L A P C A D 3

- A modeling program for MSC/pal -

Gert M. Lundgren

Chula Vista, California, December 12, 1987

Abstract

LAPCAD3 is a modeling program for interaction with MSC/pal on the Macintosh.

Objects are created and viewed in 3-D solid and shaded perspective. The finite element modeling features include the creation of nodes in the 3-D space. The nodes are connected into bars and triangular and quadrilateral plates.

Models are stretched, copied, mirrored, moved, rotated or erased. Element properties, materials, boundary conditions and external loads are implemented under mouse control. Element shrink mode and hidden line removal is implemented. A standard Macintosh-like interface is utilized.

Stiffness and loads data is saved and printed in either MSC/pal or MSC/NASTRAN format.

(1) INTRODUCTION:

LAPCAD3 was developed by a structural analyst, having more than twenty years of stress analysis experience in the aerospace industry. This experience has been implemented in the LAPCAD3 program to make it an intuitive modeling tool, requiring a minimum of training or finite element background. The following discussion explains some of the features available within LAPCAD3.

(2) MODEL GEOMETRY AND STIFFNESS:

The construction of a model may take several different routes or combinations thereof. A model may start at the lowest level via the digitizing of nodes, with subsequent addition of connectivities, properties and materials.

It can also be automatically generated by utilizing the built-in Standard Object Modeler.

Finally an existing model may serve as a starting point, to which additions or modifications are made. Existing or newly created model entities can also be merged.

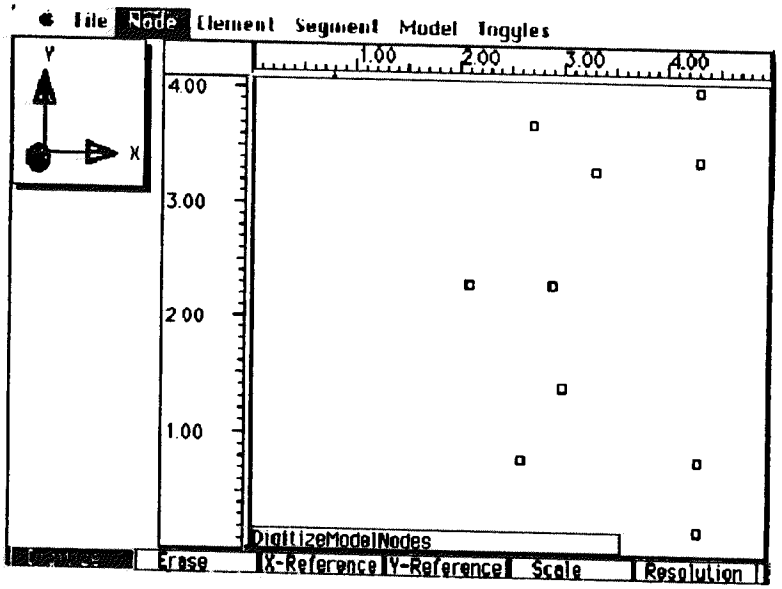
(2.1) Construction via Digitizing:

(2.1.1) Select Nodes:

Activating 'Digitize' under the 'Node' pull-down menu provides the user with a sub-menu along the lower edge of the screen. This menu allows the user to set the scale, as well as to select the global coordinates for the local digitizing coordinate system.

The node creation is accomplished by moving the cursor to a desired location on the screen, while watching the x and y tickmarks moving along the horizontal and vertical scales. When a satisfactory position has been found, the button on the mouse is depressed, resulting in a small square being drawn on the screen. Additional nodes are constructed in the same fashion.

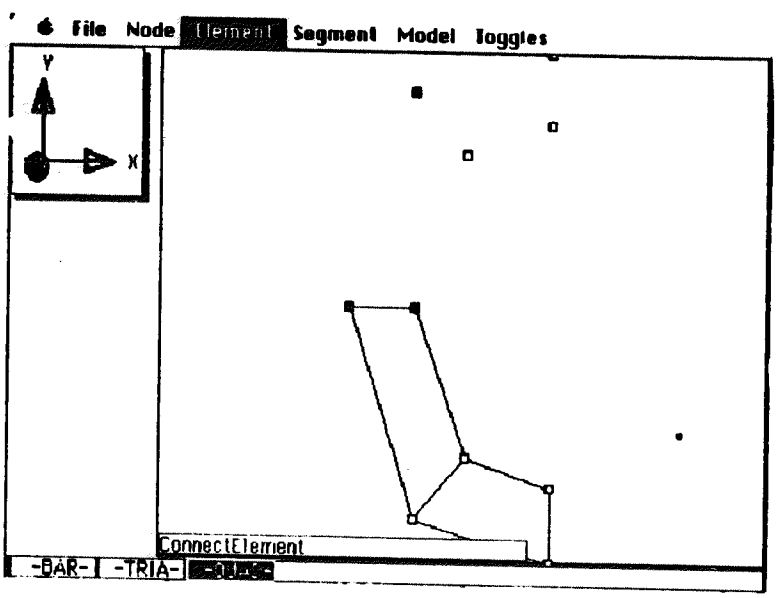
When all nodes have been input, exit the 'Digitize' environment by clicking the mouse cursor over the upper main menu bar.



