

Copyright

**© Copyright 1987-2011
LAPCAD Engineering, Inc.**

All Rights Reserved

This micro-computer program is copyrighted and **all rights are reserved by LAPCAD Engineering, Inc.** The distribution and sale of this program is intended for use by the original owner only, and for use only on the computer system that is specified on the application CD. The program may be used only under the provisions of the license agreement that is displayed on the following page.

The manual is copyrighted and all rights are reserved. This document or any part thereof may not be copied or reproduced in any form or with any media, without the written approval of LAPCAD Engineering, Inc.

LAPCAD Engineering, Inc. reserves the right to make changes to this manual and the LapFEA computer program without notice.

While LAPCAD Engineering, Inc. has made every effort to ensure that the information in this manual is correct, LAPCAD Engineering, Inc. does not assume responsibility for any errors that may appear.

AutoCAD is a registered trademark of Autodesk, Inc. Macintosh is a registered trademark of Apple Computer. Windows is a registered trademark of Microsoft Corporation. NASTRAN is a registered trademark of the National Aeronautics and Space Administration. MSC/NASTRAN is an enhanced, proprietary version developed and supported by The MacNeal-Schwendler Corporation (MSC), now known as MSC.Software. MSC/pal 2 is a registered trademark of MSC.

License agreement

LAPCAD Engineering, Inc. Customer License Agreement

Important Notice - Please Read Carefully

The enclosed software is licensed by LAPCAD Engineering, Inc., to its customers for their use only on the terms set forth below. Use of this manual or the software indicates your acceptance of these terms. If these terms are unacceptable, this manual and software should be returned immediately to LAPCAD Engineering, Inc., 8305 Vickers Street, Suite 202, San Diego, California 92111, U.S.A.

1. LICENSE: LAPCAD Engineering, Inc. hereby grants you the non-exclusive and nontransferable license to use LapFEA on a single computer, subject to the provisions hereof.
2. COPYRIGHT: The Program and its documentation are copyrighted. YOU MAY NOT USE, COPY, MODIFY, REVERSE COMPILE, REVERSE ASSEMBLE OR TRANSFER THIS PROGRAM OR ITS DOCUMENTATION, OR ANY COPY, MERGED PORTION THEREOF, IN WHOLE OR IN PART, EXCEPT AS EXPRESSLY PROVIDED FOR HEREIN.
3. LIMITED WARRANTY FOR DISKETTES: LAPCAD Engineering, Inc. warrants the diskette on which the Program is recorded to be free of defects in material and workmanship under normal use and service, for a period of ninety (90) days from date of purchase, as documented by receipt.
4. No Warranty of Performance: LAPCAD Engineering, Inc. DOES NOT WARRANT THE PERFORMANCE OR RESULTS THAT MAY BE OBTAINED BY USING THIS PROGRAM. THE PROGRAM AND ITS DOCUMENTATION ARE LICENSED "AS IS" WITHOUT WARRANTY AS TO THEIR PERFORMANCE, MERCHANTABILITY OR FITNESS FOR ANY PARTICULAR PURPOSE. YOU ASSUME THE ENTIRE RISK AS TO THE RESULTS OR PERFORMANCE OF THIS PROGRAM. NEITHER LAPCAD Engineering, Inc. NOR ANYONE ELSE WHO HAS BEEN INVOLVED IN THE CREATION, PRODUCTION OR DELIVERY OF THIS PROGRAM SHALL BE LIABLE FOR ANY DIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES RESULTING FROM THE USE OF THIS SOFTWARE PRODUCT OR ARISING OUT OF ANY BREACH OF ANY WARRANTY.
5. TERM: This license is effective until terminated. It will terminate if you fail to comply with any term or condition of this License Agreement. You are obligated in the event of such termination to either destroy the Program and Documentation and any copies, or return this material to LAPCAD Engineering, Inc.
6. ACKNOWLEDGMENT: YOU ACKNOWLEDGE THAT YOU HAVE READ THIS AGREEMENT, THAT YOU UNDERSTAND THE AGREEMENT, AND THAT YOU AGREE TO BE BOUND BY ITS TERMS AND CONDITIONS. YOU AGREE THAT THIS AGREEMENT IS THE EXCLUSIVE AND COMPLETE STATEMENT OF AGREEMENT BETWEEN YOU AND LAPCAD Engineering, Inc., AND THAT THIS AGREEMENT SUPERSEDES ALL PRIOR AGREEMENTS OR PROPOSALS, COMMUNICATED VERBALLY OR IN WRITING.
7. MISCELLANEOUS: the laws of the State of California shall govern this license agreement.

Getting started

Dear Colleague:

Thank you for purchasing our integrated FEM/FEA package titled **LapFEA**. Let me tell you briefly about this product, as well as about the two major components it is comprised of.

LapFEA

This integrated product is a combination of the LapCAD modeler described below, and a finite element solver. This solver supports linear static analysis, modal frequency analysis, and limited frequency and transient response analysis.

This combination also provides step-wise nonlinear geometry capability.

LapCAD

LapCAD was first introduced at a conference in Pasadena, in the spring of 1988. It initially functioned as a finite element preprocessor on the Macintosh.

Since then, this product has been expanded to include post processing. Support for NASTRAN has also been added, and in 1991 the first release running on Microsoft's Windows 3 was introduced.

The product has been enhanced through the years, and currently runs on Macintosh OS9.1, Windows 2000, XP, Vista and Windows 7.

About the documentation

The documentation consists of six pdf files:

- a_LapFEA_Get_Started.pdf
- b_LapFEA_Samples.pdf
- c_LapFEA_Addendums.pdf
- d_LapFEA_Modeler_Reference.pdf
- e_LapFEA_Solver_User.pdf
- f_LapFEA_Solver_Reference.pdf

Modeler capacity

The capacity of the modeler portion of LapFEA is greater than what the solver handles. The reason for this is to provide extra elbow room during the modeling phase.

Memory requirement

Hard disk space:

The expanded application and related files require a free space of approximately forty (40) megabytes. When solving a finite element model, the required space varies from a few MB for a small model, to several hundred MB for a large model. It is recommended that a hard drive with 20 gigabyte of free space is utilized for models approaching 30,000 nodes.

Random access memory:

A minimum of 512 MB of RAM is recommended. In order to minimize swapping between real memory and the hard drive, 1GB or more of RAM is preferred.

Installation (Please read any ReadMe file for the latest installation instructions)

Your LapFEA application was shipped on a CD, or sent compressed as an e-mail attachment. After you have installed the application you need to e-mail to LAPCAD Engineering, Inc. some specifics about your hardware, which will emerge when you attempt to launch LapFEA for the first time.

LAPCAD Engineering, Inc. will e-mail back to you a file titled 'LF_00_00.dll'. Copy this file to the LapFEA folder on your hard drive. This will replace a non-functional file with the same name, that was placed there during the installation of LapFEA.

Your LapFEA application is now ready to be used.

Bug report

If you find that LapFEA or any of its components behave incorrectly or inconveniently, or if you find errors in the documentation, please send us a note, to

FAX No.: (858) 467-1947, or

e-mail: info@lapcad.com

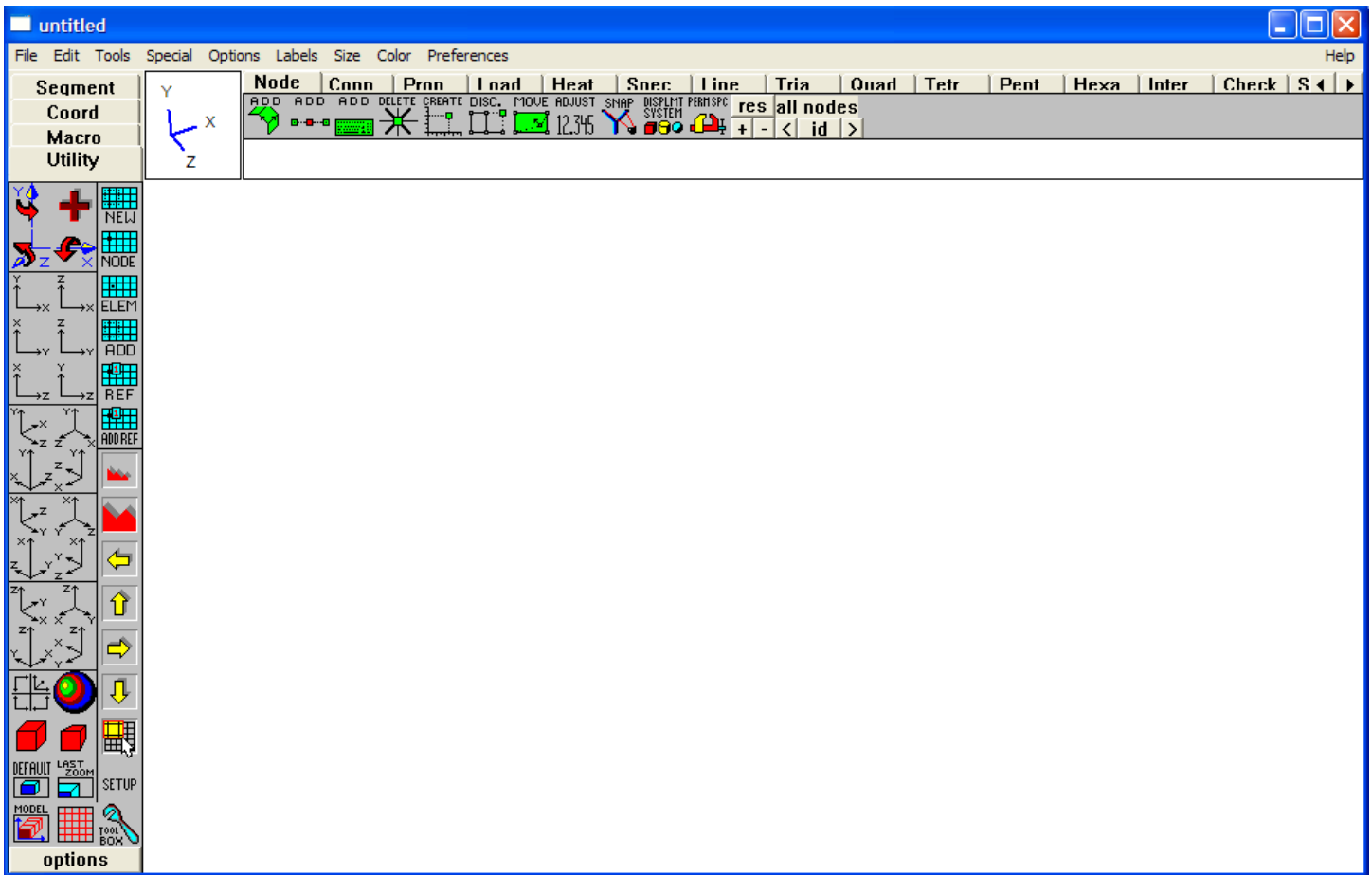
Best regards, Gert Lundgren, President, LAPCAD Engineering, Inc.

