# ANSYS EXERCISE Modal Analysis of a Spring-Mass System

In this exercise, you will solve for the natural frequencies and mode shapes of the 2-DOF system shown below. Step-by-step instructions are provided beginning on the following page.

<u>Notes</u>: ANSYS has an option that will allow for the masses to be modeled as point masses. Using this option, no dimensions or material properties would be necessary for the masses. However, so that the mode shapes are easier to understand when they are animated, in this case, the masses will be modeled as blocks, 1 m x 1 m, with unit thickness. So, they have a volume of  $1 \text{ m}^3$ . The densities will be specified to produce the correct masses. Also, the free lengths of the springs are irrelevant in this analysis, as only the stiffnesses matter. But, they will both be assumed to have a length of 5 m.



The steps that will be followed, after launching ANSYS, are:

### **Preprocessing:**

- 1. Change jobname.
- 2. Define element types (#1 Solid42 element; #2 Combin14 element).
- 3. Specify material properties for the masses.
- 4. Specify real constants for the springs. (Spring stiffnesses.)
- 5. Create rectangles (actually, "squares")
- 6. Specify mesh controls for the masses.
- 7. Mesh the areas.
- 8. Create a node at the wall.
- 9. Create the spring elements.
- 10. Apply constraints.
- 11. Make rigid body constraints for each mass.

#### Solution:

12. Specify analysis type and options.

13. Solve.

## **Postprocessing:**

- 14. List the natural frequencies.
- 15. Animate the mode shapes.
- 16. List the mode shapes.

#### Exit

17. Exit the ANSYS program, saving all data.

#### Launching ANSYS:

It is probably best to save your work on a Zip Disk in the computer IOMEGA Zip Drive. On any machine in the computer lab (on the first floor of Crounse Hall), insert a Zip Disk in the drive. Then, simply click on the ANSYS icon on the Windows Desktop. The ANSYS Launcher menu should appear. It is shown at the top of the next page. The only input you will likely need on this menu is specification of the Zip Drive as your "Working Directory". If you don't have a Zip Disk, or if you prefer to work on the computer hard drive instead, then specify any directory (or, "Folder") as your working directory. To browse and find the desired working directory, click on the button with the three dots to the far right on the Launcher Menu on the line that says "Working Directory". Once the working directory is specified, click on "Run" at the bottom of the Launcher Menu.

The ANSYS menus should open up. You will see a Main Menu, illustrated on the following page, and a large black graphics window. You are now ready to begin creating the model and performing the analysis.

ANSYS Launcher Menu:

MInteractive 5.6		_ 🗆 X
Product selection	ANSYS/Faculty Research	•
Working directory	C:N	
Graphics device name	win32 💌	
Initial jobname	file	
Memory requested (megabyte	\$]	
for Total Workspace	64	
for Database	32	
Read START.ANS file at star	t-up? Yes 💌	
GUI configuration		
Parameters to be defined (-par1 val1 -par2 val2)		
Language Selection	[english]	
Execute a customized ANSYS	6 executable	
Run Close	Reset Cancel	About

ANSYS Main Menu:

📣 ANSYS Main Menu	×
Preferences .	• •
Preprocessor	>
Solution	>
<b>General Postproc</b>	>
TimeHist Postpro	>
Topological Opt	>
Design Opt	>
Radiation	>
Run-Time Stats	>
Session Editor .	
Finish	

**Note:** Most of the required tasks are performed using menu picks from the ANSYS Graphical User Interface, as specified in italics in the step-by-step instructions below. It is sometimes more

convenient, however, to enter certain commands directly at the command line. The command line is seen on the screen as:



The method of direct command line entry is used in some cases in this exercise, whenever this method is more convenient than using menu picks.

Often, as an alternative, an input file, known as a "batch file", is created, which is simply an ASCII text file containing a string of ANSYS commands in the appropriate order. ANSYS can read in this file as if it were a program, and perform the analysis in "batch mode", without ever opening up the Graphical User Interface. The batch file option is not covered in this exercise.