(2.1.2) Connect Elements:

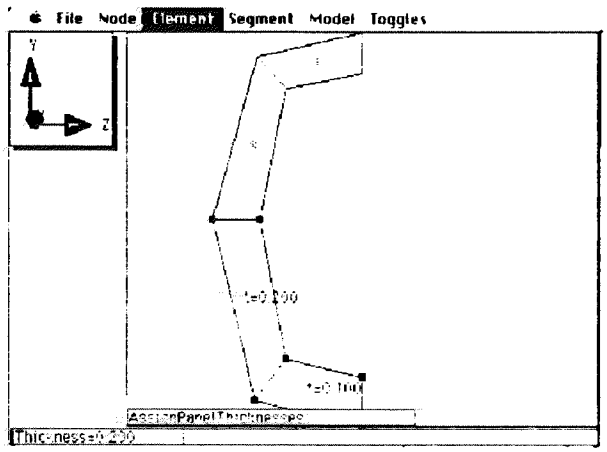
The next step is to create connectivities, and this is initiated by activating 'Connect Element' under the 'Element' pull-down menu.

Three element choices, BAR TRIA and QUAD, appear in the lower menu. Either is activated by clicking on the applicable box. Elements are then created by clicking in the proper order on the nodes that should be connected. When all desired connections have been made, again exit by clicking on the main menu bar.



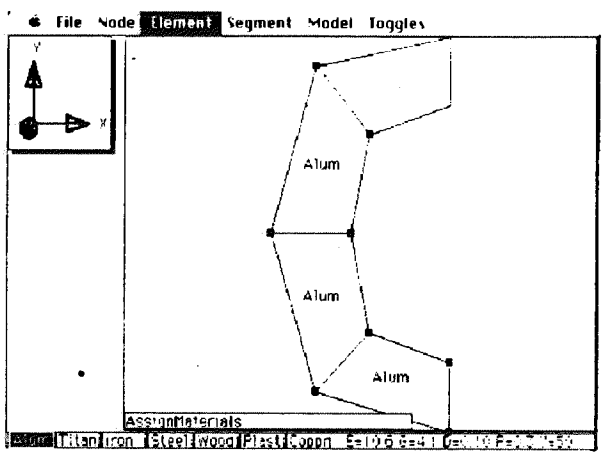
(2.1.3) Assign Properties:

Now pull down 'Assign Properties' under the 'Element' pull-down menu, and proceed by setting the desired thickness using the dial box. Then click in the vicinity of the center of each element, for which this perticular property setting applies. When completed, exit by clicking on the main menu bar.



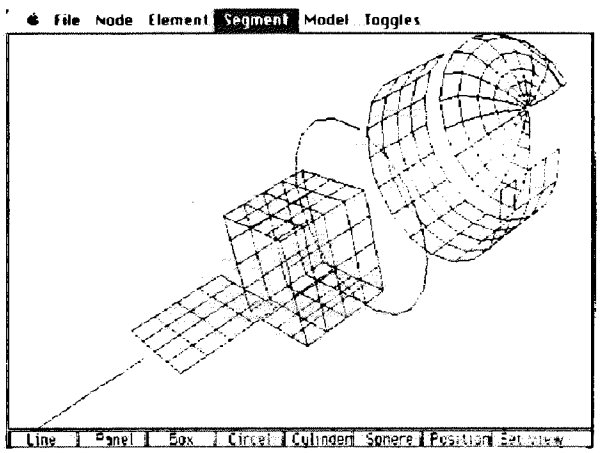
(2.1.4) Assign Materials:

Selection of materials is made under 'Assign Material', again under the 'Element' pull-down menu. In addition to user-defined materials lists, seven default choices are available, and either is activated by clicking on the applicable box. Materials are then assigned by clicking in the vicinity of the center of each element. The properties for the active material are shown to the right in the lower menu bar.



(2.2) Construction via Standard Object Modeler:

The Standard Object Modeler is activated with 'Create Model Segment' under the 'Segment' pull-down menu. After setting the desired global orientation, select the preferred type of model by clicking on the appropriate box. Applicable preset parameters will then appear, that can be dialed to any alternate user-selected value. Clicking on the 'Go-Ahead' box will draw the selected model segment, while 'Remove' will remove it.



