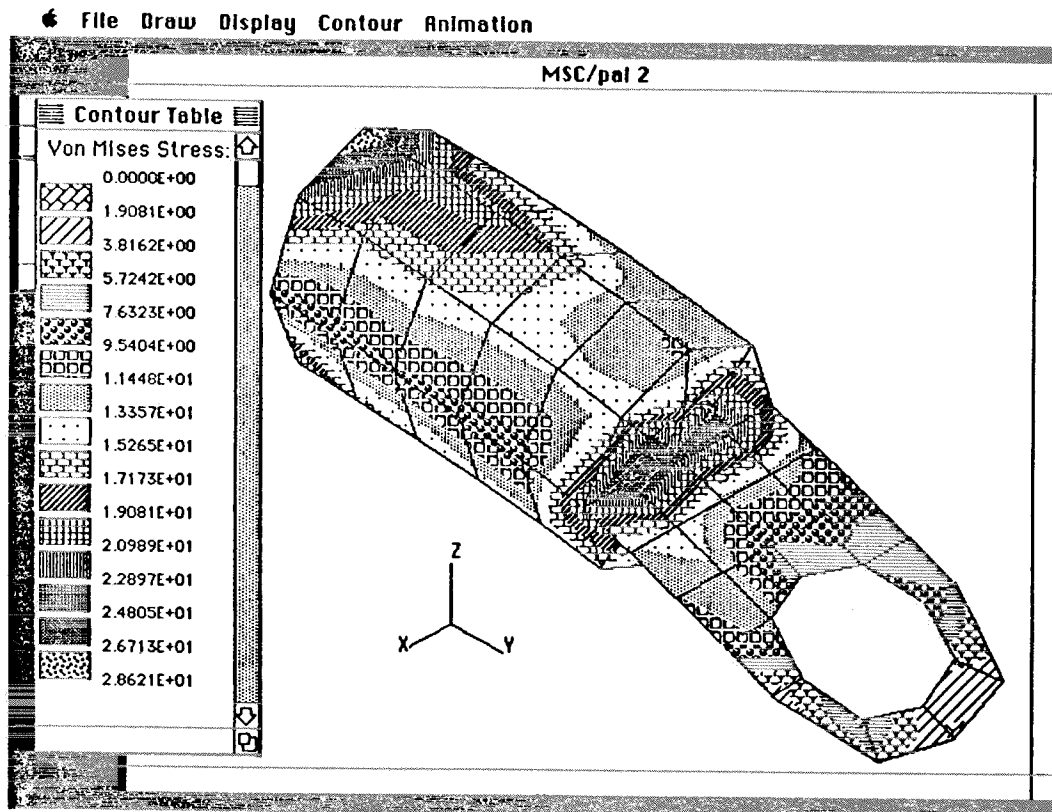


Figure 2 (contd.). Stress contours computed with MSC/pal 2.