# \*\*\*\*IMPORTANT\*\*\*: AS YOU WORK THROUGH THIS EXERCISE, WITHIN ANSYS, ON THE ANSYS TOOLBAR (UPPER RIGHT), CLICK ON "SAVE\_DB" OFTEN!!! THIS TOOLBAR APPEARS AS:



At any point, if you want to resume from the previous time the model was saved, simply click on "RESUM\_DB" on this same Toolbar. Any information entered since the last save will be lost, but this is a nice feature in the event that you make an input mistake, and are unsure of how to correct it.

There are a number of ways to model a system and perform an analysis in ANSYS. The steps below present only one method.

# **Preprocessing:**

- 1. Change jobname. On the Utility Menu across the very top of the screen, select: *File -> Change Jobname* Enter "modalsys1", and click on "OK".
- **2. Define element types:** *Preprocessor -> Element Type -> Add/Edit/Delete* 
  - Click on "Add". The "Library of Element Types" box appears, as shown. Scroll down to highlight "Solid", and "Quad 4node 42" as shown. Click on "APPLY".



Now, scroll down to highlight "Combination" and "Spring-Damper 14" as shown, then click on "OK" and "Close".

👯 Library of Element Types			×
Library of Element Types	Boyce Visco Solid Contact Combination Thermal Mass Link Solid Shell ANSYS Fluid	Spring-damper 14 Nonlin spring 39 Combination 40 Revolute joint 7 Control elem 37 Pre-tension 179 Spring-damper 14	
Element type reference number OK	2 Apply C	Cancel	Нејр

#### **3.** Specify material properties for the masses:

Preprocessor -> Material Props -> -Constant- Isotropic

On the box, shown on the next page, enter "1" in the "EX" input field (EX is the modulus of elasticity), and enter "2" in the "DENS" input field, as shown, then click on "APPLY". Note that an arbitrary value was entered for EX, because we will couple all DOF on each

mass so that they move as rigid bodies. So, the modulus of elasticity for each mass, in this particular analysis, is irrelevant. However, we chose the density to produce a block of mass=2 kg. Our block will have a volume of  $1 \text{ m}^3$ .

After entering the input, click on "APPLY". The "Isotropic Materials Box" should be showing on the screen, and at this point, the material number is still set to "1". Change the material number to "2", and click on "OK". The material properties box, like that below, re-opens. Again, enter "1" for EX, but this time, enter "1" for density. Click on "OK".

🗱 Isotropic Material Properties 🛛 🔀		
Isotropic Material Proper	ties	-
Properties for Material N	umber 1	
Young's modulus	EX	1
Poisson's ratio	NUXY	
Thermal expansion coeff	ALPX	
Reference temperature	REFT	
Friction coefficient	MU	
Damping multiplier	DAMP	
Density	DENS	2
Thermal conductivity	кхх	
Specific heat	С	
Enthalpy	ENTH	
Convection film coefficie	nt HF	
Emissivity	EMIS	
Heat generation rate	QRATE	
Viscosity	VISC	
Sonic velocity	SONC	
Relative permeability	MURX	
Magnetic coercive force	MGXX	
Electrical resistivity	RSUX	
Relative permittivity	PERX	
ОК Арріу	Cancel	Help

# 4. Specify real constants for the springs

Preprocessor -> Real Constants -> Add/Edit/Delete -> Add

The box below will open. Be sure that "Type 2 COMBIN14" is highlighted, and click on "OK"

🐮 🖪 Elem	ient Type fo	or Real Const	tants		×
		Choose	element typ	ie :	
	Туре Туре	1 2	PLANE42 COMBIN14		~
					<b>v</b>
		ок		Cancel	
					•

For real constant set "1", choose "K=5", as shown below. Leave the other fields blank, and click on "Apply". Then, change the Set Number to "2", and enter a value of "20" for K, then click on "OK", then "Close". Note that, for this model, we will not need any real constants for the PLANE42 elements. We are using the option that assumes a unit thickness for these elements.