(2.3) Construction based on an existing model:

Read a MSC/pal model or a MSC/NASTRAN model by activating either of 'Read MSC/pal Model' or 'Read MSC/NASTRAN BULKDATA' under the 'File' pull-down menu. Models developed for either of the MSC/pal or the MSC/NASTRAN finite element programs can also be merged within LAPCAD3, by simply reading each model and, when prompted, select the option 'Add to Existing Model'. Completion of the model to the user's specifications can then be accomplished.

In reading existing models created manually or by other sources, certain advanced features may not currently be translated by the LAPCAD3 modeling program. As an example LAPCAD3 is not equipped with coordinate card equivalents. When a certain model segment needs to be oriented radically different from the rest of the model, this segment may first be modeled in an orientation most convenient to the user, and then rotated and translated to a desired position. Other examples of translational limitations include MPC equations, orthotropic material selection, and others. The LAPCAD3 program will expand in translational capability as the user community demands it.

(3) MODIFICATION OF AN EXISTING MODEL:

There are several ways to modify an existing model - nodes as well as elements may be manipulated. In addition, groups of elements or model segments can be extensively modified.

(3.1) Node-related changes:

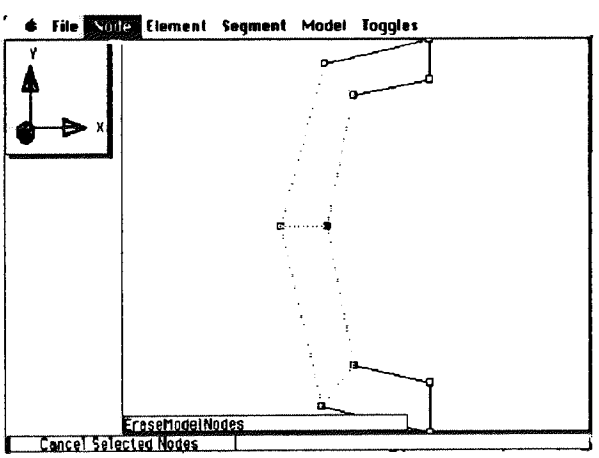
(3.1.1) Add Node:

Nodes may be added to an existing model by activating 'Add Node' under the 'Node' pull-down menu. The emerging lower menu bar allows the user to position a local coordinate system on the existing model. The origin is selected by clicking on the node that the user wants to be the center for the manipulation. The second node that the user will select defines the direction of the local x-axis, and the third selected node will define the plane in which the local y-axis will reside.

The user can now add nodes in this local plane, by moving the cursor around, and defining points by clicking on the mouse button. This activity is similar to what was described for digitizing of nodes. Exit is made by clicking on the upper main menu bar.

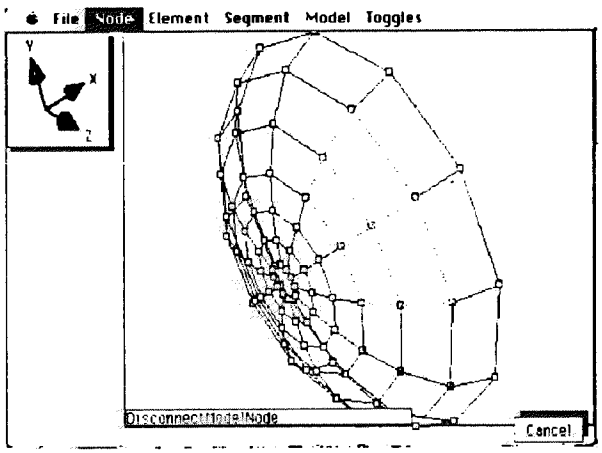
(3.1.2) Delete Node:

Elimination of nodes is made by entering the 'Delete Node' under the 'Node' pull-down menu. Nodes are then erased by simply clicking with the mouse button. Selected nodes will turn black, as a verification that contact was made. If 'Cancel' is activated, all deletions will be canceled that were selected after the last entry into this mode.



(3.1.3) Disconnect Node:

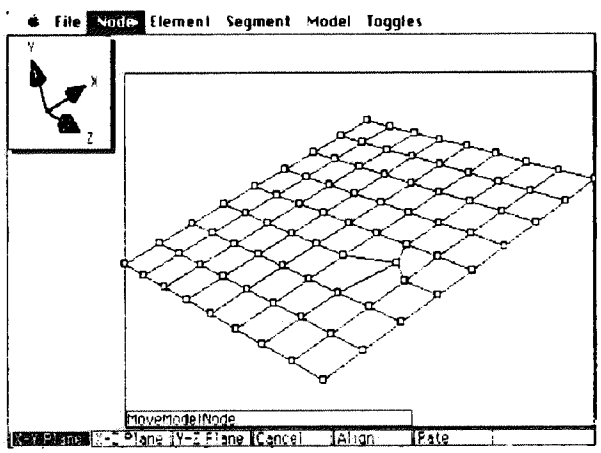
All elements can also be disconnected from an individual node, by entering 'Disconnect Node', under the 'Node' pull-down menu. Upon the selection of a node by clicking on it, all elements that were connected to that node will turn dim, as a verification. All disconnections can be cancelled before leaving this mode of operation. Clicking on the upper main menu bar provides an exit from this mode.



