

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

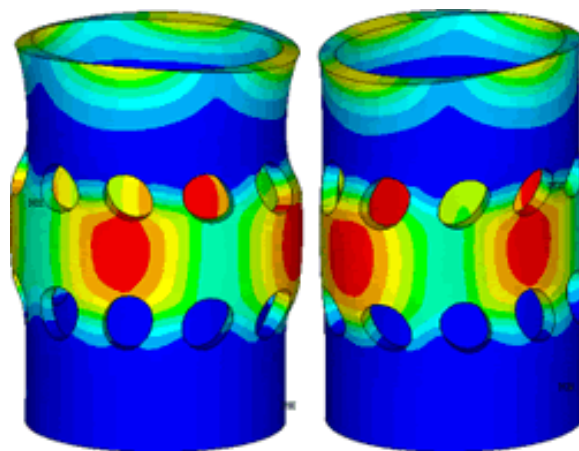
- [Annotation in ANSYS](#)
- [Eigen Buckling with Cyclic Symmetry](#)
- [Are You an APDL Guru?](#)

On the Web

- [ANSYS "Solutions" Magazine Back to Hardcopy Format](#)
- [PADT: CFX Overview Seminar](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - *The Focus* Library
 - Contributor Information
 - Subscribe / Unsubscribe
 - Legal Disclaimer



ANSYS
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.



A Publication for ANSYS Users

Post Processing Tips & Hints: Annotation in ANSYS

by Eric Miller, PADT

Post processing is an oft-maligned part of the analysis process. Real analysts focus on meshing and running their models. But the fact is that getting meaningful answers is the whole point of doing a simulation, so spending a little bit of time to learn how to produce more concise and digestible results is a good investment. To that end, we will be covering a couple of key ways to produce higher quality results using POST1 and POST26, including listing, graphing, animation, and for our fourth article, the ins and outs of creating annotated plots.

The purpose of a plot created in ANSYS is to capture some key piece of information. Often, the standard image and text do not convey everything that you want to capture. The most common solution, and probably the simplest, is to put text in your report in the form of narrative or a table. However, this is a time consuming and error prone process. It would be much better if you could capture the information you want while you are creating plots and place it directly on the plot. Annotation is the way to do that. Furthermore, you can use annotation as a way to clean things up and present a more polished look to your plots.

Annotation Basics

The first thing to understand about annotation is that there are two flavors in ANSYS: 2D and 3D. You can think of 2D annotation as writing on a transparent window on top of your graphics. 3D annotation creates objects that are attached to ANSYS entities like nodes, areas or lines. This means that when you move your model by rotating or panning, 2D annotations stay put and 3D annotation move with the entities they are attached to.

Another basic technique of annotation is that it is that you should really set things up using the GUI. There are tools for both 2D and 3D annotation that are easy to use, and everything you do can be written to a file for inclusion in a macro. Since it's tough to locate things by typing in values, using the GUI can save a lot of time.



A Publication for ANSYS Users

2D Annotation

Figure 1 shows the 2D annotation tool. This control is used to create and edit any 2D annotations you create. As shown, the tool changes based upon what type of annotation object you want to create. Once you have set your options, look at your prompt in the ANSYS graphics window for what to do next. Generally, you specify a location for placing your object. Figure 2 shows examples of all the annotations you can create.