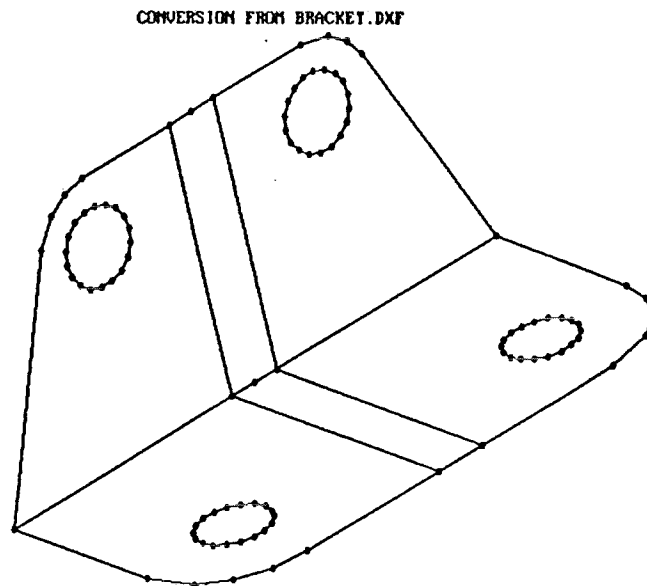
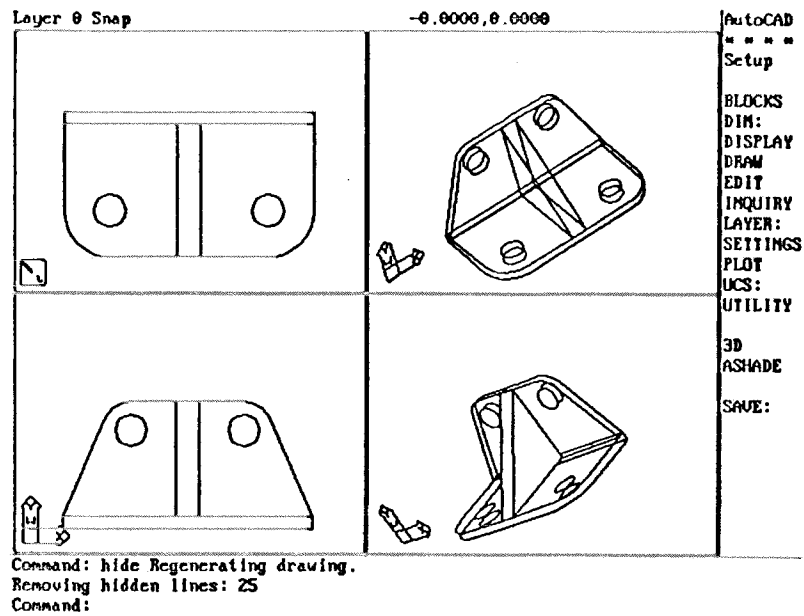


Figure 3. (Top) AutoCAD drawing of the bracket.
 (Bottom) Bracket geometry, read and reproduced in MSC/mod.

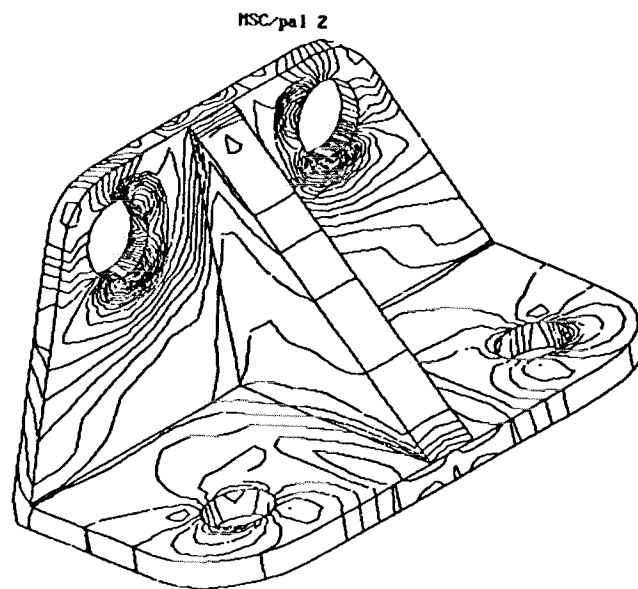
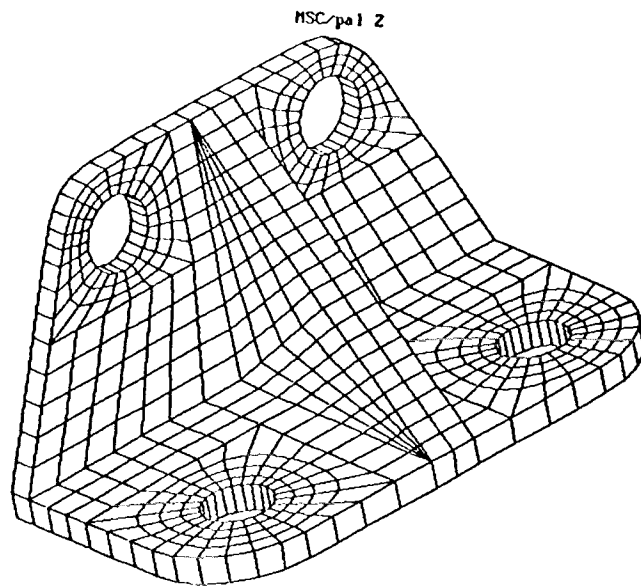


Figure 3 (contd.). (Top) MSC/pal 2 bracket model.
(Bottom) Bracket contours computed and displayed in MSC/pal 2.