(3.1.4) Move Node:

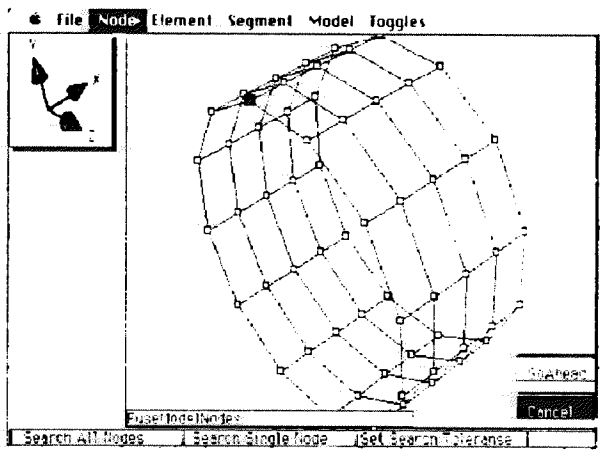
Any node can be moved in the 3-D space by entering the 'Move Node' environment under the 'Node' pull-down menu. The user first selects one of three planes in which the movement of the nodes will take place, and the user can then at his option align the model to this plane, or leave it as is.

Any node that is visible on the screen can then be moved by depressing the mouse button while the cursor resides over it. If the user then moves the mouse while the mouse button remains depressed, the node will move along, and element connectivities will be updated in real time. Clicking on the upper main menu bar again provides an exit from this mode.



(3.1.5) Identify and eliminate duplicate node:

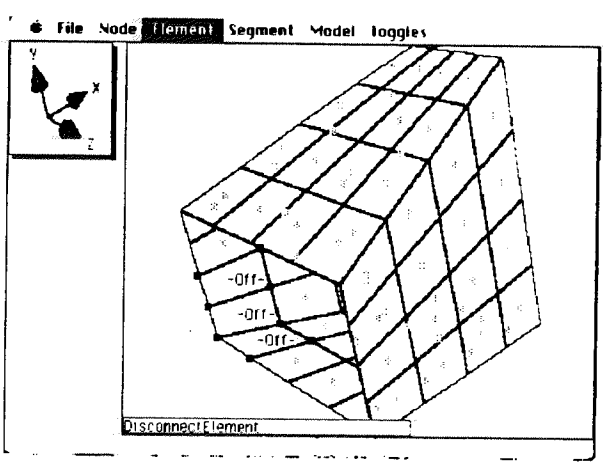
As model segments are copied and added to the original model, coincidental nodes may result. By entering 'Fuse Nodes' under the 'Node' pull-down menu, such duplications can be eliminated. The size of the search space is preset to a small percentage of the size of the model, however it can be adjusted to any magnitude by the user.



(3.2) Element-related changes:

The element-related change is limited to the disconnection of elements. This change differs from the disconnection of a node, in that disconnection of an element is limited to one element at a time, while disconnection of a node affects all elements connected to the node being focused on.

Activate 'Disconnect Element' under the 'Element' pull-down menu. The approximate center of each element is highlighted with a shaded square. By clicking on the center of a certain element, that element will be disconnected. Clicking on the upper main menu bar provides an exit from this mode.



(3.3) Segment-related modifications:

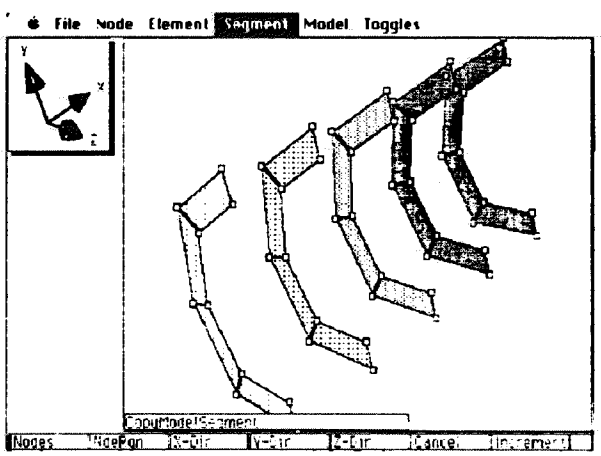
A model segment is any entity of nodes or nodes with connectivities. Such a segment can be copied, mirrored, moved, rotated or expanded/contracted. A segment selected in any of these change modes maintains its identity until redefined.

(3.3.1) Copy Segment:

Activate the 'Copy Model Segment' under the 'Segment' pull-down menu. The segment that should be duplicated is identified by clicking on the bounding nodes.