INTRODUCTION

If LapFEA is not already on your hard disk, please first go back to the Installation Section.

After having familiarized yourself with the various tool tabs shown below, please consider working through the introductory sample, starting on the next page. This is a complete analysis of a simple sixteen-noded bracket, that includes the creation of geometry, modification of the default physical and material properties, the application of loads and boundary constraints, solving of the model, and finally post-processing of the result.

For a complete review of all the tool tabs, please refer to the Slide Show.

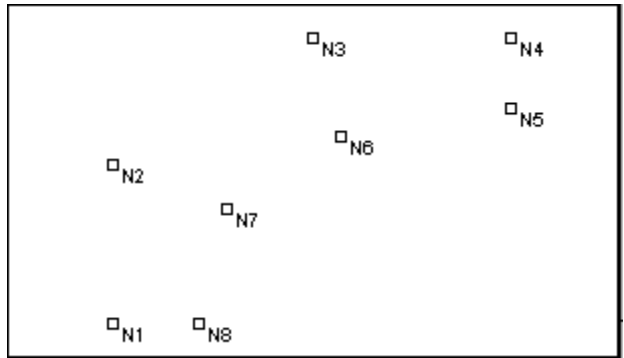


Static analysis

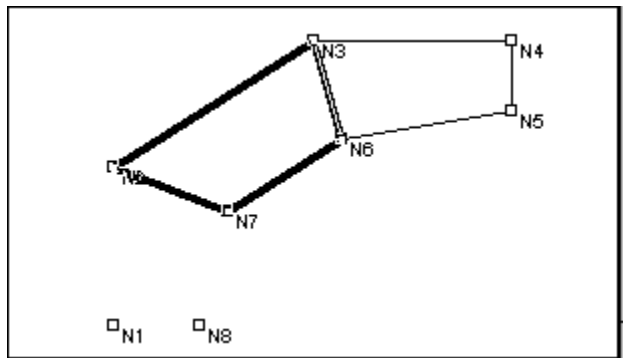
Create geometry in LapFEA

1) With the mouse cursor, select the **CREATE** icon in the Node tool tab, then digitize the following nodes: (Please note that if a node ends up in the wrong place, do not exit this mode. Instead drag it in place, or Undo and create a new one. Also, actual location is not important for this exercise)

Horizontal (X)	Vertical (Y)
0.5	0.5
0.5	2.5
3.0	4.0
5.5	4.0
5.5	3.3
3.5	3.0
2.0	2.0
1.5	0.5

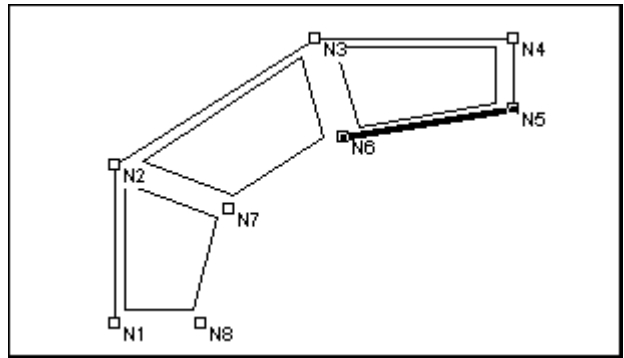


2) Turn on the Labels/Numbering/Node pull-down menu item. Then select the **QUAD** icon in the Connectivities tool tab, and connect one element by moving the cursor to within the center of four nodes such that a heavy outline appears, then click once. Do the same for the two remaining quadrilateral elements.



3) Set the Options/Displ Mode/Shrink pull down menu item, then select the **LINE** icon in the Connectivities tool tab. Move the cursor to a point half way between nodes 1 and 2. When the heavy outline appears, click once.

Connect the following pairs of nodes in the same manner: 2-3, 3-4, 4-5, 5-6, 6-7, 7-8, 8-1, 2-7, 3-6. The purpose of this step is to create line elements that can be extruded into surfaces (flanges).

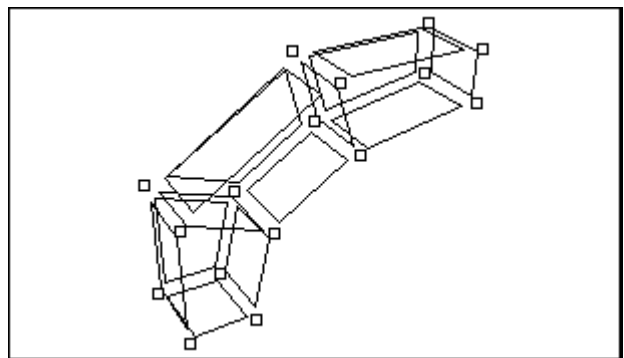


Click on the **left sixs icon from top** of the Utility tool tab, in order to make the following steps more visible in the 3-D space.

4) Select All, either via Ctrl/All, or use the Edit pull down menu.

Select the **EXTRUDE** icon in the Segment tool tab, then the Extrude Surface >>, then Z-Surface. This should add flanges as shown in the picture to the right.

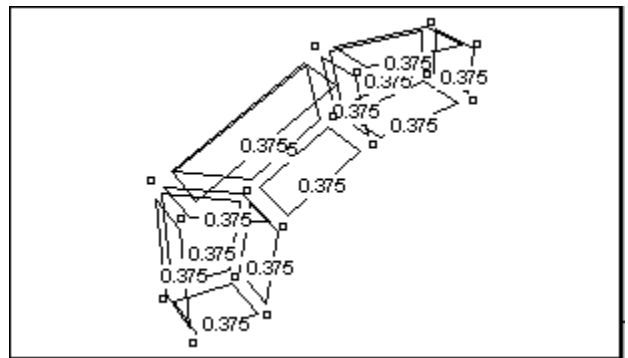
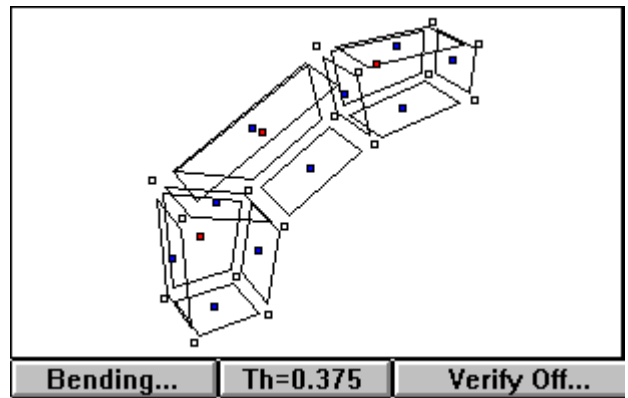
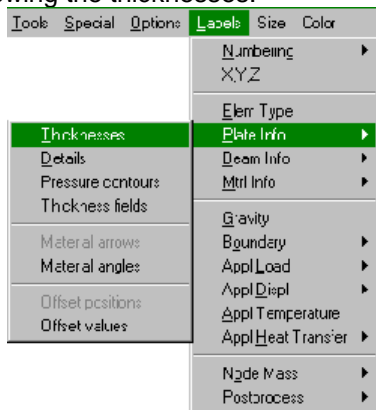
Hit the **DEFAULT** icon in the Utility tool tab. Rotate the model around the Y-axis, by repeatedly clicking the **upper-left** icon in the Utility tool tab. Return to the initial view by again clicking on the **left third icon from bottom** of the Utility tool tab.