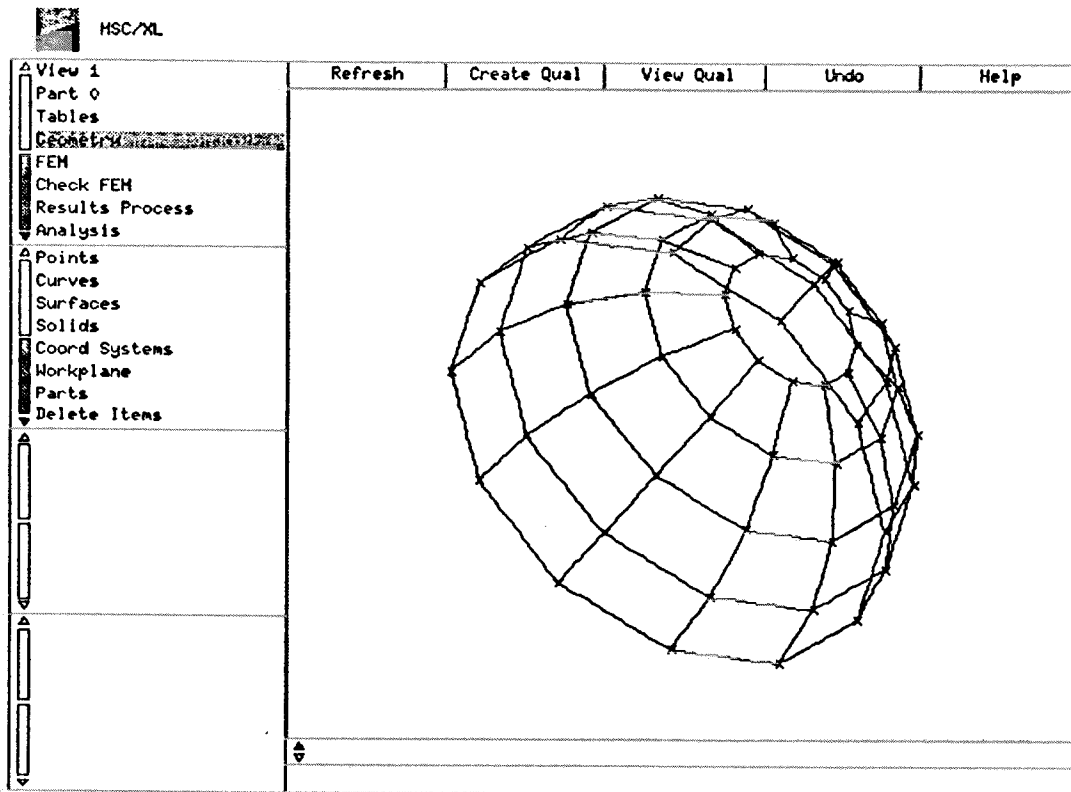
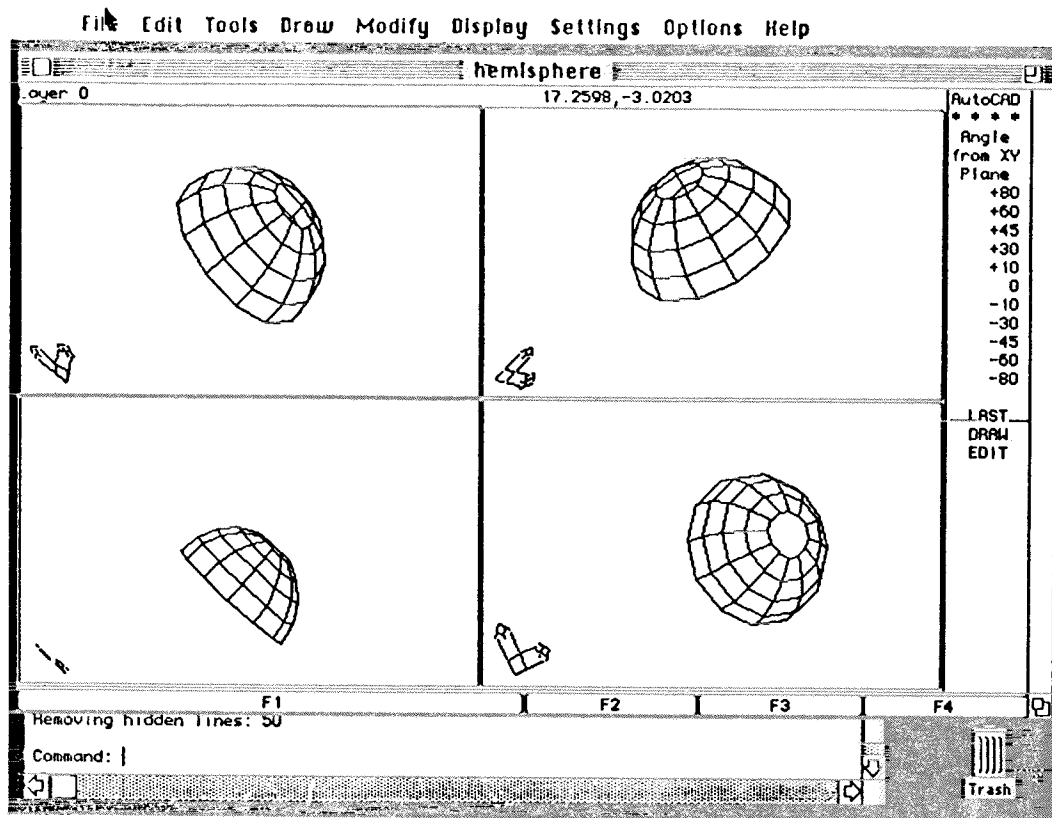