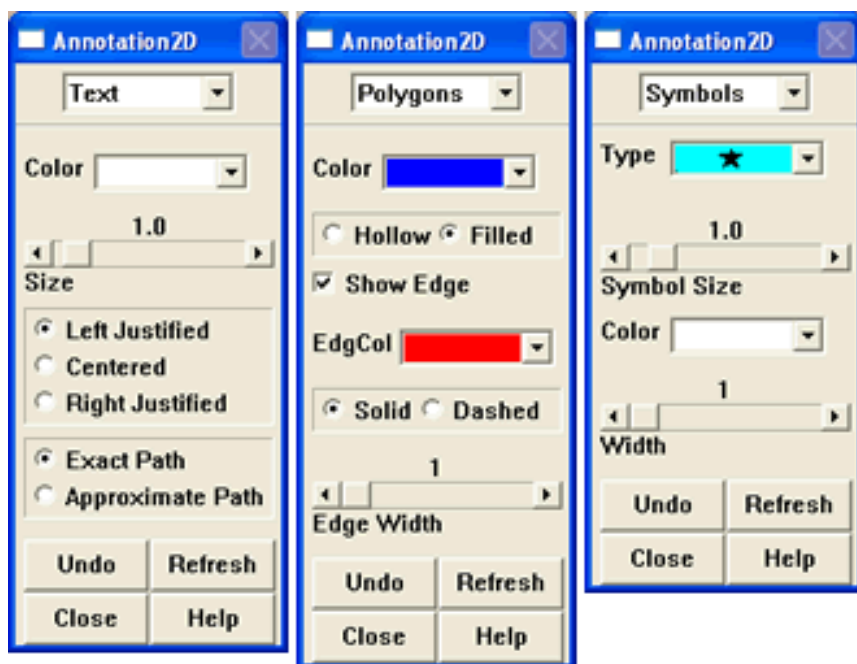


Figure 1. 2D Annotation Tool.

The key thing to remember while writing APDL that uses 2D annotation is that the object locations are in the window coordinate system, not your model's coordinate system. The lower left hand corner is $(-1, -1)$ and the upper right is $(1.68, 1)$.



A Publication for ANSYS Users

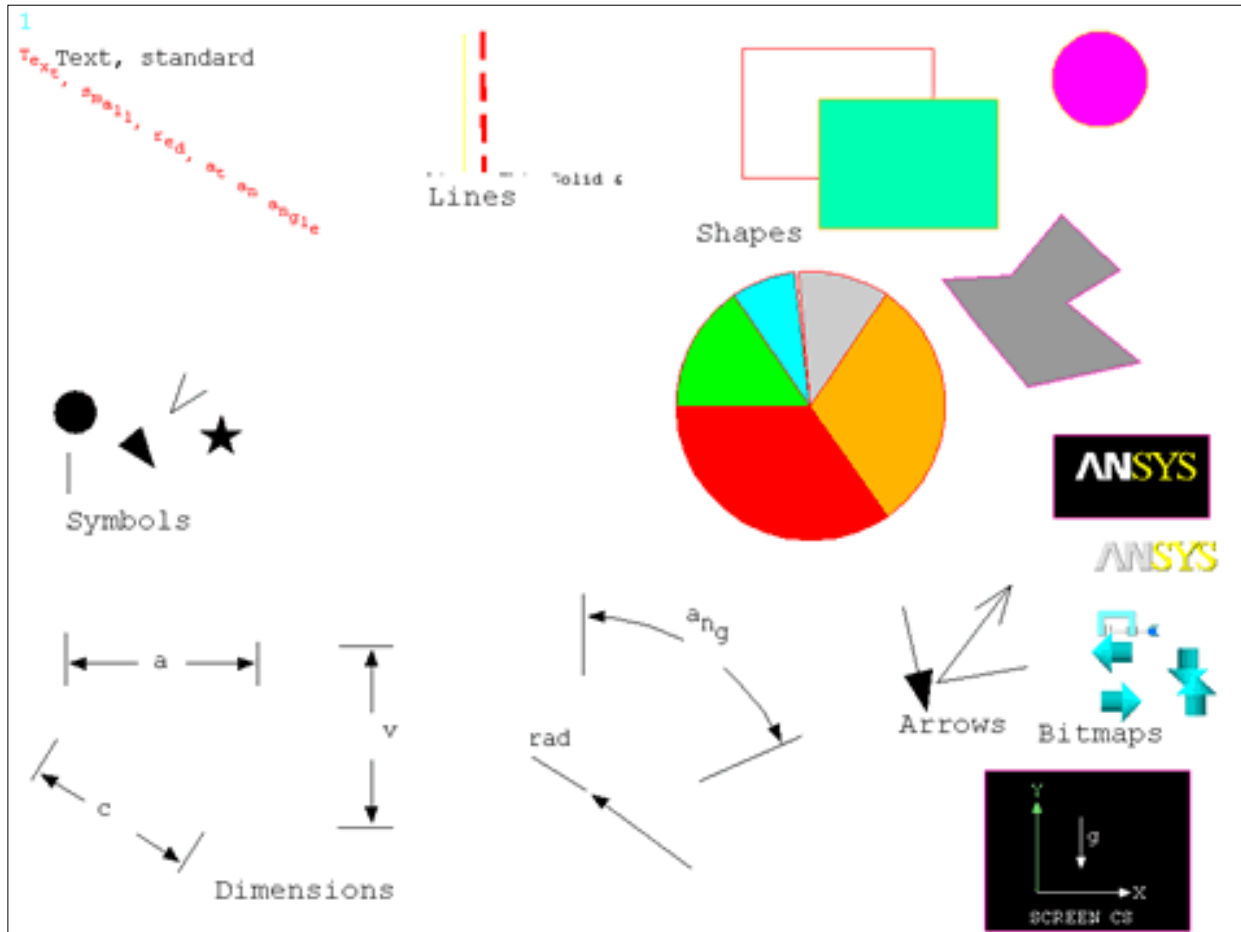


Figure 2. 2D Annotation Options.