🗱 Real Constant Set Number 1, for Cl	DMBIN14	×
Element Type Reference No.	2	
Real Constant Set No.		1
Spring constant	К	5
Damping coefficient	CU1	
Nonlinear damping coeff	CU2	
ОК Аррју	Cancel	Help

# 5. Create rectangles:

Preprocessor -> Modeling- Create -> -Areas- Rectangle -> By Dimensions

Fill in (X1,X2)=(5,6), and (Y1,Y2)=(0,1), as shown below. Then, click on "Apply".

👯 Create Rectangle by Dimensions	X
[RECTNG] Create Rectangle by Dimension	s
X1,X2 X-coordinates	5 6
Y1,Y2 Y-coordinates	0 1
OK Apply Can	cel Help

Create the block for the other mass by filling in (X1,X2)=(11,12), and (Y1,Y2)=(0,1), and then click "OK".

### 6. Specify mesh controls for the masses.

As stated previously, this model involves creating blocks to model the masses, but they are going to be set so that each mass moves as a rigid body. In general, in finite element modeling, the finer the mesh, the more accurate the results. However, in this case, the mesh density for each separate mass is irrelevant. So, each mass will be modeled as a single element. Choose:

Preprocessor -> -Meshing- Size Cntrls -> -Lines- Picked Lines

This opens up a Picking Menu, as shown on the next page. This box would allow for picking individual lines. But, in this case, we want a single element division for every line. So, in the Picking Menu, choose "Pick All". This opens up an "Element Sizes" box, also shown on the next page. Input "1" for "NDIV", as shown, and click on "OK".

Element Size on Picked Lines	
Element Sizes on Picked Lines	×
Pick O Unpick [LESIZE] Element sizes on picked lines	
• Single • Box SIZE Element edge length	
O Polygon O Circle NDIV No. of element divisions	
(NDIU is used only if SIZE is blank or zero)	
Count = 0	
Maximum = 8	
SPACE Spacing ratio	
Minimum = 1	
Line No. = ANGSIZ Division arc (degrees)	
For Keyboard Entry: ( use ANGSIZ only if number of divisions (NDIV) and	
• List of Items element edge length (SIZE) is blank or zero)	
O Min, Max, Inc	
OK Apply Cancel Help	
Reset Cancel	
FICK AII Help	

#### 7. Mesh the areas.

First, in case the areas are not currently plotted, along the top toolbar:

الله المراجعة الم المراجعة المراجع

Choose: *Plot -> Areas* 

Then, choose:

Preprocessor -> -Meshing- Mesh -> -Areas- Mapped -> 3 or 4 sided

A Picking Menu opens (which has the label "Mesh Areas" at the top of the menu). Click on the block on the left (which will be the 2 kg mass). It should highlight. Then, click on "OK" in the Picking Menu. Since we are only creating a single element in each block, at this point, a plot of the elements looks just like a plot of the areas. But, there are now nodes at the corners of the left-hand block, and this block contains a single PLANE42 finite element. Probably, an element plot has appeared showing only the single meshed element.

Re-plot the areas, as done above, by choosing, on the top toolbar: Plot -> Areas

Now, before meshing the second area, the material number needs to be changed to number 2, because the density of the two masses is different, and we defined different materials to account for this. To do this, use the path:

Preprocessor -> -Modeling- Create -> Elements -> Elem Attributes

The box below opens. Change the [MAT] entry to "2", as shown. No other changes are needed at this time, so click "OK".



Now, to mesh the second mass, choose:

Preprocessor -> -Meshing- Mesh -> -Areas- Mapped -> 3 or 4 sided

A Picking Menu opens (which has the label "Mesh Areas" at the top of the menu). Click on the block on the right (which will be the 1 kg mass). It should highlight. Then, click on "OK" in the Picking Menu.