Add properties in LapFEA

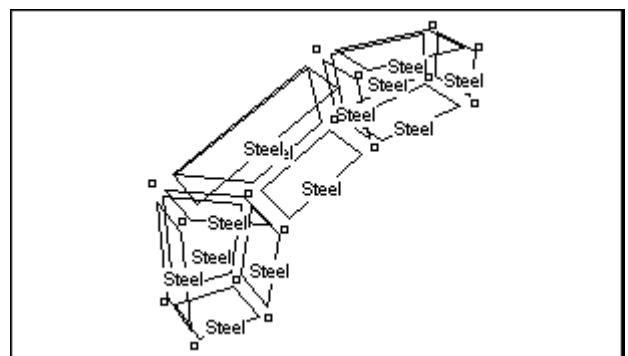
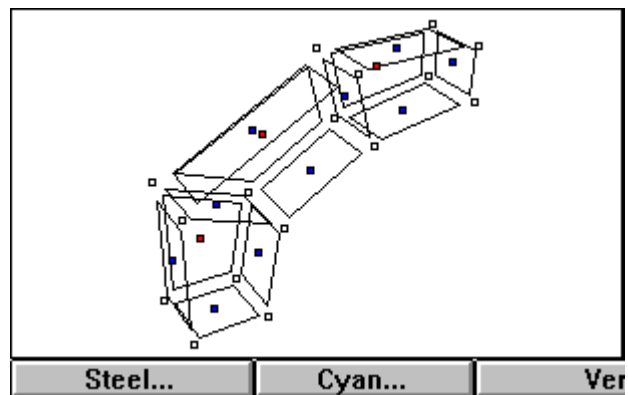
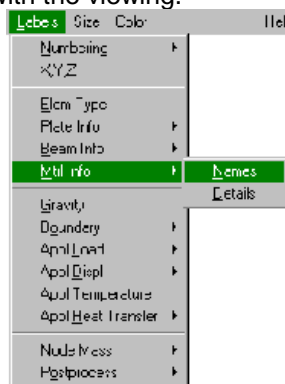
1) Click on the **PLATE** icon in the Properties tool tab (the first plate icon from left). Set the thickness (Th) to 0.375 using third button from left in the emerging lower sub menu. Click on the Model button in order to implement the thickness on the entire model.

2) Set the Labels/Plate Info/Thickn pull down menu item, in order to display the thicknesses. De-select the same item, when done viewing the thicknesses.



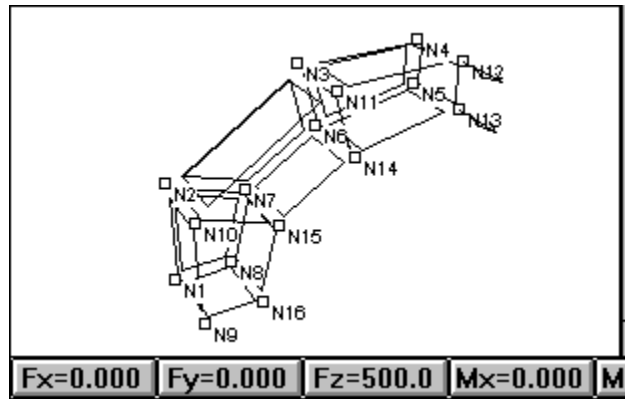
3) Click on the **MATERIAL** icon, also in the Properties tool tab. Toggle the up and down button in the lower emerging sub menu until it reads 'stl', then click on the button labeled Model, thereby changing the default aluminum material to steel for the entire model.

4) Set the Labels/Mtrl Info/Names pull down menu item, in order to display the element materials. De-select the same item, when done with the viewing.

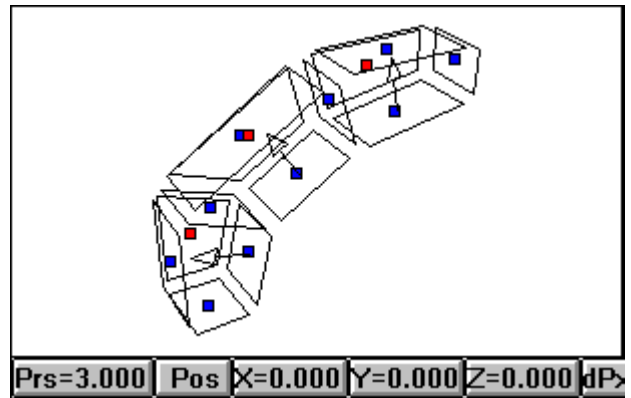


Apply Loads in LapFEA

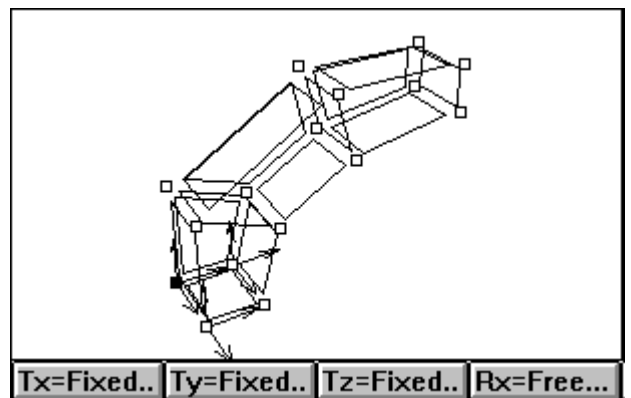
1) In order to see the node numbers, first select the Labels/Numbering/Node numbers pulldown menu item. Then click on the **FORCE** icon in the **Loads** tool tab, then Concentrated Forces >>. Set the force in the Z-direction to 500, using the third button from left in the emerging lower sub menu. Click on nodes 12 and 13 in order to implement the force on the model. You can turn on the node numbers again, if desired.



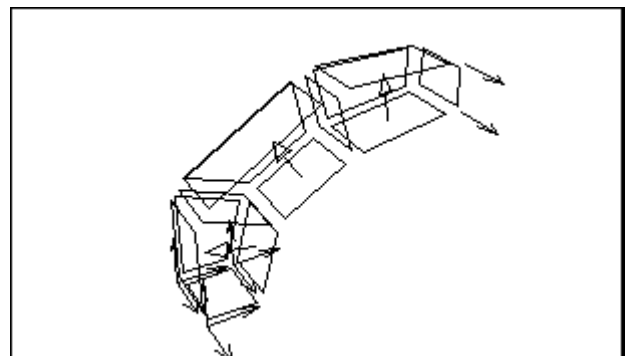
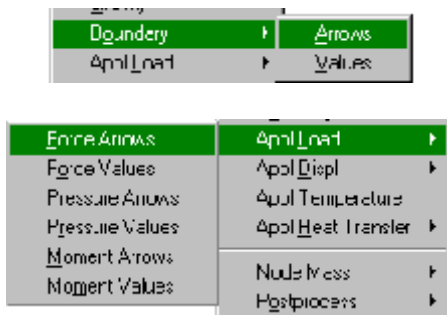
2) Click on the **PRESSURE** icon in the **Loads** tool tab. Set the pressure to 3.0 using the first button from left in the emerging lower sub menu. Click on a couple of elements by aiming at the small square in the center of each element.



3) Click on the **FIXITY** icon in the **Loads** tool tab. Toggle each of the three buttons from left in the emerging lower sub menu, until they read 'Fixed'. Click on three nodes at the base of the model, in order to attach it to the ground.



4) In the Labels pulldown menu, set the Boundary/Arrows, Appl Load/Force Arrow and Appl Load/Press Arrow items, in order to view the applied environment. Save the model in the LapFEA type using the File pull down menu.



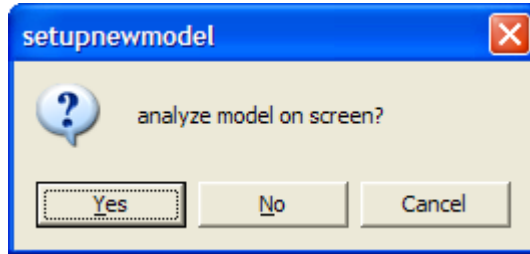
Solve using LapFEA

This is the simplest part, since it requires only clicking one icon.

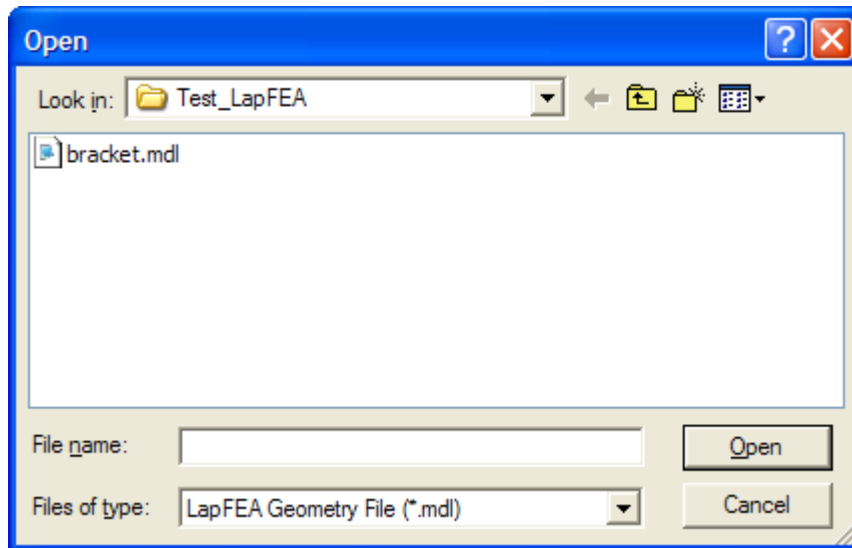
Click on the icon to the far left in the Solver tool tab.