Here are some pointers on the key types:

Text

This is the most common type of annotation. I find that things work best if I choose the approximate orientation option. This snaps things to horizontal and vertical.

When you are ready to enter your text, click once for the start point and then click a second time to specify the orientation. Then type in your words and enter a return. The most common error with this command is forgetting to hit a return. If you go on to the next object, your typing will be lost.

You can change the font by going to PlotCtrls->Font Controls->Anno/Graph Fonts. I find that the default Courier font can sometimes be hard to read, especially at smaller sizes. Arial works nice.

The Focus



A Publication for ANSYS Users

Arrows

It is often nice to use arrows to point to things. Do note that in 2D annotation, the arrow does not move with the object it is pointing at. Because of this, I tend to use 3D annotation for that sort of thing and use the 2D arrows to point at other annotation objects.

Pies

If you need to show simple pie charts this annotation command works well. You can eyeball it using the GUI, then save the annotation commands (see Options below), and finally put in your real values. The macro in Figure 3 shows a very simple pie chart with a legend made with annotation.

```

/annot,delete
sz = .8
xloc = 0
yloc = 0

*dim,data,,5
data(1) = 12,15,28,10,32
hsz = sz/2

/pspec,0,1,1
/poly,4,xloc-hsz,yloc-hsz,1.8*(xloc+hsz),yloc-hsz,
    1.8*(xloc+hsz),yloc+hsz,xloc-hsz,yloc+hsz

x0 = xloc + hsz
y0 = yloc + .7*hsz
lof = .05

*vscfun,dsum,sum,data(1)
/LSPE, 15, 0, 1.000
/TSPEC, 15, 0.700, 1, 0, 0
ang1 = 0
*do,i,1,5
    ang2 = ang1 + (360*data(i)/dsum)
    /PSPE, 2*i, 1, 1
    /PWED, xloc,yloc,sz*.4, ang1,ang2
  
```



A Publication for ANSYS Users

```
/poly,4,x0,y0,x0+lof,y0,x0+lof,y0+lof,x0,y0+lof
pv1 = 100*data(i)/dsum
/tlab, x0+1.5*lof,y0, %pv1% %

y0 = y0 - 1.5*lof
ang1 = ang2
*enddo
```

Figure 3. Example Pie Chart.

Bitmaps

One of the least known but cooler options in annotation is the ability to add a bitmap to your plot. By default, the program comes with the 8 bitmaps shown in Figure 2, including two versions of the ANSYS logo. You can also add your own bitmaps with the /TXTRE command with FILE as the first option. Give it a number and a bitmap name and you can place the image using the /LSYMBOL command. Figure 4 shows both a PADT logo and an image of actual hardware on the plot.

One trick with bitmaps: Black pixels are treated as transparent. You need to make them slightly grey to get them to show up. Most graphics editors can do that. This feature is nice for logos, but if you want to do more, I recommend that you save your ANSYS plot as a bitmap and edit it elsewhere.