# 8. Create a node at the wall:

Preprocessor -> Create -> Nodes -> In Active CS

In the box that opens, as shown on the next page, there is no input required for this case. This is only because we are going to create the node at (X,Y,Z)=(0,0,0). Also, we do not need a node number, because ANSYS will use the lowest available node number if we leave the node number field blank. Just click on "OK".



It is probably a good idea now to turn off the display of the X-Y-Z triad, because it covers up the node at the wall in the plots. Along the top toolbar:

IN ANSYS/Faculty Research Utility Menu Ele Select List Blot Plot⊆tris WorkPlane Pagameters Macro MeguCtris Help



In the box that opens, for [/TRIAD], select "Not shown", then click on "OK".

I/PLOPTS1 Window Options     INFO   Display of legend   Auto Legend     LEG1   Legend header   Important of legend   Important of legend     LEG2   Uiew portion of legend   Important of legend   Important of legend     LEG3   Contour legend   Important of legend   Important of legend     FRAME   Window frame   Important of legend   Important of legend     FITILE   Title   Important of legend   Important of legend     MINM   Min-Max symbols   Important of legend   Important of legend     LOG0   ANSYS logo display   Graphical logo   Important of legend     WINS   Automatic window sizing -   Important of legend   Important of legend     WP   WP drawn as part of plot?   Important of legend   Important of legend
INF0   Display of legend   Auto Legend     LEG1   Legend header   ✓ On     LEG2   View portion of legend   ✓ On     LEG3   Contour legend   ✓ On     FRAME   Window frame   ✓ On     TITLE   Title   ✓ On     MINM   Min-Max symbols   ✓ On     LOG0   ANSYS logo display   Graphical logo     WINS   Automatic window sizing -   ✓ On     - when entire legend turned on or off   WP   WP drawn as part of plot?
LEG1   Legend header   ♥ On     LEG2   View portion of legend   ♥ On     LEG3   Contour legend   ♥ On     FRAME   Window frame   ♥ On     TITLE   Title   ♥ On     MINM   Min-Max symbols   ♥ On     LOG0   ANSYS logo display   Graphical logo     WINS   Automatic window sizing -   ♥ On     • when entire legend turned on or off   Wo
LEG2   View portion of legend   ♥ On     LEG3   Contour legend   ♥ On     FRAME   Window frame   ♥ On     TITLE   Title   ♥ On     MINM   Min-Max symbols   ♥ On     LOG0   ANSYS logo display   Graphical logo     WINS   Automatic window sizing -   ♥ On     - when entire legend turned on or off   WP     WP   WP drawn as part of plot?   No
LEG3   Contour legend   ✓ On     FRAME   Window frame   ✓ On     TITLE   Title   ✓ On     MINM   Min-Max symbols   ✓ On     LOG0   ANSYS logo display   Graphical logo ▼     WINS   Automatic window sizing -   ✓ On     - when entire legend turned on or off   WP     WP drawn as part of plot?   No
FRAME   Window frame   Image: On     TITLE   Title   Image: On     MINM   Min-Max symbols   Image: On     LOGO   ANSYS logo display   Image: Graphical logo     WINS   Automatic window sizing -   Image: On     - when entire legend turned on or off   Image: On     WP   WP drawn as part of plot?   Image: No
TITLE   Title   ✓ On     MINM   Min-Max symbols   ✓ On     LOGO   ANSYS logo display   Graphical logo   ✓     WINS   Automatic window sizing -   ✓ On     - when entire legend turned on or off   WP   WP drawn as part of plot?   No
MINM   Min-Max symbols   ✓ On     LOGO   ANSYS logo display   Graphical logo   ✓     WINS   Automatic window sizing -   ✓ On   ✓     - when entire legend turned on or off   WP   WP drawn as part of plot?   No
LOGO ANSYS logo display Graphical logo ▼ WINS Automatic window sizing - ▼ On - when entire legend turned on or off WP WP drawn as part of plot? No
WINS Automatic window sizing - 🔽 On - when entire legend turned on or off WP WP drawn as part of plot? 🔽 No
- when entire legend turned on or off WP WP drawn as part of plot? No
WP WP drawn as part of plot? No
[/TRIAD] Location of triad Not shown
[/REPLOT] Replot Upon OK/Apply? Replot
OK Apply Cancel Help