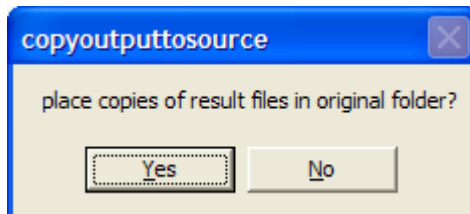
The following dialog box will emerge:



Select the NO button. Then Open dialog box emerges:



Select your model with the .mdl file extension and click the Open button. The same dialog will appear again, this time expecting you to select your loads file with the .st file extension. The model input and subsequent messages will stream by in the temporary DOS type of window. If the execution was successful, the following dialog box will appear:

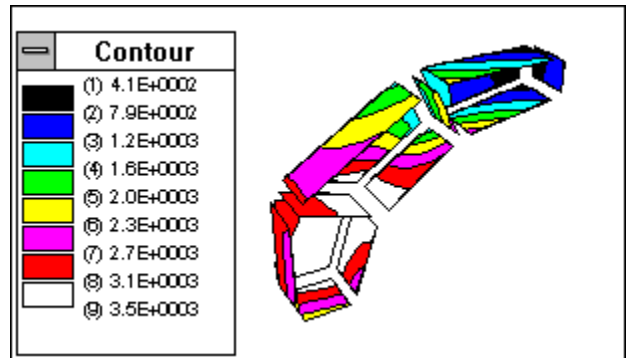


Select the YES button, and the solver portion of the analysis is completed.

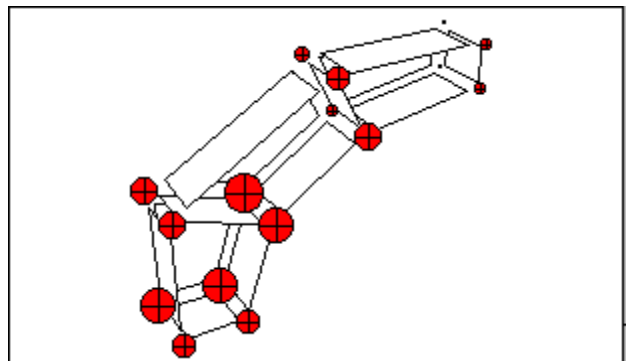
Post-process in LapFEA

1) Open the model geometry file, using the File pull down menu, with the LapFEA(*.mdl) file type. (*The loads file opens automatically, if the file name, less the file extension, is the same as the model geometry file.*) Next, open the solution or result file, using the LapFEA(*.res*) file type. Note that the steps described above are performed automatically, if a successful solver session, as described on the previous page, precedes the post processing session.

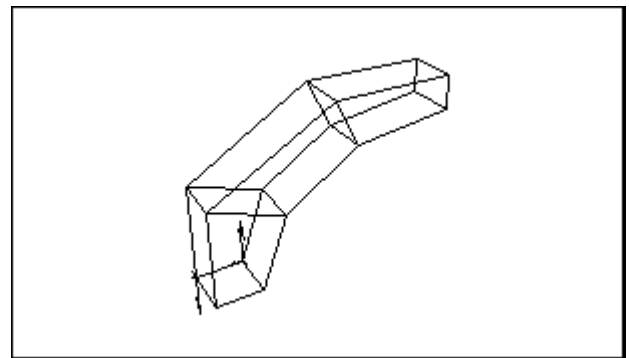
Now click on the upper left icon labeled **SHADE** in the emerging Post Processing window.



2) Click on the **BULLETS** icon. This will display the nodal stresses in a bullet format, where the diameter of the bullets indicate the relative magnitude.

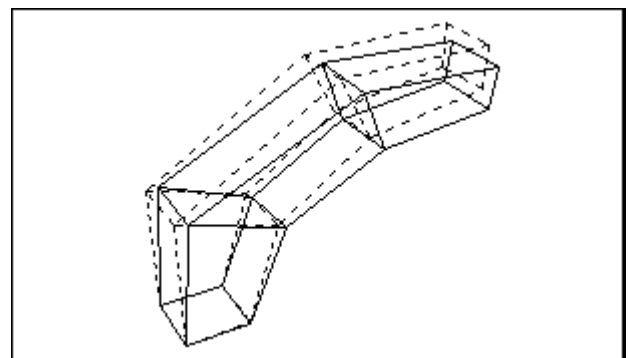


3) Select the Labels-Postproc-Force Reaction Arrows pull-down menu item.



4) Click on the **ANIMATE** icon in order to animate the deformed shape.

Exit LapFEA, when completed.



Modal analysis



Modal analysis does not require a specific change to the loads file, however attention should be paid to using the desired boundary constraints.

Solve the modes by clicking on the 'MODES' icon in the Solver tool tab. When prompted to select a loads file, select the previously created one. As the analysis progresses, and the program reads the static loads file, error messages will appear when entities such as force and pressure loadings are evaluated. This can be ignored, since those quantities are not used in the modal analysis.

If the model size is less than 25 nodes, the direct technique is used. If the model is larger, the subspace iteration method will be utilized. In this case, the model size limitation is the same as for static analysis.



When the Post pallet emerges, click on the Animation icon in that pallet. This will force the post processor to survey the number of modes, and enter these as subcases in the Loads tool tab. Other modes can be viewed by advancing the subcase number in the Loads tool tab with the < and > buttons. The subcase number as well as the current frequency can be viewed in the Message line under the horizontal row of tool tabs.

Node	Conn	Pron	Load	Heat	Spec	Line	Tria	Quad	Tetr	Pent	Hexa	Inter	Check	Solve
FORCE	LINLOAD	PRESSURE	FIXITY	DISPMT	TEMP	GRAVITY	TRANS	FREQU	6	#				
									1	1			add	new
													del	res
														< >
MODE NO. 1 AT 4.65041E+02 CPS														

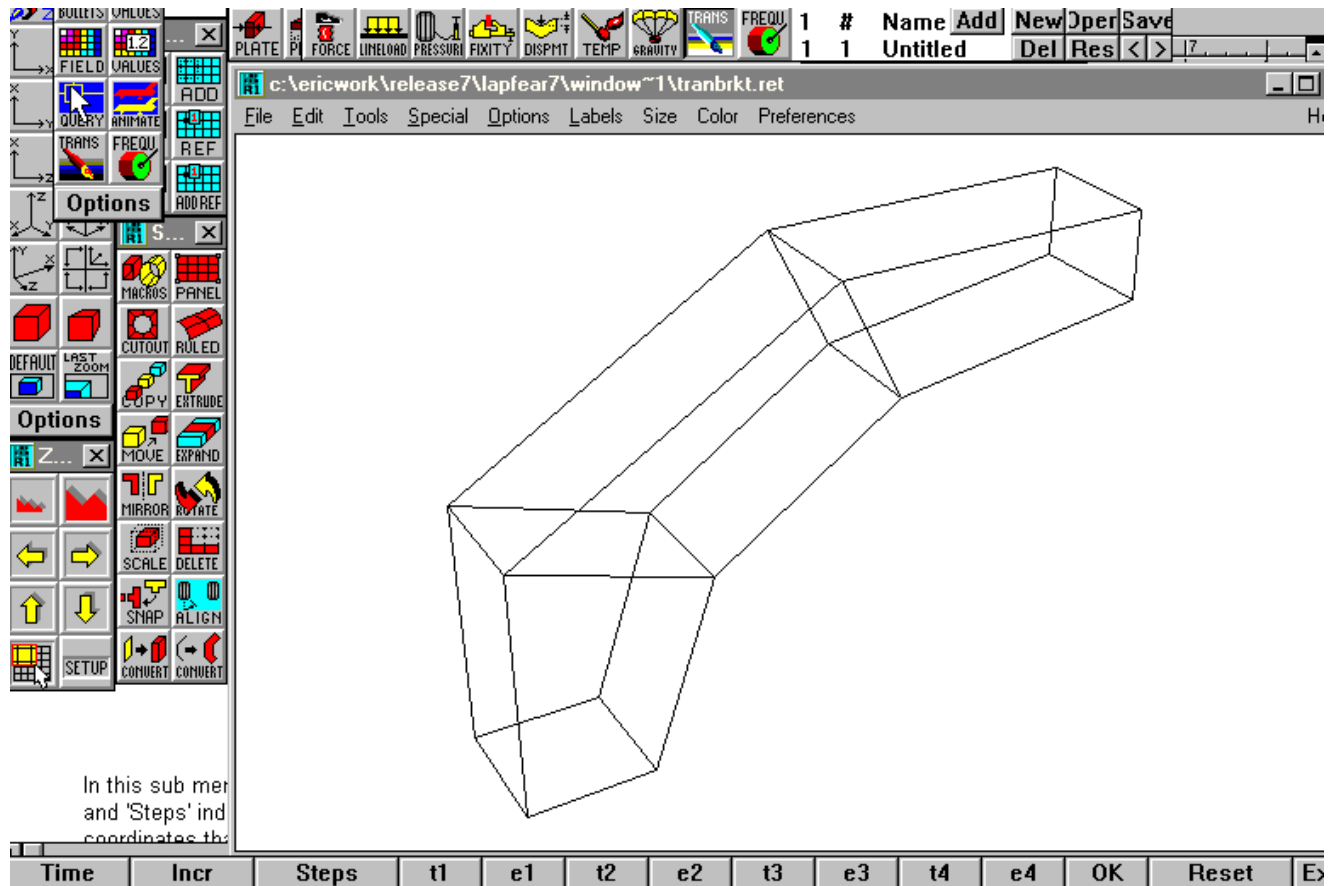
Note that the entire solution data is contained in a file titled 'your model name.rem'. The content can be viewed with a text editor, and data may also be imported to a spreadsheet program for further dissemination.