The Focus



A Publication for ANSYS Users

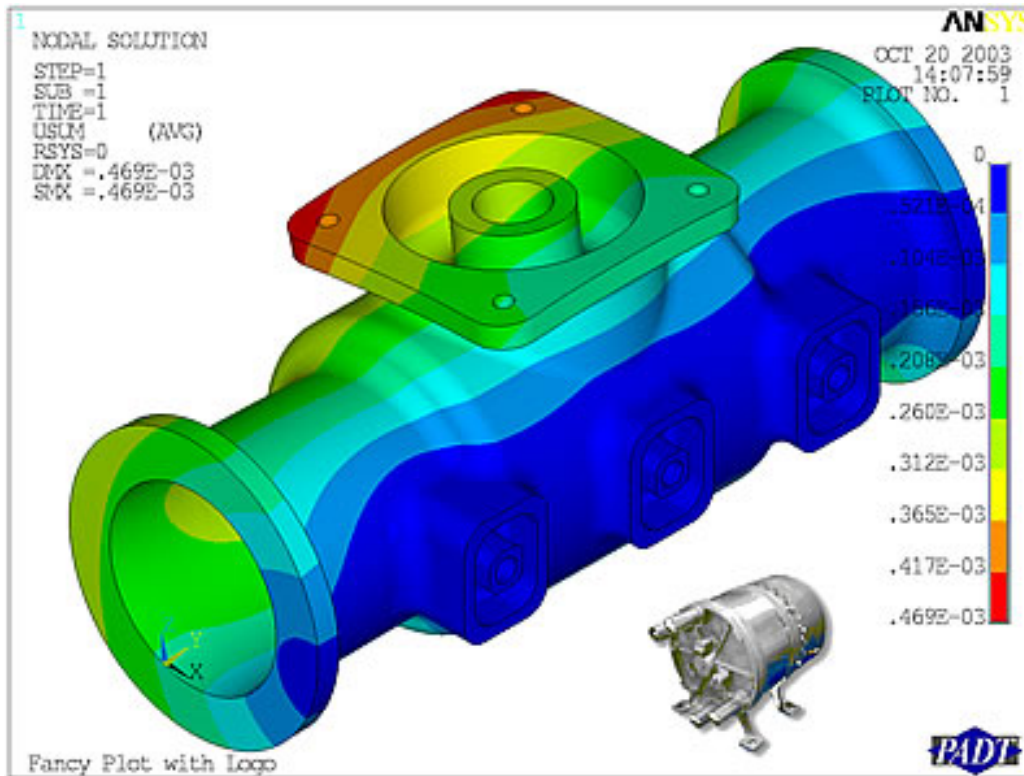


Figure 4. Bitmap Example

Options

The most important control in the 2D Annotation Tool is the options control. Figure 5 shows a picture of this configuration of the tool. You can use it to copy, move, resize, and delete annotation objects by choosing one of those options from the selection area. In addition, you can add a reference grid, delete it, or snap to it, as well as set its grid size. I have found that the enable snap, with the grid off, is a very useful way to get a nice layout with everything sort of lined up.

You can also use the options option to read and write annotation files. By default ANSYS will save in jobname.ano, but I like to give it my own name. If you want to script, this is where you will capture your objects as laid out in the GUI. There is also a Delete All button, which is the same as the /anno,dele command.



A Publication for ANSYS Users



Figure 5. Option Configuration of 2D Annotation Tool.

2D Annotation “Unique Behavior”

This feature in ANSYS was developed awhile ago before the widespread access to Macs or usable versions of MS Windows. When people ran on terminals or workstations there was really no way to dress up a plot. So, being old and set up for some low level devices, how you use it can sometimes surprise you. Here are some hints to help you get along better:

- When you create an annotation object you are basically scribbling on a sheet of glass over the graphics space. Unless you delete the object or clear the annotation, things will show up on all of your plots. /ERASE and plotting commands do not clear anything.
- There is very little documentation on the mouse clicks you need to place various objects. I read the prompt in the ANSYS window and use the Undo and Delete options a lot.
- Don't forget to enter a return after you have typed in text.
- Every entity has a “hot spot” that is used for the Copy, Move, Resize and Delete options. I tend to leave the command /ANUM out of my scripts. You will notice a /ANUM before every object if you look at a *.ano file. I just delete them unless I plan on messing with positions through the GUI.
- You can use parameter substitution using %var%. However, every time you



A Publication for ANSYS Users

plot it gets updated so if var changed, the annotation changes. You need to be especially careful if you are making your own animations using annotation.

- Trying changing the font. This makes a big difference in the readability of the annotation if you have a lot of information.

3D Annotation

A very nice tool for annotating results, boundary conditions and other “model space” items is 3D annotation. This is very similar to 2D annotation except you specify locations in the global coordinate system. So, as you spin and pan, the annotations move, too.

Figure 6 shows the controls for two entity types for 3D annotation. The user specifies where to place the object by one of three methods, depending on what option they choose: Specifying the location type in the second drop-down and clicking on a model entity; specifying an X,Y,Z position; or by picking a point on the screen. If you are clicking on a model entity, ANSYS converts that entity’s location into an X,Y,Z position. If you click on the screen, it places the point on the view plane directly under where you pick.