Now, it is probably a good idea to turn the node numbers on. They may be showing already at this time, but to make sure they are, on the top toolbar:

Back on the top toolbar, again choose "PlotCtrls", then "Numbering", and the box below opens. Toggle the "Node" option from "Off" to "On" by clicking on the box. You should see a check mark appear in the box, as shown below.

R Plot Numbering Controls	×
[/PNUM] Plot Numbering Controls	
KP Keypoint numbers	🗖 Off
LINE Line numbers	☐ Off
AREA Area numbers	☐ Off
VOLU Volume numbers	🗖 Off
NODE Node numbers	🔽 On
Elem / Attrib numbering	No numbering 💌
TABN Table Names	☐ Off
SUAL Numeric contour values	□ Off
[/NUM] Numbering shown with	Colors & numbers 💌
[/REPLOT] Replot upon OK/Apply?	Replot 💌
OK Apply Canc	el Help

Although nodes are probably already plotted now, to make sure, on the top toolbar, choose *Plot -> Nodes*.

# 9. Create the spring elements.

First, the element type needs to be changed to type 2, which is the element type number we used in defining the Combin14 (spring) element type. To do this, use the path:

Preprocessor -> -Modeling- Create -> Elements -> Elem Attributes

The box below opens. Change the [TYPE] entry to "2 COMBIN14" as shown. No other changes are needed at this time, so click "OK".

Rement Attributes	×
Define attributes for elements	
[TYPE] Element type number	2 COMBIN14
[MAT] Material number	1 🔽
[REAL] Real constant set number	1 🗖
[ESYS] Element coordinate sys	0 🔽
[SECNUM] Section number	None defined 💽
[TSHAP] Target element shape	Straight line 💌
OK Cancel	Help

Now, to create the first spring element, choose:

Preprocessor -> -Modeling- Create -> Elements -> -Auto Numbered - Thru Nodes

A picking menu opens. It should be that the numbered nodes are plotted on screen at this time. Node number 9 should be the node at the wall, and node number 1 should be the node at the bottom left-hand side of the 2 kg mass.

Click once, with the left mouse button, on node 9, then click again, once, on node 1. Then, in the picking menu, click on "OK".

Now, before creating the other spring, the real constant set number needs to be changed to set 2, because the spring constant is different for the two springs. To do this, use the path:

*Preprocessor -> -*Modeling- *Create -> Elements -> Elem Attributes* 

In the box that opens, change the [REAL] entry to "2". No other changes are needed at this time, so click "OK". Then, choose:

```
Preprocessor -> -Modeling- Create -> Elements -> -Auto Numbered - Thru Nodes
```

A picking menu opens. Node number 2 should be the node at the bottom-right corner of the left-hand mass, and node 5 should be the node at the bottom-left corner of the right-hand mass. Click once, with the left mouse button, on node 2, then click again, once, on node 5. Then, in the picking menu, click on "OK".

Plot the elements, by choosing, on the top toolbar: Plot-> Elements

# **10.** Apply constraints.

It is assumed that only X-direction motion is allowed. So, the Y and Z direction motion must be constrained to zero at all nodes.

```
Preprocessor -> Loads -> Apply -> Displacement -> On Nodes
```

A picking menu applies. We want this constraint to apply to all nodes in the model. So, in the picking menu, choose "Pick All". Another box opens. **Be sure** to Un-highlight "All DOF", by clicking on it. Then, highlight "UY" and "UZ" by clicking on these labels, as shown, and then click on "Apply".