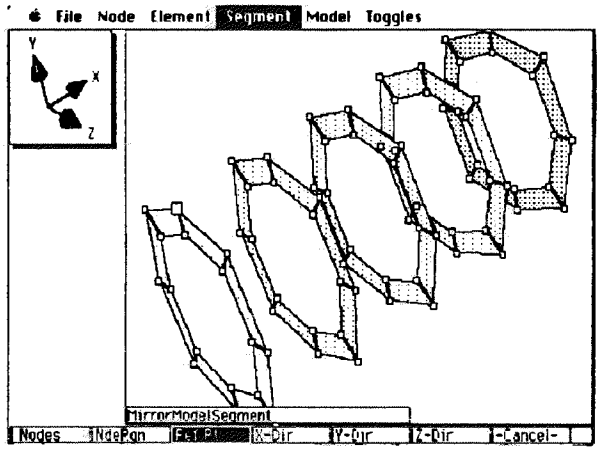
Before a copy of the segment is made, the user may adjust the default distance that will separate each segment, by dialing the 'Increment' button. Then a copy is created by simply pressing one of the x, y or z buttons. Additional copies are created by pressing repeatedly on one of these buttons.

Erasure of copies created as described above is accomplished by pressing the 'Cancel' button. Exit is again accomplished by clicking on the main menu bar.



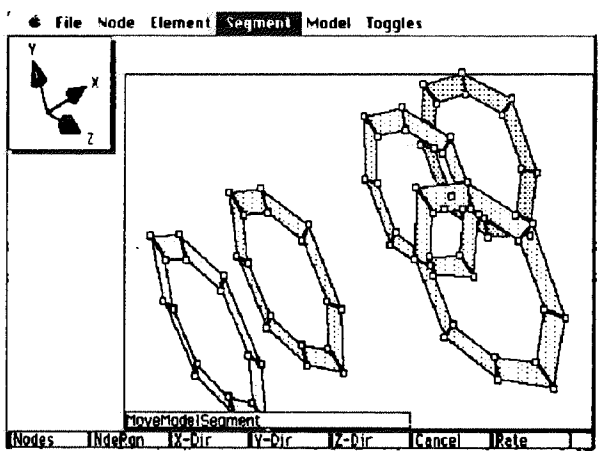
(3.3.2) Mirror Segment:

Activate the 'Mirror Model Segment' under the 'Segment' pull-down menu. Then select nodes that represent the segment that should be mirrored. Next select a reference point relative to which the mirroring will take place. Finally select the x, y, or z direction for segment mirroring. This last step results in the segment being mirrored as specified. Clicking on the same direction once more, will result in the mirroring reversing itself.



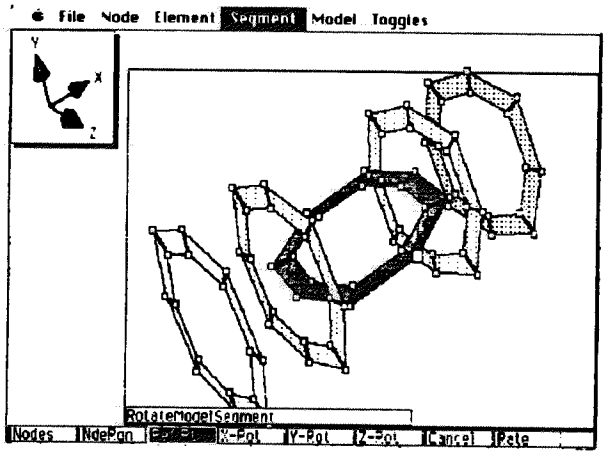
(3.3.3) Move Segment:

Activate the 'Move Model Segment' under the 'Segment' pull-down menu. Proceed with selection of nodes as described above. Then select direction of the move. The cursor is now placed in the general drawing area, clicked and then dragged, resulting in the selected segment moving along in real time. Clicking once more freezes the segment at the new location. The rate of movement can be adjusted with a dial, if the 'Rate' button is clicked.



(3.3.4) Rotate Segment:

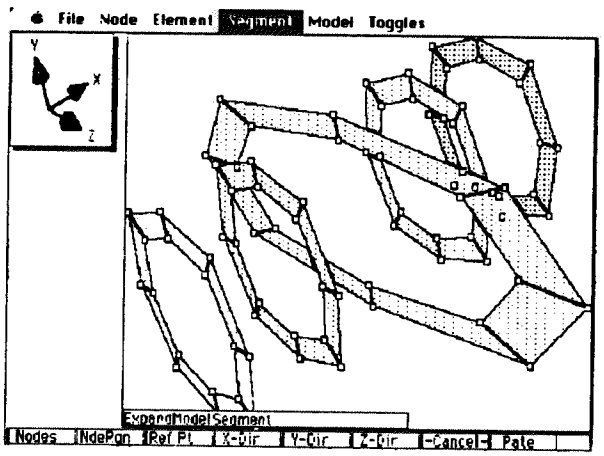
Rotate follows the same sequence of activities as move, with the exception that the segment is rotated around an axis parallel to the selected axis instead of moved along it. The user-selected reference point serves as a pivot for the rotation. This activity is initiated with the 'Rotate Model Segment' under the 'Segment' pull-down menu.