Compute transient response

1. **Preprocess**- Click once on the 'TRANS' icon in the 'Loads' pallet. Note: There are three 'TRANS' and 'FREQ' icons in LapFEA. For this step you must use the 'TRANS' in the 'LOADS' pallet shown below:
- 2.



Click once on the 'TRANS' icon to bring up the sub menu shown below:



In this sub menu shown above in the screen capture, 'Time' sets the time range, 'Incr' specifies the output interval, and 'Steps' indicates the number of solution times per output time period. 't1' and 'e1', 't2' and 'e2' etc. sets the coordinates that define the percent load level versus time.

Set Time=0.1, Incr=0.01, Steps=1, t1=0.000, e1=0.0, t2=0.001, e2=1.0, t3=0.100, and e3=1.0. Do not enter any values for t4, and e4.

The underlined lines in the loads file shown below reflect these settings:

```
TITLE brkttran.ST
C *****
C lapcadsubcase      1 X-LOADING
C
DISPLACEMENTS APPLIED
TX   1 0.00e+0
TY   1 0.00e+0
TZ   1 0.00e+0
TX   8 0.00e+0
TY   8 0.00e+0
TZ   8 0.00e+0
TX   9 0.00e+0
TY   9 0.00e+0
TZ   9 0.00e+0

FORCES AND MOMENTS APPLIED
FX   4 100.000
FX   5 100.000
FX  12 100.000
FX  13 100.000

TIME RANGE 0.1 0.01 1
EXCITATION DEFINITION
0.000 0.0 0.001 1.0 0.100 1.0

SOLVE
QUIT
```

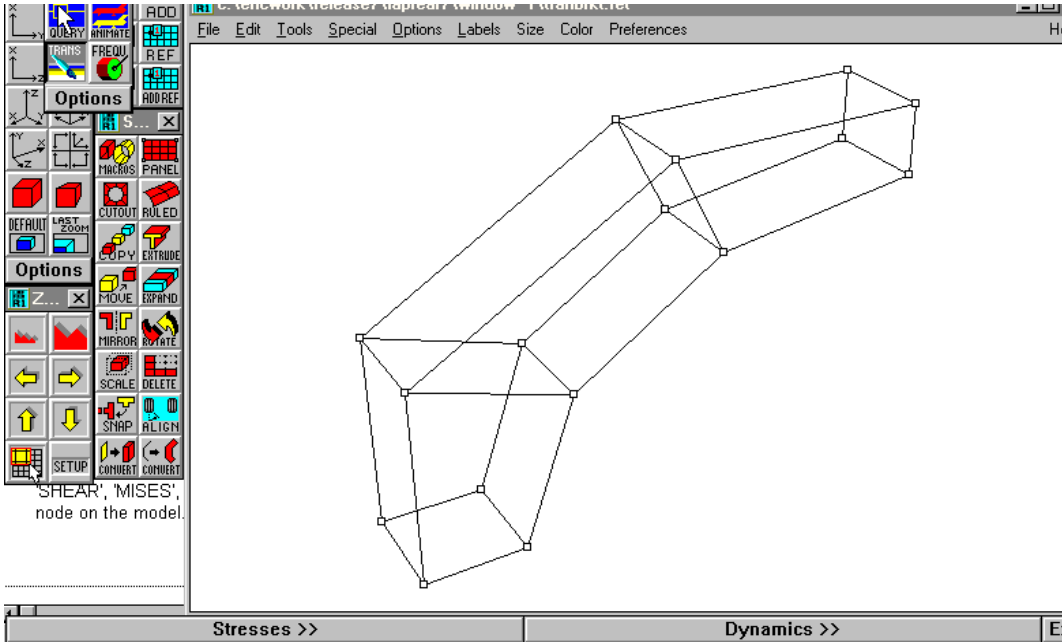
The TIME RANGE specifies that the final time is 0.1 seconds, output will occur at every 0.01 seconds, and that there is going to be one solution time per output time period. The EXCITATION DEFINITION specifies that there is no initial loads, and that the load reaches 100% at 0.001 seconds, and remains constant to 0.100 seconds.

2. Solve the model-

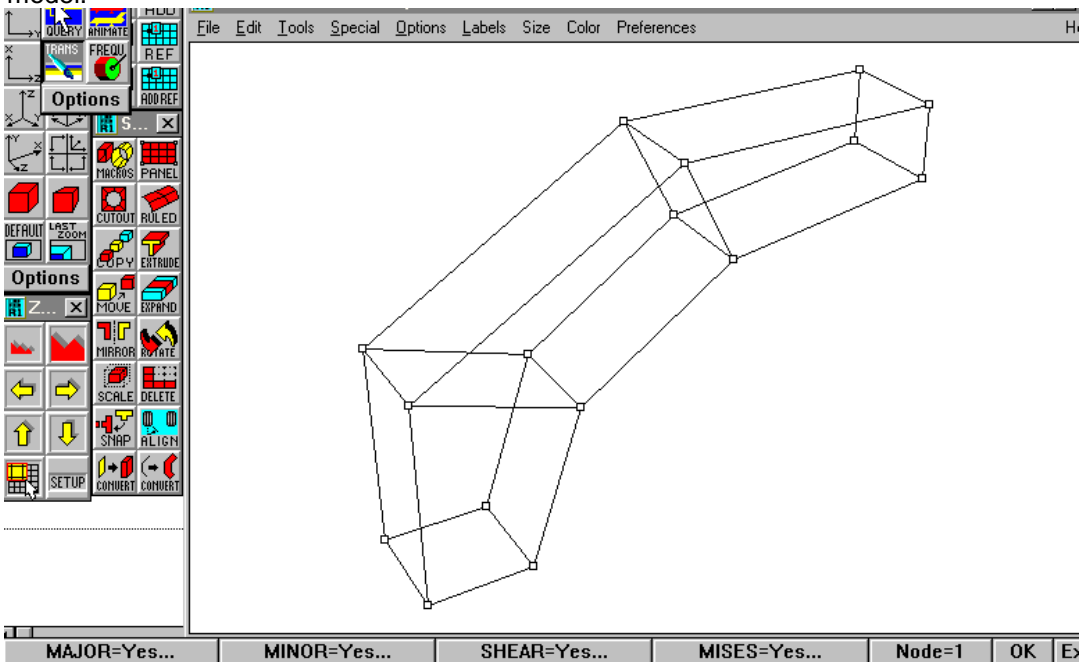
Solve the transient analysis by clicking on the 'TRANS' icon in the 'Solver' pallet. When prompted to select a loads file, remember to select the one that contains the statements specific to this analysis, as discussed above.

3. Perform Transient Post Processing-

When the Post Processing pallet emerges, click on the 'TRANS' icon and you will be prompted with a sub-menu containing 'Stresses' and 'Dynamics' shown in the screen capture below. This sample will first guide you through the 'Stresses' found on the left bottom of the screen.

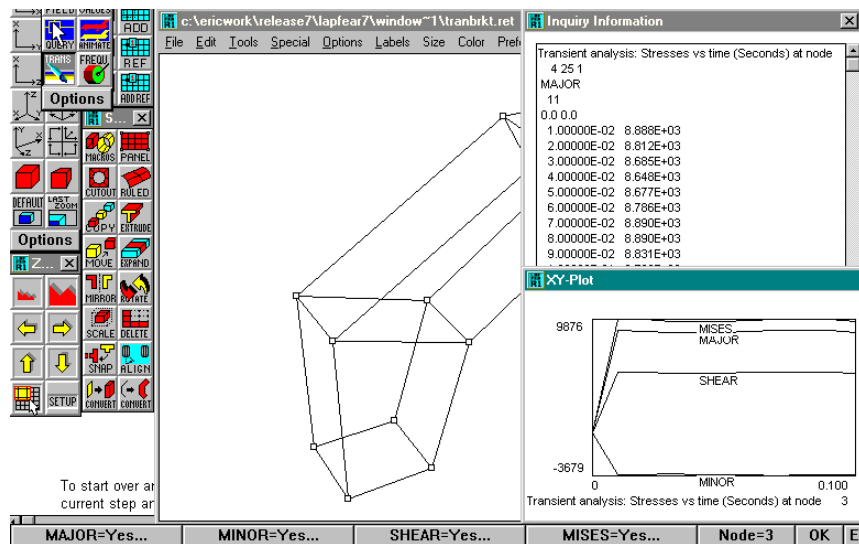


Click once on the 'Stresses' sub menu button to bring up the sub menu shown below. Toggle on/off 'MAJOR', 'MINOR', 'SHEAR', 'MISES', enter the Node number you want to see and click on the 'OK' button. You can also click directly on a node on the model.