R Apply U,ROT on Nodes	×
[D] Apply Displacements (U,ROT) on No	des
Lab2 DOFs to be constrained	All DOF UX UY UZ
Apply as	Constant value 💌
If Constant value then:	
VALUE Displacement value	
ОК Аррју	Cancel Help

The picking menu should remain open. Click on the node at the wall (this should be node number 9), and then click on "OK" in the Picking Menu. This node should not move in any direction. So, choose, in the box that opens, as shown on the previous page, click on "All DOF", and then click on "OK".

## 11. Make rigid body constraints for each mass.

Each mass is supposed to move as a rigid body. We can produce this effect by coupling all nodes on each mass so that they move together. To do this, choose:

*Preprocessor -> Coupling / Ceqn -> Couple DOFs* 

A picking menu opens. Click on nodes 1-2-3-4, to highlight all four nodes that define the left-hand mass. Then, in the Picking Menu, click on "Apply". A box, as shown below, opens. Enter "1" for "NSET", as shown, and leave the label "UX" for "Lab". Then, click on "Apply".

🐮 🖥 Defi	ne Coupled DOF:	5			×	
ECP 1	Define Set	of Coupled DOF	s			
NSET	Set refere	nce number	1	1		
Lab	Degree-of-freedom label		UX	UX		
	ок	Apply	Cancel	Help		

The picking menu should re-open. Click on nodes 5-6-7-8, to highlight all four nodes that define the right-hand mass. Then, in the Picking Menu, click on "OK". A box, as shown above, re-opens. This time, Enter "2" for "NSET", and again leave the label "UX" for "Lab". Then, click on "OK".

## 12. Specify analysis type and options.

Return to the main ANSYS Menu, and choose:

Solution -> -Analysis Type- New Analysis

In the box that opens, select "Modal" and click on "OK".

Then, select: Solution -> Analysis Options

The box below opens. Enter "2" for "No. of modes to extract" and "2" for "NMODE No. of modes to expand". Choose "OK", and the box at bottom opens. Enter "1000" for "FREQE", as shown, then click on "OK".

👯 Modal Analysis	×
[MODOPT] Mode extraction method	
	💿 Subspace
	🔿 Block Lanczos
	O Powerdynamics
	O Reduced
	🔿 Unsymmetric
	🔿 Damped
	O QR Damped
No. of modes to extract	2
<pre>(must be specified for all methods exce</pre>	pt the Reduced method)
[MXPAND]	
Expand mode shapes	🔽 Yes
NMODE No. of modes to expand	2
Elcalc Calculate elem results?	No
[LUMPM] Use lumped mass approx?	No No
-For Powerdynamics lumped mass	approx will be used
[PSTRES] Incl prestress effects?	🗖 No
OK Canc	el Help

👯 Subspace Modal Analysis	×
[MODOPT] Mode Extraction Options	
FREQB Start Freq (initial shift)	0
FREQE End Frequency	1000
Nrmkey Normalize mode shapes	To mass matrix 💌
[RIGID] Known rigid body modes	All DOF UX UY UZ
[SUBOPT] Subspace iteration options	
SUBSIZ Subspace working size	8
NPAD No. of extra vectors	4
NPERBK No of modes/memory block	6
Number of subspace iterations	
NUMSSI Maximum number	0
NSHIFT Min, before shift	0
Strmck Sturm sequence check	At shift+end pts 💌
OK Car	Help

# 13. Solve

Choose: Solution -> -Solve- Current LS

A green box will appear. Just click on "OK" in the box. You should then, very quickly, get a yellow box, with the note "Solution is done!". You can close this box.