A Publication for ANSYS Users

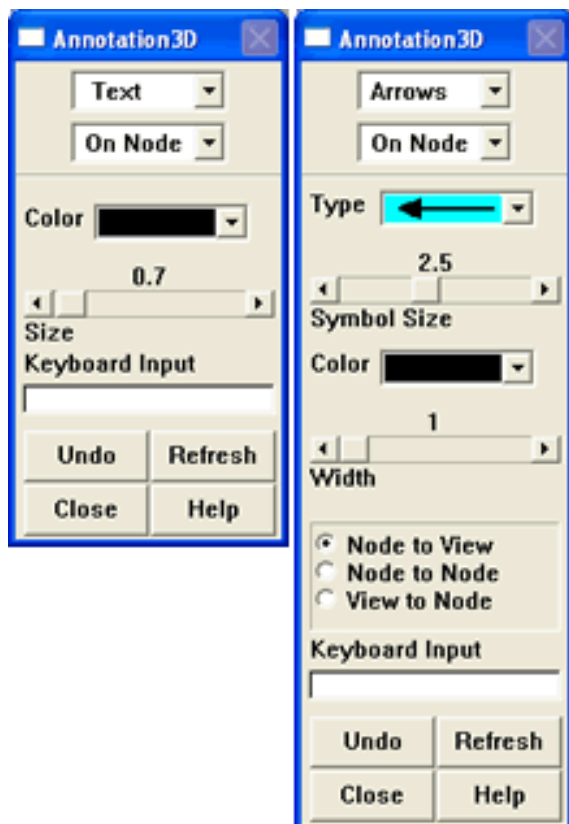


Figure 6. Annotation Control.

A simple example of how to use 3D annotation is to create leaders from text to a node. The example below specifies the Z position of each layer of nodes in a simple stack of square elements. It places the text, a line, and the end of an arrow at a point in space, then forces the head of the arrow to point to a given node (see Figure 7).

```

/TSPE, 15, 1.000, 1, 0, 0
*do,i,1,91,10
zval = nz(i)
/AN3D,TEXT, 0.0, -2.5 ,nz(i)+.3 ,Z = %zval%
/AN3D,line, 0.0, -2.0 ,nz(i)+.5 ,0.0, -2.5,nz(i)+.5
/AN3D,ARRO, 1,0.0,-2.0,nz(i)+.5,nx(i),ny(i),nz(i)
*enddo
  
```

Figure 7a. 3D Leader Lines Example.