(3.3.5) Expand Segment:

This procedure is activated with 'Expand Model Segment' under the 'Segment' pull-down menu. This manipulation is similar to the above procedures. If a reference point is selected in the middle of the segment, the segment grows or shrinks equally on either side of that point in the user selected direction.

If the reference point is selected as one of the nodes at one extreme of the selected model segment, the segment grows or shrinks in proportion to the distance from that point. In other words the reference point remains unchanged while the segment around it changes in proportion to their distance to the reference point.

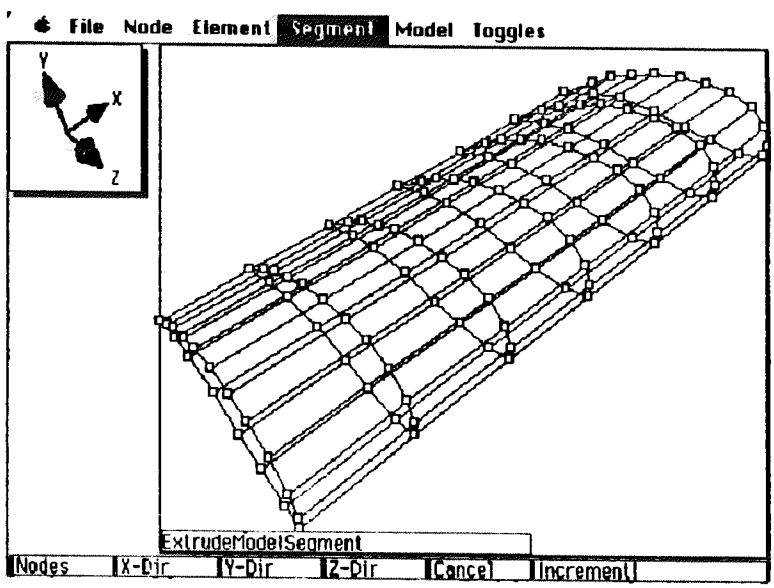


(3.3.6) Extrude Segment:

'Extrude Model Segment' under the 'Segment' pull-down menu allows the user to develop a tubular model segment with an arbitrary cross section. An example of such a structure is an airplane wing.

The arbitrary cross section is first constructed by digitizing a selected number of nodes. These nodes are then clicked in the sequence desired, and a pitch is dialed under the 'Increment' button.

Finally a model segment is extruded by clicking on either of the x, y or z direction buttons. Additional segments are created by repeated clicking on these buttons.



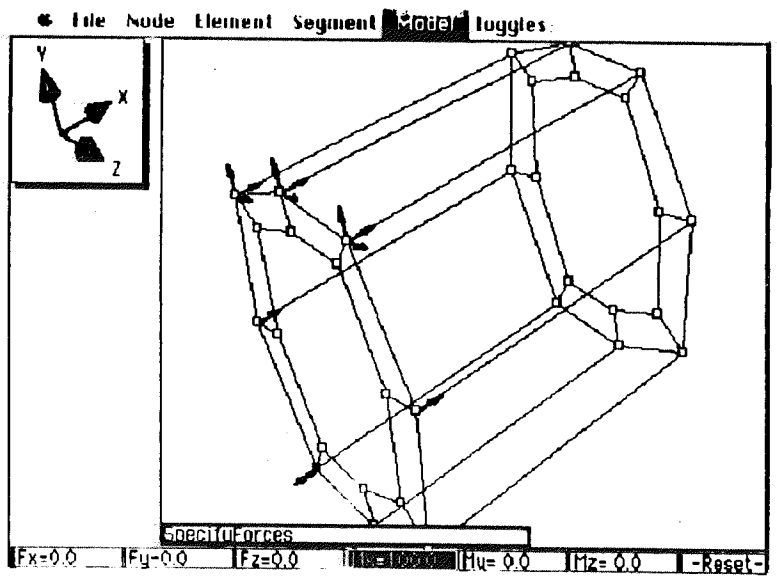
(4) EXTERNAL LOADS:

As with all other input to the LAPCAD3 modeling program, external loads are entered entirely under mouse control.

(4.1) Apply Node Forces:

Execute 'Apply Node Forces' under the 'Model' pull-down menu. The lower menu bar then shows six entries for x, y, and z forces, as well x, y, and z moments, respectively. These are all initially preset to zero. By clicking on these values the user can dial the proper combination of forces.