# **14. List Natural Frequencies**

Return to the ANSYS Main Menu, and choose:

General Postproc -> Results Summary

You should get a box listing two natural frequencies. One is for "Set 1" and the other is for "Set 2". Set 1 is the set of results corresponding to Mode 1, and Set 2 is the set of results corresponding to Mode 2. Note the natural frequencies (these are listed in units of "Hz"), then close the listing box.

### **15.** Animate the Mode Shapes

Choose: General Postproc -> -Read Results- First Set

Then, on the top toolbar, choose: *PlotCtrls -> Animate -> Mode Shape* 

In the box that opens, you may want to change "No. of frames to create" to some higher number than the default of 10. For instance, you may enter 20. Also, you may want to change the time delay to some smaller number than the default of 0.5. For instance, you may enter 0.2. Then, in that box, click on "OK".

It may take a couple of minutes for the animation to begin, but eventually, you should see, on the screen, the expected motion corresponding to the first mode shape. At any time, you can click on "Close" in the blue "Animation Controller" box to end the animation.

To animate the second mode shape, choose:

General Postproc -> -Read Results- Next Set

Again, back on the top toolbar, choose: *PlotCtrls -> Animate -> Mode Shape* 

Repeat the procedure outlined above, and you should see the animation of the second mode shape.

## **16. List the Mode Shapes**

Choose: General Postproc -> -Read Results- First Set

Then, choose: General Postproc -> List Results -> Nodal Solution

In the box that opens, as shown below, keep the defaults by just clicking on "OK".

Clist Nodal Solution			X
[PRNSOL] List Nodal Solution			
Item,Comp Item to be listed	DOF solution Stress Strain-total Nonlinear items Strain-elastic Strain-plastic Strain-plastic Strain-creep Strain-other	All DOFs DOF Translation UX UZ All U's UCOMP Rotation ROTX ROTY All DOFs DOF	• - - -
[AUPRIN] Eff NU for EQU strain	Ø		
OK	Apply	Cance1	Help

A listing of the relative displacements for each node for mode 1 is shown, and the frequency for the mode is also shown.

You can get a hard copy of the information in this box by clicking, in this listing box: *File -> Print*. It <u>should</u> also be possible to save this information to a file using the option, *File -> Save As*. However, this "Save As" option does not appear to be working properly in the current ANSYS version. Another option, however, is to use the mouse to block this information in (hold down left mouse button and drag to highlight the text and numbers), then, on the keyboard, simultaneously hit the Control Key, and the letter "c" (Ctrl-C), then, open up either "Notepad", or some word processing program, such as Microsoft Word. In the program of your choosing, choose "Paste", or else use "Ctrl-V", to insert the text copied from the listing box. If desired, you may close the listing box to get it out of the way. To do this, in that listing box, choose: *File -> Close*.

Now, list the second mode shape in a similar way. First, choose:

*General Postproc ->* -Read Results- *Next Set* 

Then, as above, choose: General Postproc -> List Results -> Nodal Solution

In the box that opens, choose "OK". You can again get either a hard copy print-out of the information, or save it in a file, as outlined above.

# 17. Exit the ANSYS program, saving all data.

On the ANSYS Toolbar, shown below, choose:

*Quit ->Save Everything -> OK* 

🚸 ANSYS Toolbar 🔀 🔀		
SAVE_DB		
RESUM_DB		
QUIT		
POWRGRPH		

To recall the model and solution at a later date, assuming you have deleted no files, simply re-launch ANSYS, specify the same working directory as before, re-issue the same jobname as used in Step 1 of these instructions, and then click on "RESUME\_DB" on the ANSYS Toolbar shown above.

To see the resumed model in the graphics window, you may then need to click on "Plot" on the top Utility Menu:

<mark>W ANSYS/Faculty Research Utility Menu (platetmp)</mark> Elle Select List Plot Plot<u>C</u>tris <u>W</u>orkPlane Parameters Macro MenuCtris <u>H</u>elp

Then, choose either "Elements", "Nodes", or "Areas", depending on which entities you wish to plot.