The Focus



A Publication for ANSYS Users

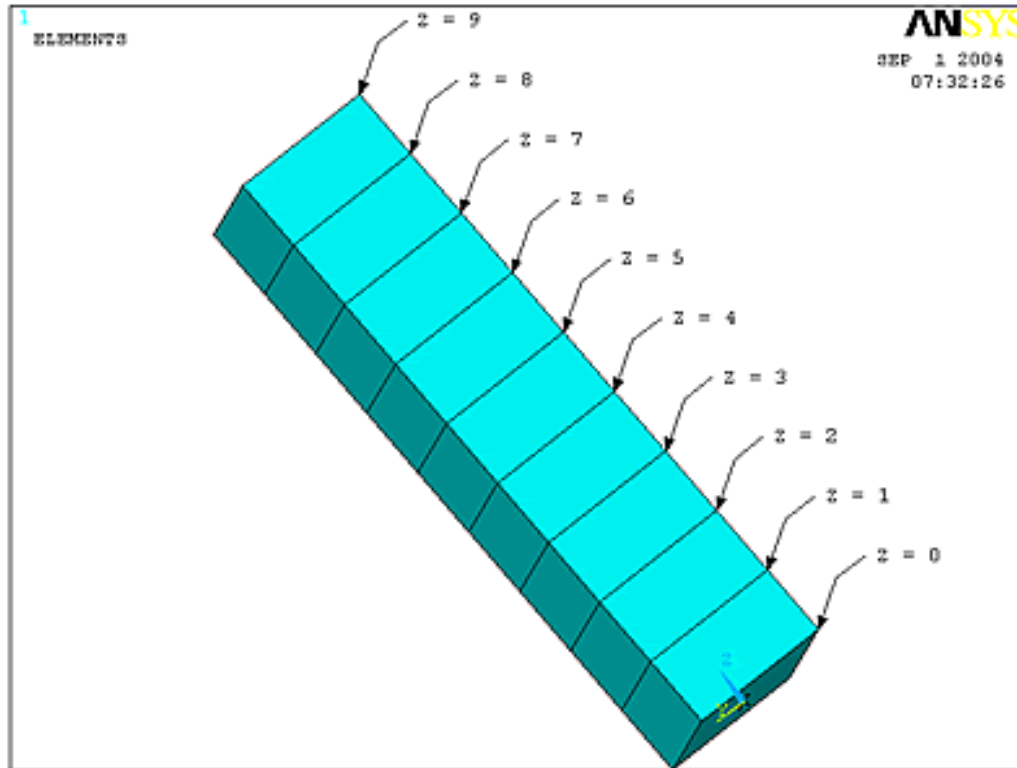


Figure 7b. 3D Leader Lines Example.

A typical modification of this is to find the max result value in a model and have an arrow point to it. Here is an example for putting an arrow to the max USUM in our simple model for a model shape (see Figure 8).

```

nsort,u,sum
*get,usmx,sort,,max
*get,nusmx,sort,,imax
*get,dsf,graph,1,dscale,dmult
x1 = nx(nusmx) + ux(nusmx)*dsf
y1 = ny(nusmx) + uy(nusmx)*dsf
z1 = nz(nusmx) + uz(nusmx)*dsf
/AN3D,ARRO, 1, x1+1,y1+1,z1+1,x1,y1,z1
/AN3D,line, x1+1,y1+1,z1+1,x1+1.5,y1+1.5,z1+1
/AN3D,TEXT, x1+1.5,y1+1.5,z1+1,Max = %usmx% @ %nusmx%

```

Figure 8a. Example of Max Value Leader.

The Focus



A Publication for ANSYS Users

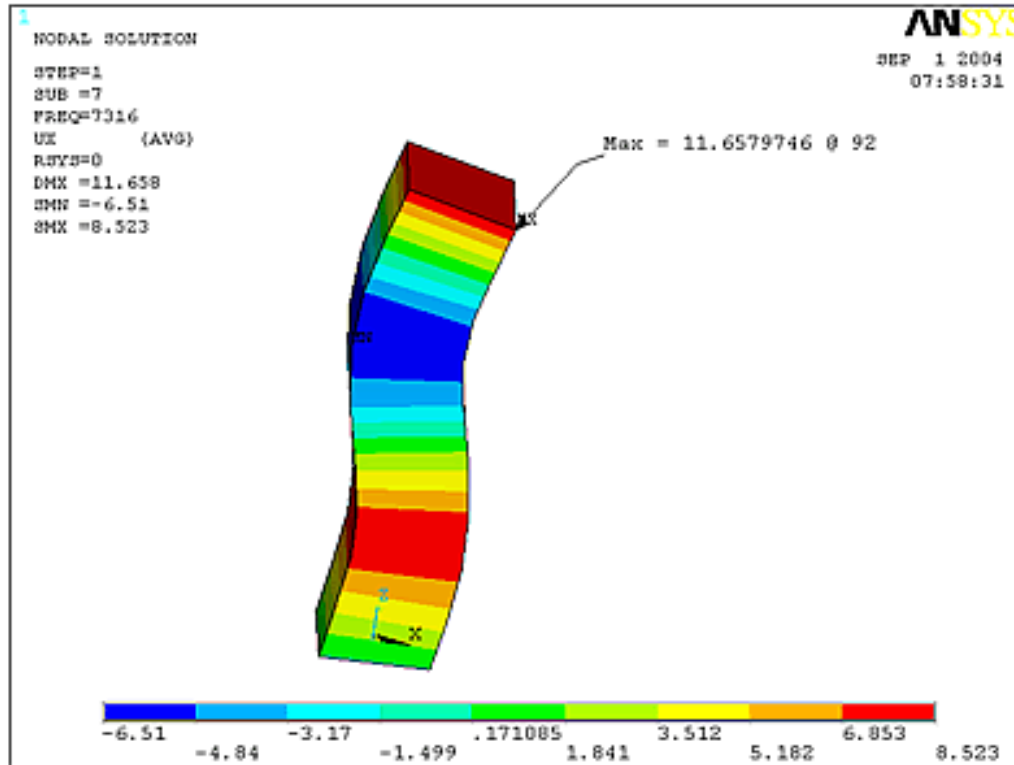


Figure 8b. Example of Max Value Leader.