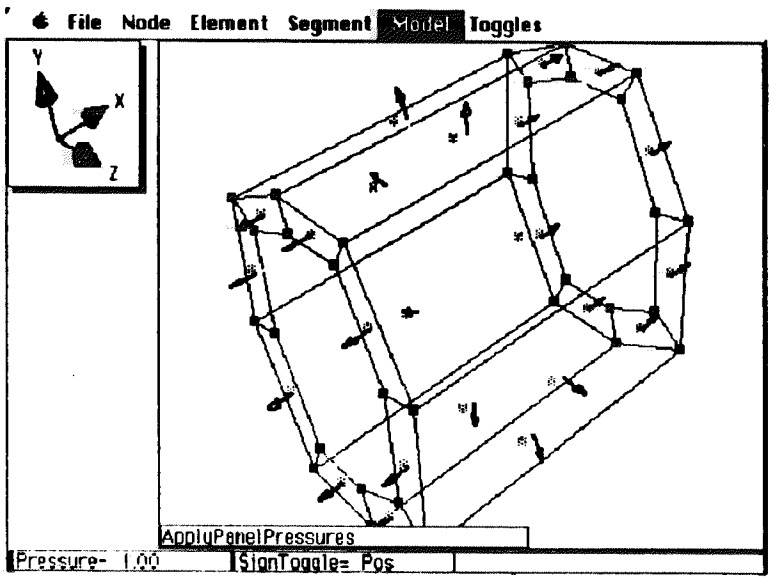
If clicking on any of the nodes, this force setting will be applied to that node, and the relative magnitude will be highlighted in the form of proportional length arrows in the applicable directions. The force setting will remain the same for all nodes that are clicked on, until the user adjusts them to a different value. Exit is provided from this mode by clicking on the upper main menu bar.



(4.2) Apply Panel Pressures:

This procedure is activated with 'Apply Panel Pressures' under the 'Model' pull-down menu. The magnitude of the applied pressure is set by the user via a dial. Pressures are then applied to individual panel elements by clicking in the proximity of the element center. A force vector will be displayed, with a direction perpendicular to the panel surface.

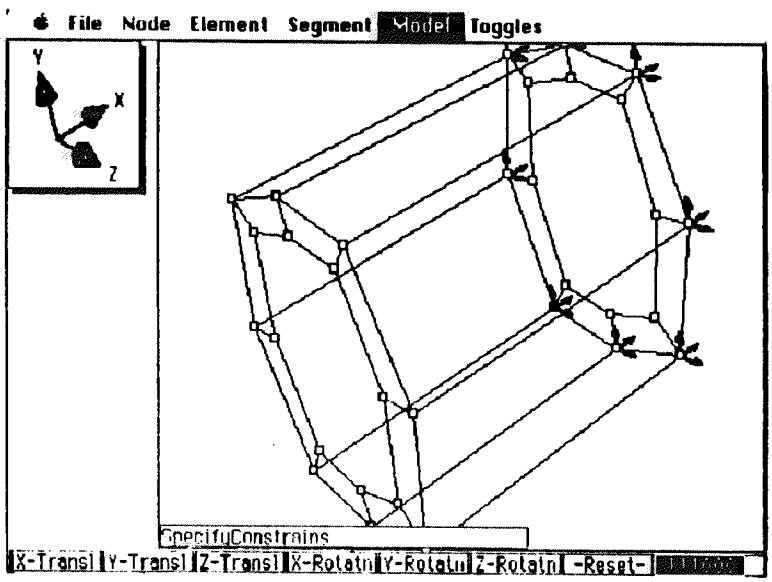
The direction will tell the user if the correct sign has been applied. Since the direction in MSC/pal as well as in MSC/NASTRAN is dependent on the order in which the element is connected to the nodes, it is possible that the resulting direction is opposite to what the user expected. In that case the user should dial a change in the sign of the applied pressure, and then again click on the center of the element. Exit is provided by clicking on the upper main menu bar.



(5) BOUNDARY CONSTRAINTS:

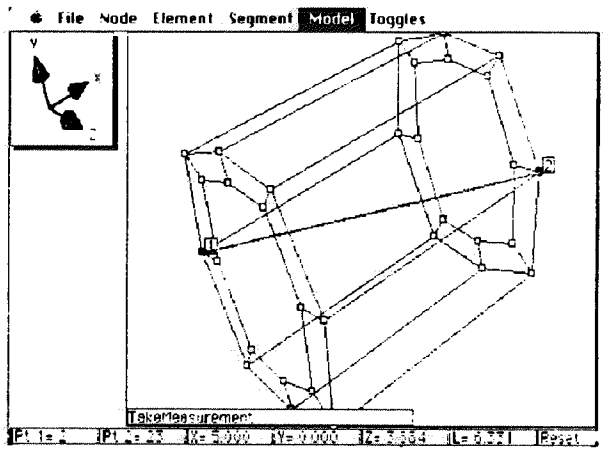
Application of constraints is similar in procedure to that of applying external node forces. The routine is activated with the 'Apply Node Constraints' under the 'Model' pull-down menu. The lower menu bar shows six boxes that represent translational and rotational constraints in the x, y and z directions.