Figure 4. (Top) AutoCAD drawing of the hemisphere.
(Bottom) Hemisphere geometry, read and displayed in MSC/XL.

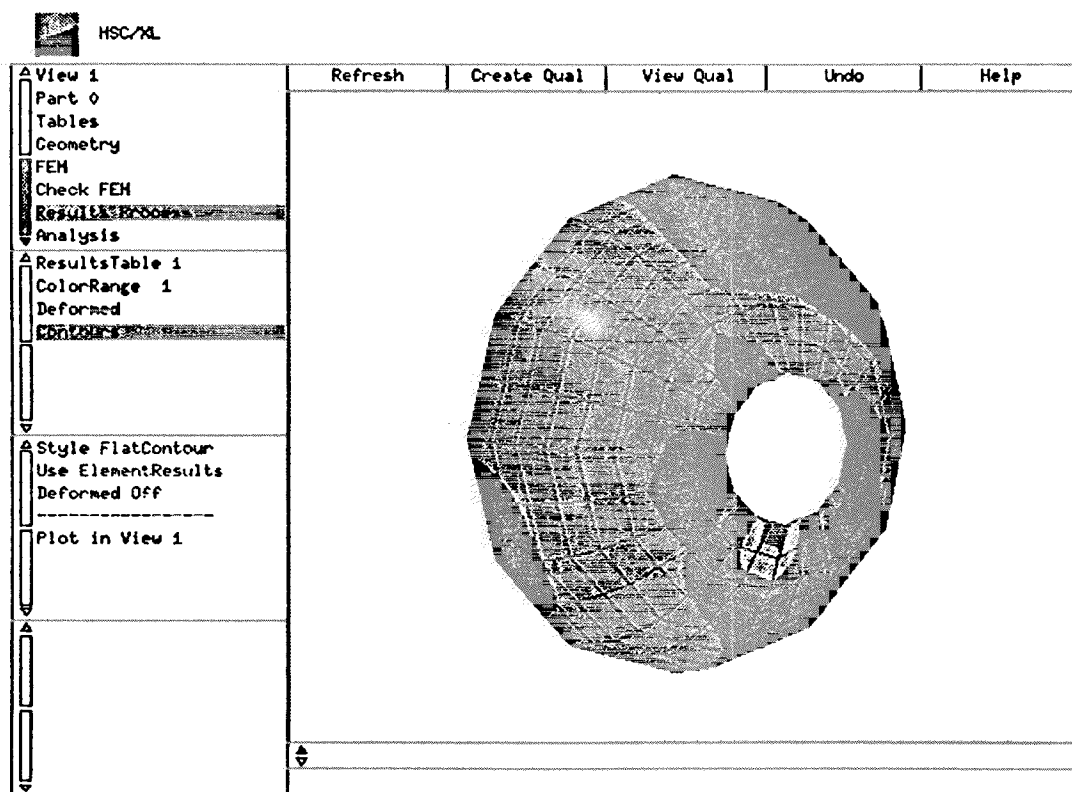
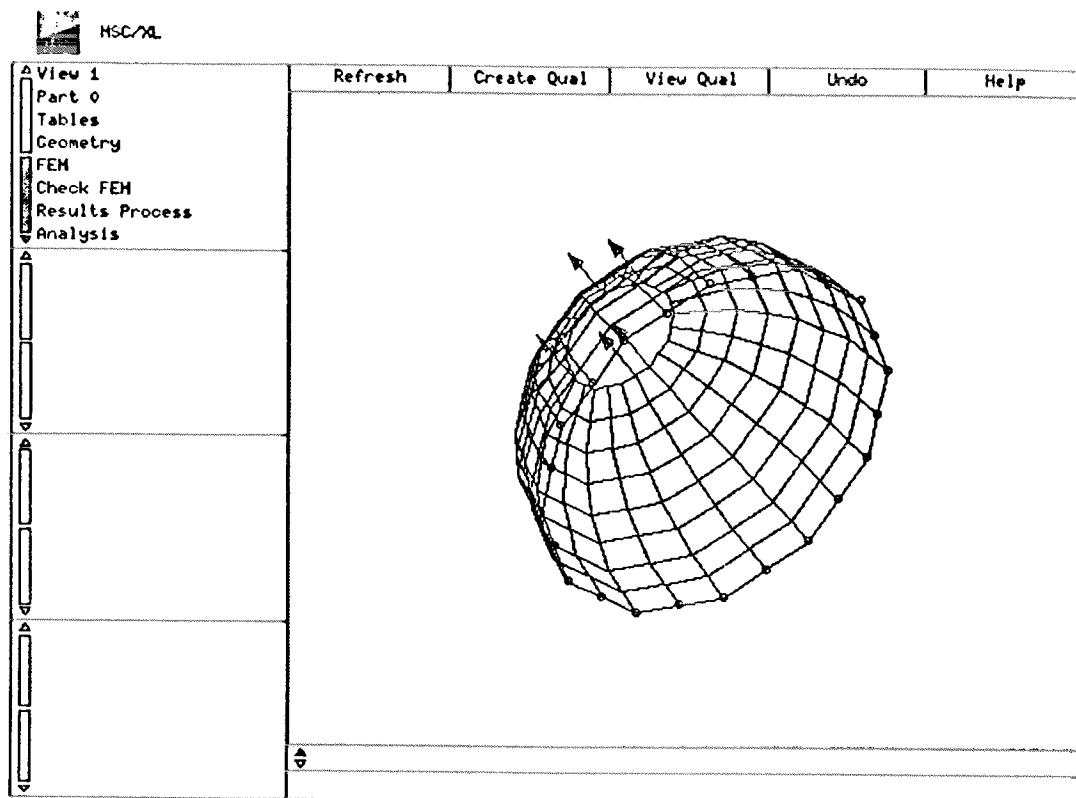


Figure 4 (contd.). (Top) Hemisphere finite element model, created in MSC/XL.
 (Bottom) Stress contours, computed with MSC/NASTRAN and displayed with MSC/XL.