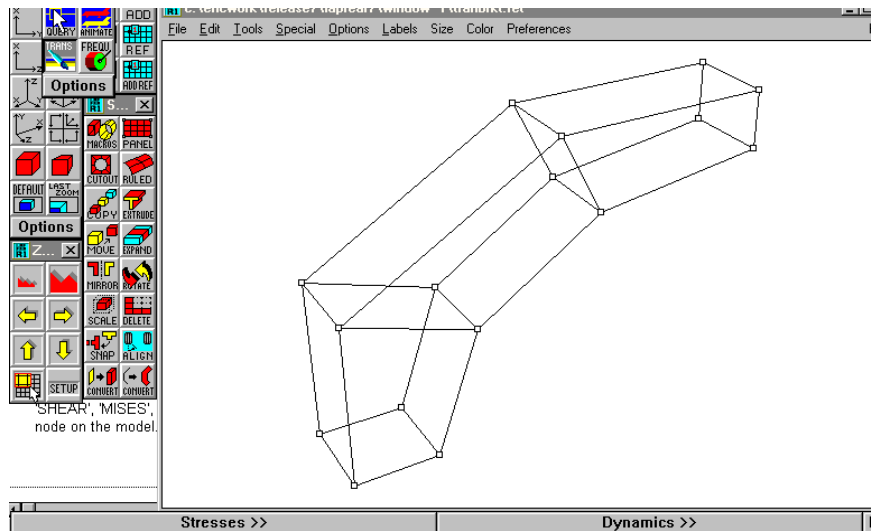


Having clicked on a node, the 'Inquiry Information' and 'XY-Plot' windows will appear, as shown below. If you would like to change the toggle preferences (Major, Minor, Shear, Mises...) but keep the same node, just toggle the items to your liking, and click on the 'OK' button in the same sub-menu. The windows will be updated to the new settings. The windows displaying 'Inquiry Information' and 'XY-Plot' can be captured and imported to a word processor by pressing the F9 button on your keyboard, bringing up the hand. Simply drag a square encompassing the item you want to capture. The 'Inquiry Information' and 'XY-Plot' windows can be selected and moved to make viewing the model easier.

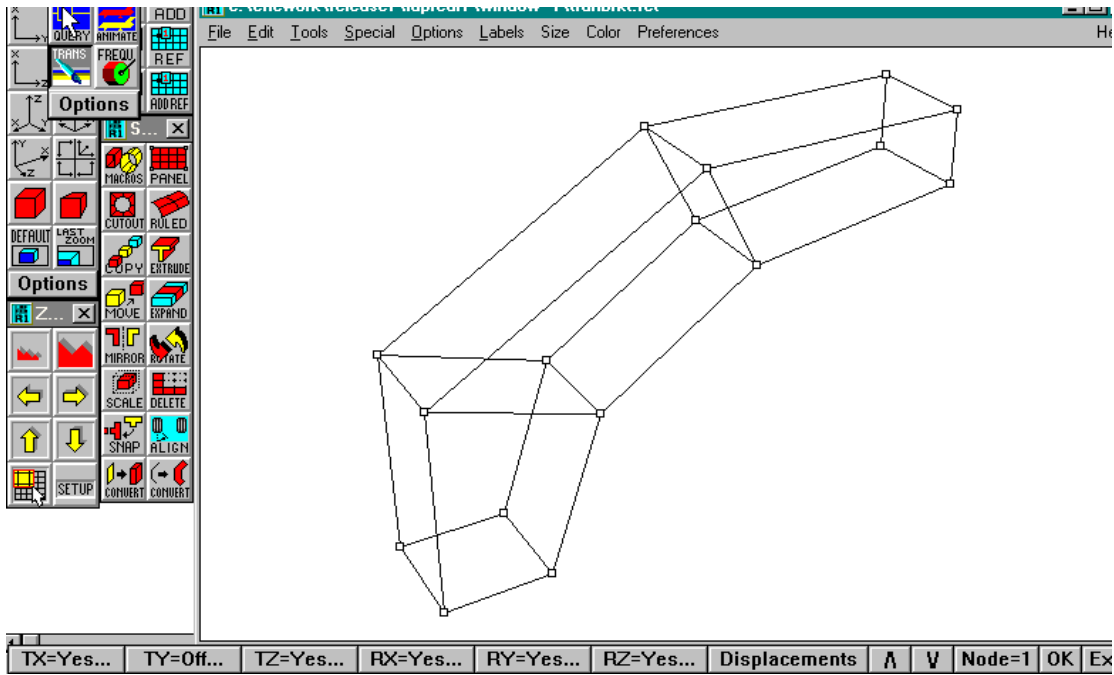
Also, text in the 'Inquiry Information' window can be copied and pasted as editable text into a wordprocessor.



To start over and do 'Dynamics' just click once on the 'TRANS' icon in the Post Processing pallet. This will exit the current step and bring you back to the 'Stresses' or 'Dynamics' sub menu shown in the screen capture below. This time select 'Dynamics'.

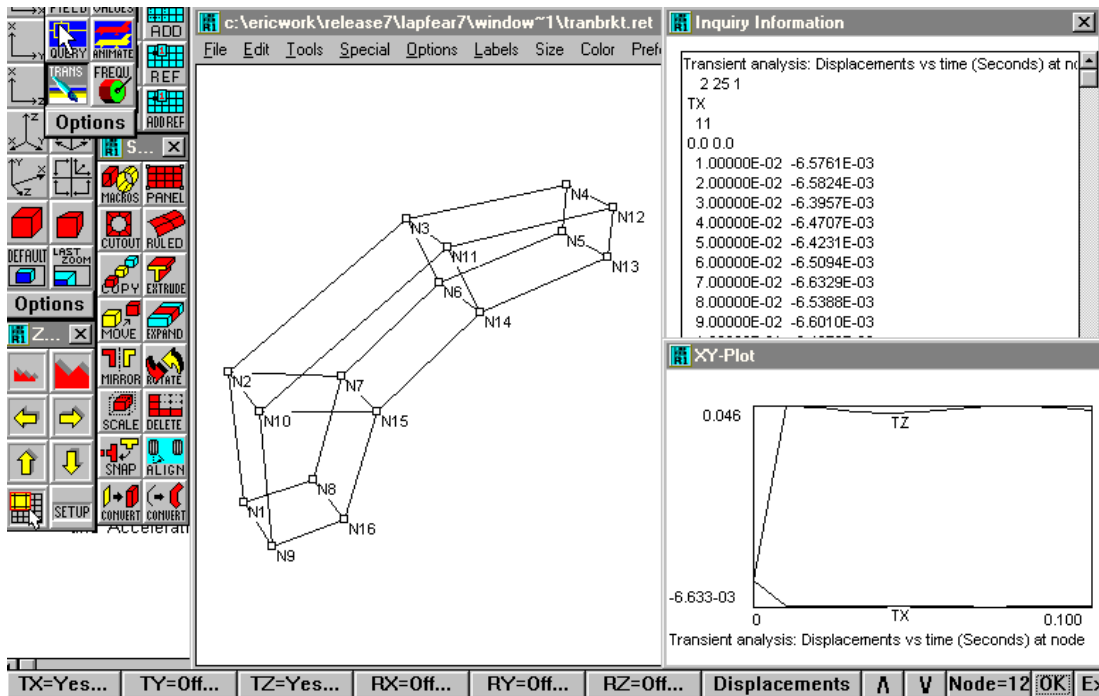


You will be presented with a different sub menu shown in the screen capture below. By clicking the up and down arrows **Displacements** \updownarrow in this sub menu you will toggle between 'Displacements', 'Accelerations', and 'Velocities'.

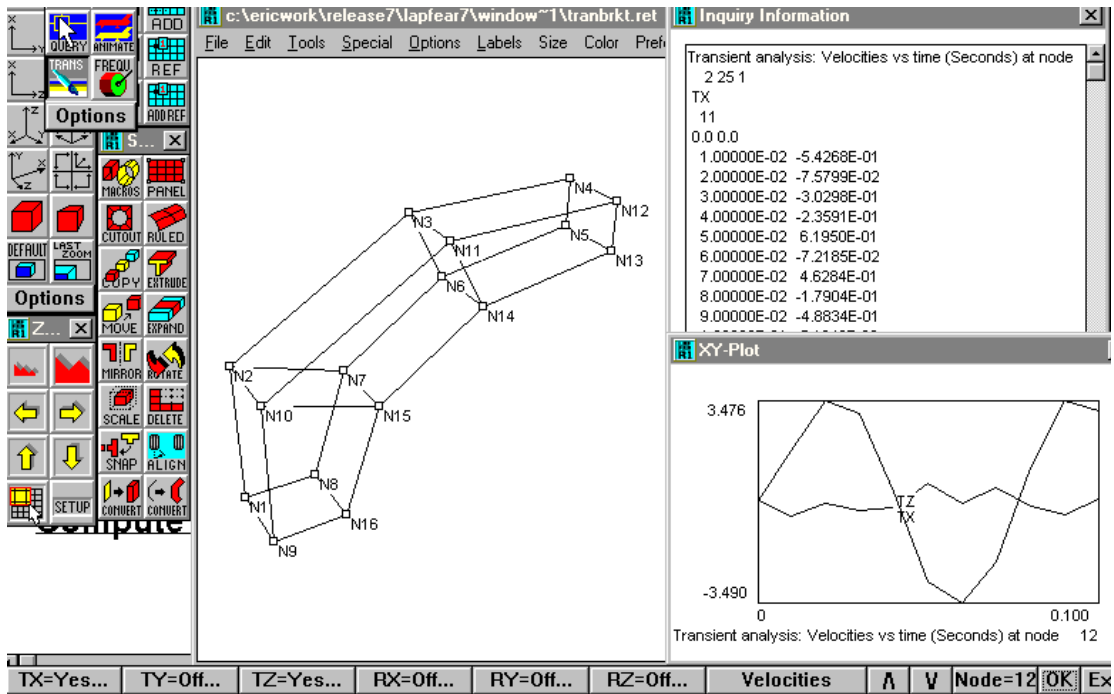


In the example, we will leave 'TX', 'TY', and 'TZ' on, and Toggle off 'RX', 'RY', and 'RZ'. The arrows will be used to toggle the setting to 'Displacements'. Then 'Node#' will be set to 12. Finish by clicking on 'OK' to bring up the 'Inquiry Information' and 'XY-Plot' windows. Shown below are displayed the output for each of the 'Displacement', 'Velocity', and 'Acceleration'.