All boxes are initially set to zero, and a summary of the setting is shown to the far right in the lower menu box. By clicking on any or all of these boxes, the user can toggle the values to either zero or one, with the summary to the right adjusted accordingly. By clicking on any node, the current setting of constraints is applied to that node. Verification at that node is shown in the form of arrows. Exit is again provided by clicking on the upper main menu bar.



(6) MODEL MEASUREMENTS:

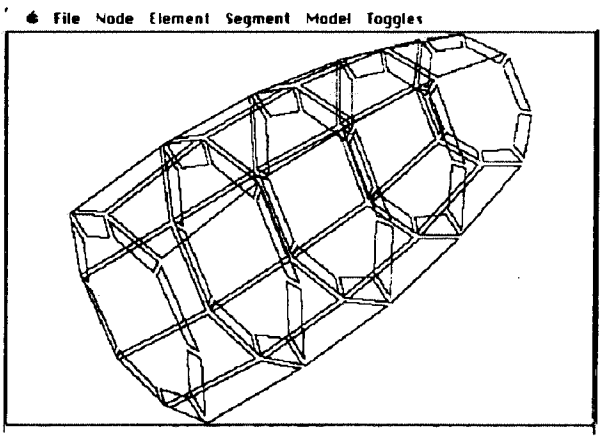
This procedure is activated with the 'Take Measurements' under the 'Model' pull-down menu. By clicking on two nodes the user computes the coordinate differentials and the absolute distance between these two nodes. The resulting quantities are displayed in the lower menu bar. In order to increase visibility, a shaded line is also drawn between the two selected points.

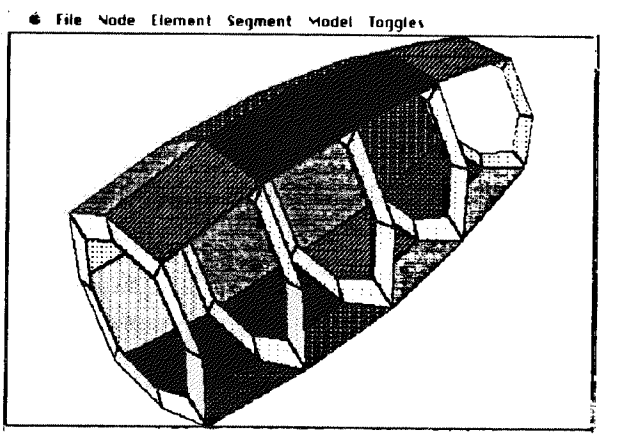


(7) VIEWING OF THE ENTIRE MODEL:

If the user desires to redraw the model, he can do so at any time with 'Draw Model' under the 'Model' pull-down menu. Additional viewing options are available under the same main menu under 'Rotate Model' and 'Zoom in on Model Detail'.

Enhancements such as element shrink mode and removal of hidden lines is contolled with toggles as described below.





(8) TOGGLES:

A variety of on/off options exist under the 'Toggles' pull-down menu. These features include Coordinate Axis, Grid Numbers, X-Y-Z Coordinates, Properties, Materials, Constraints, Force Arrows, Force Values, Pressure Arrows, Pressure Values, Moment Arrows, Moment Values, Stresses, Displacements, Measurements, Remove Hidden Lines, Shaded Elements, Announcement, etc...

When any of these toggles are switched, the corresponding quantity is included in any screen re-draw of the model.

(9) SAVING OF RESULTING MODEL INPUT:

The user has the option to save the created or modified model in either or both of two formats, one for the MSC/pal, and the other for MSC/NASTRAN. Either format is selected under the 'File' pull-down menu.

If 'Write MSC/pal Model' is selected, two files are written, one for the model stiffness, and another with the static loads data. The file name specified by the user is utilized for both files, however the stiffness file name is automatically supplemented with '.MDL', while the static loads file name is complemented with '.ST'.

If 'Write MSC/NASTRAN BULKDATA' is choosen, stiffness as well as static loads data is written in the appropriate format with user specified file name, supplemented with '.NAS'.

Printing of model drawings as well as printing of files, is also available under the 'File' pull-down menu.

If connected to a color printer, such as the HP PaintJet or similar, certain quantities such as force, pressure and constrain vectors will appear in color.

(10) CONCLUSION

The modeling features presented above are intended to offer the user a significant cost saving in the form of reduced modeling time. The program is also expected to increase the quality of the users end product, since more analysis can be performed within a given time frame.