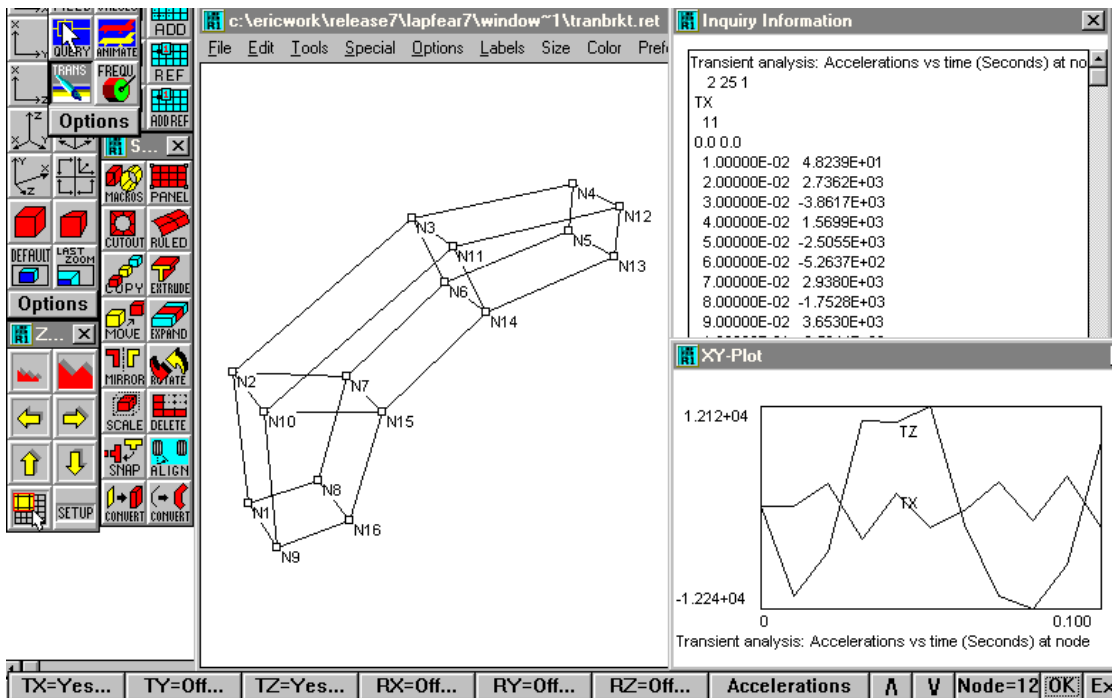
'Displacements'



'Velocities'



'Accelerations'

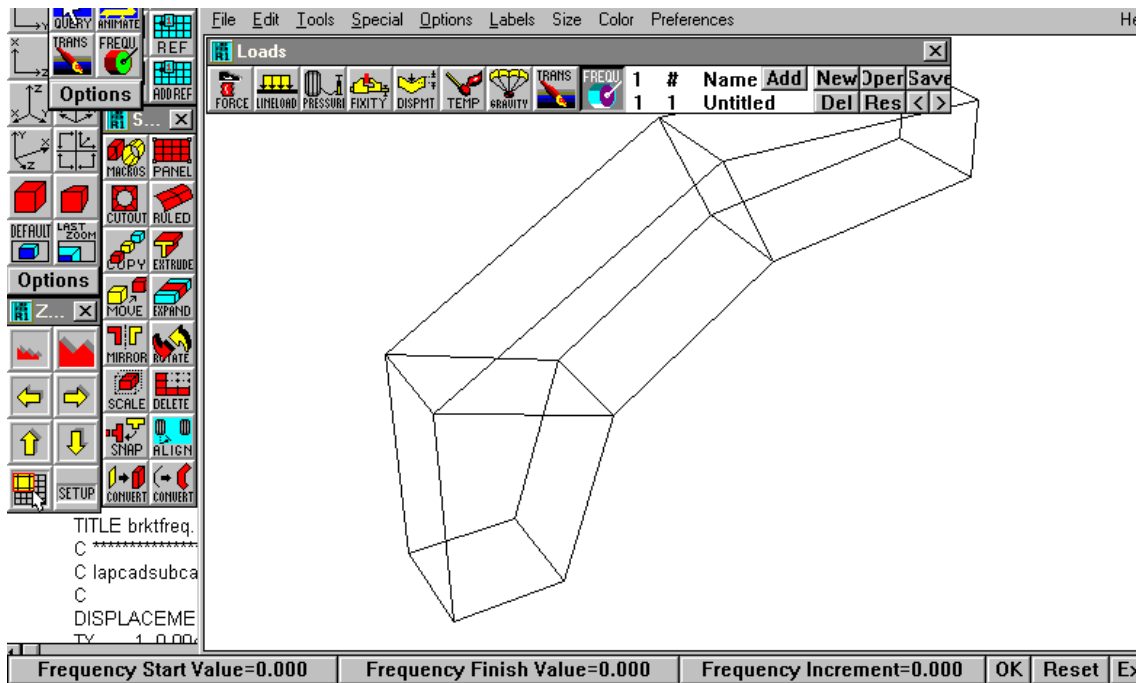


Compute frequency response

- 1. Preprocess-** Click once on the 'FREQU' icon in the 'Loads' pallet. Note: Again, there exist three different 'FREQU' icons in LapFEA. Verify that you are using the one in the 'Loads' pallet, as shown below.



Once you have clicked on the 'FREQU' icon in the 'Loads' pallet, LapFEA will bring up the sub menu shown in the screen capture below.



Using the buttons in the sub menu above, input frequency start value=0, frequency finish value=1000, and frequency increment value=50 cps.

The underlined lines in the loads file shown below reflect these settings:

```
TITLE brktfreq.ST
C *****
C lapcadsubcase      1 X-LOADING
C
DISPLACEMENTS APPLIED
TX   1  0.00e+0
TY   1  0.00e+0
TZ   1  0.00e+0
TX   8  0.00e+0
TY   8  0.00e+0
TZ   8  0.00e+0
TX   9  0.00e+0
TY   9  0.00e+0
TZ   9  0.00e+0

FORCES AND MOMENTS APPLIED
FX   4  100.000
FX   5  100.000
FX  12  100.000
FX  13  100.000

RANGE OF FREQUENCY APPLIED 0 1000 50

SOLVE
QUIT
```

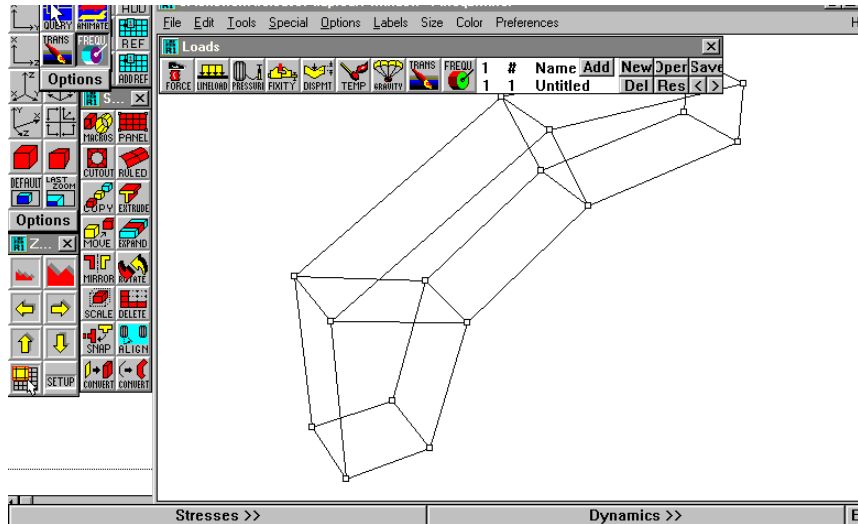
The analysis will cover a range from zero to 1000 cps, in 50 cps increments.

2. Solve the model-

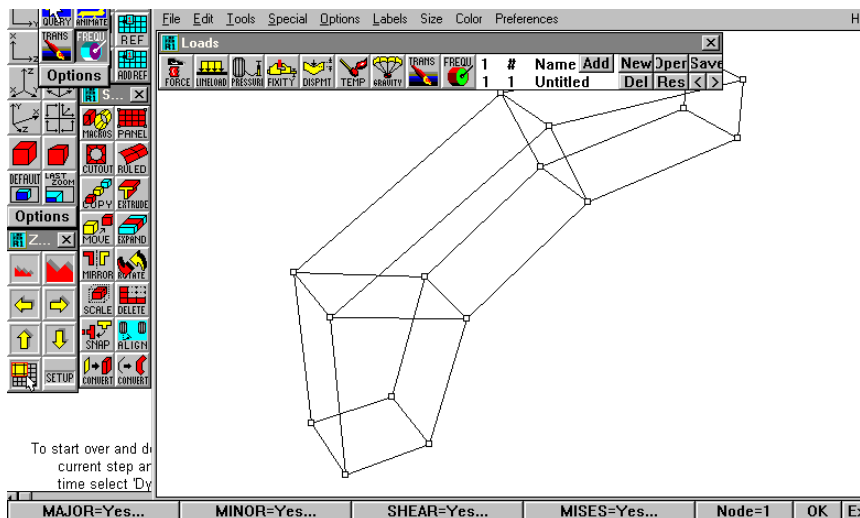
Solve the frequency analysis by clicking on the 'FRQ RESP' icon in the Solver pallet. When prompted to select a loads file, remember to select the one that contains the statements specific to this analysis, as discussed above.

3. Perform Frequency Post Processing-

When the Post pallet emerges, click on the 'FREQU' icon. You will be prompted with a sub menu containing 'Stresses' and 'Dynamics' as shown in the screen capture below. 'Stresses' will be performed first.

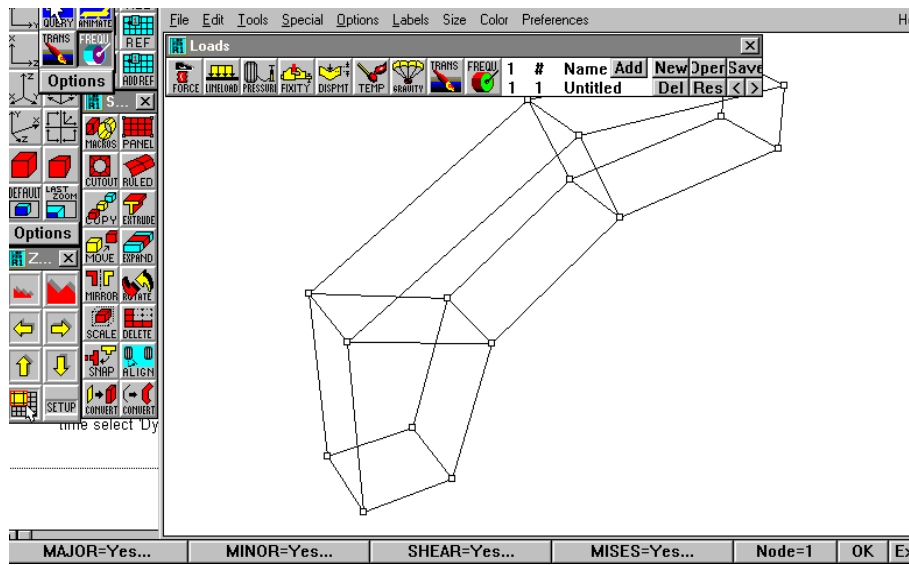


Click once on the 'Stresses' sub menu button to bring up the sub menu shown below. Toggle on/off 'MAJOR', 'MINOR', 'SHEAR', 'MISES', enter the Node number you want to see and click on the 'OK' button. You can also click directly on a node on the model.

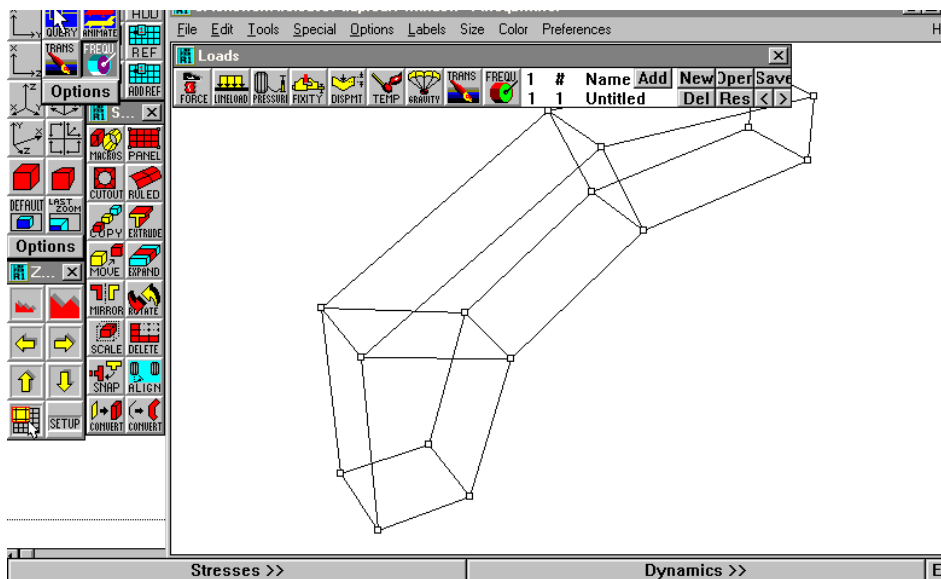


Having clicked on a node, the 'Inquiry Information' and 'XY-Plot' windows will appear, as shown below. If you would like to change the toggle preferences (Major, Minor, Shear, Mises...) but keep the same node, just toggle the items to your liking, and click on the 'OK' button in the same sub-menu. The windows displaying 'Inquiry Information' and 'XY-Plot' can be captured and imported to a word processor by pressing the F9 button on your keyboard, bringing up the hand. Simply drag a square encompassing the item you want to capture. The 'Inquiry Information' and 'XY-Plot' windows can be selected and moved to make viewing the model easier.

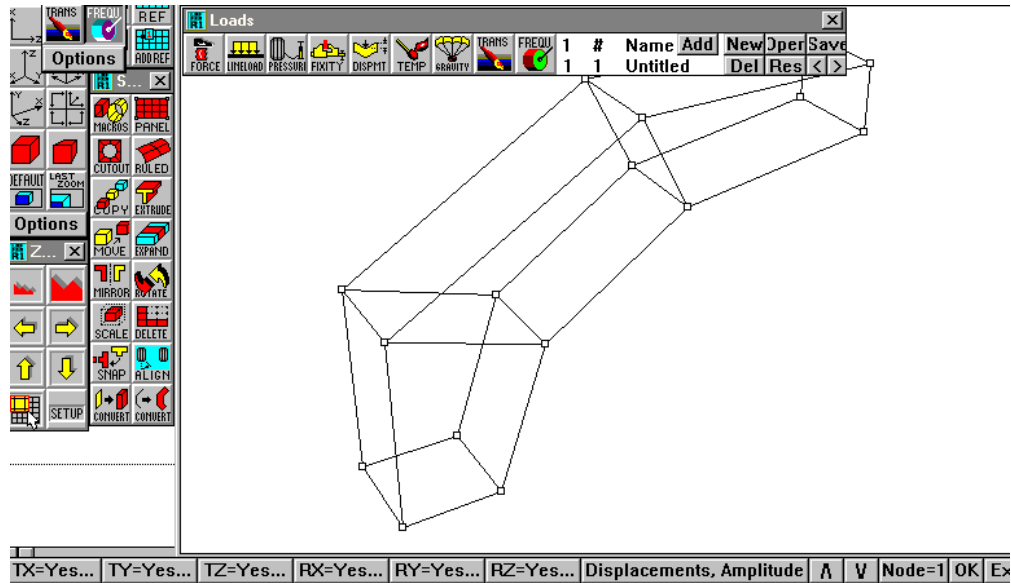
Also, text in the 'Inquiry Information' window can be copied and pasted as editable text into a wordprocessor.



To start over and do 'Dynamics' just click once on the 'FREQU' icon in the Post Processing pallet. This will exit the current step and bring you back to the 'Stresses' or 'Dynamics' sub menu shown in the screen capture below. This time select 'Dynamics'.

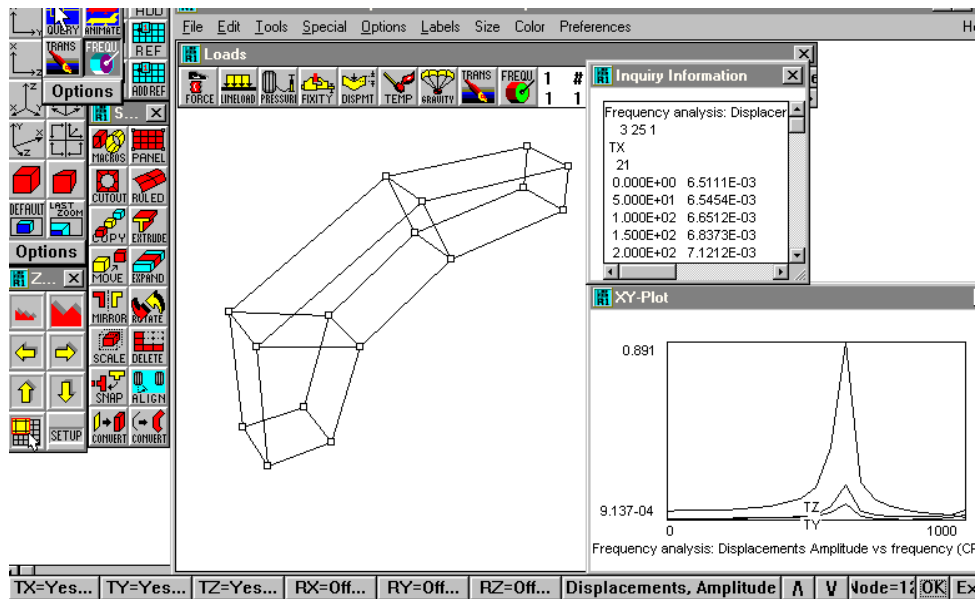


You will be presented with a different sub menu shown in the screen capture below. By clicking the up and down arrows **Displacements, Amplitude** **A** **V** you will be able to toggle between 'Displacements, Amplitude' and 'Displacements, Phase'.



For this example, we will leave 'TX', 'TY', and 'TZ' on. Toggle off 'RX', 'RY', and 'RZ'. Use the arrow buttons in the sub menu to set the response to 'Displacements, Amplitude'. Set the node# to 12. Shown below and continued on the next page are the output for both 'Displacements, Amplitude' and 'Displacements, Phase'.

'Displacements, Amplitude'



'Displacements, Phase'