Once you have played with this feature for a while, you find one drawback: you cannot easily make your macros general because the X,Y,Z position of things changes with the shape and size of your model. So leave things so you can tweak them with little effort.

Besides Lines, Arrows, and Text, 3D annotation supports Symbols, Polygons, and Bitmaps. So you can place these objects on your model and have them also move with the model. And finally, the control has the same Options configuration as 2D annotation, so you can modify, delete, save, and restore your 3D annotations.

General Thoughts

Once you have experimented with these capabilities, the power of this tool becomes quite obvious, especially if you are doing some sort of parametric study. Instead of summarizing things in a table, you can show the variables right on the plot along with the results. I find this very useful in meetings because you do not have to switch between plots and tables all the time.

My most common use for annotation is to put a little table on my plots that



A Publication for ANSYS Users

summarize the key inputs and outputs for the run. I will often split the plot into three graphics windows and use the forth for my annotation table. As with anything in ANSYS, if you script it with APDL, it is easy to do and is repeatable.

However too much of a good thing is bad. I've seen some plots where the model took up less than 25% of the area and the rest was a narrative worthy of an epic Russian novel. Keep it simple and always ask yourself if the information you are adding is really necessary. Also, if you are going to add the information only once, it may be faster to just do it in PowerPoint, Word, or your favorite graphics-editing program.



A Publication for ANSYS Users

Eigen Buckling with Cyclic Symmetry

“Look Ma,” Said Cyclic Symmetry, “Eigen Buckle Now!”

by Jeff Strain, PADT

The ability to perform a modal analysis on a cyclic symmetric structure has been around for several versions, with the procedure being completely overhauled at version 6.0. In more recent versions, the ability to perform an eigen buckling analysis on a cyclic symmetric structure, using a procedure that combines regular eigen buckling with modal cyclic symmetry, has been added.

Procedure

As I guide you through the procedure, I will use what I call “Non-Descript Hollow Cylinder with Holes” as an example. This example was inspired by a Duchamp painting I once saw at the Guggenheim, or maybe I was shooting beer cans with a .22. I don’t remember.

The Focus



A Publication for ANSYS Users

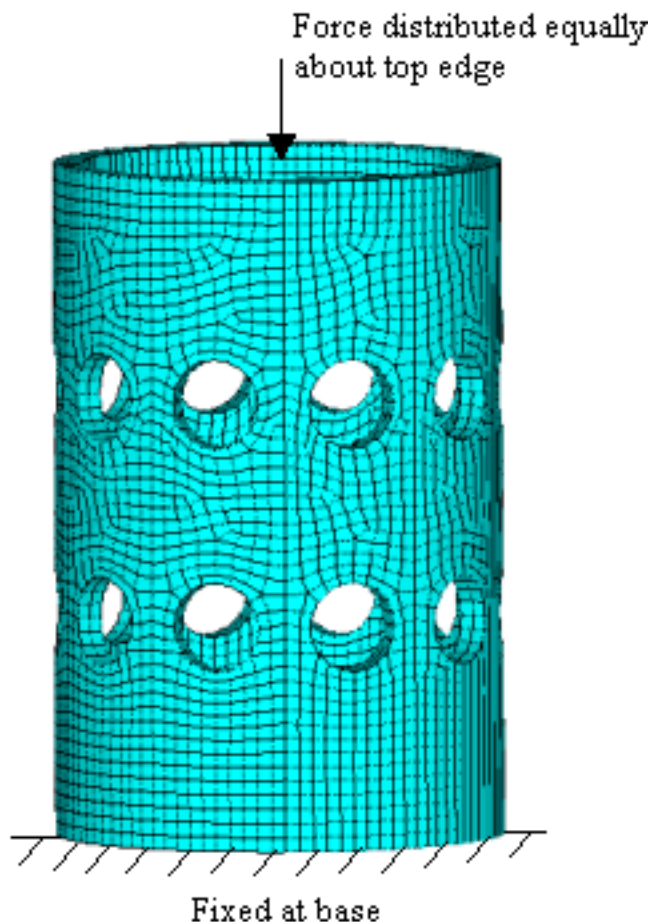


Figure 1. Cylinder buckling model.

Modeling

The first step is to model a sector as shown in Figure 2 and execute the CYCLIC command (Preprocessor > Modeling > Cyclic Sector > Cyclic Model) just as you would for a [modal cyclic symmetry analysis](#). The same caveats regarding sector angle and node matching apply.

The Focus



A Publication for ANSYS Users

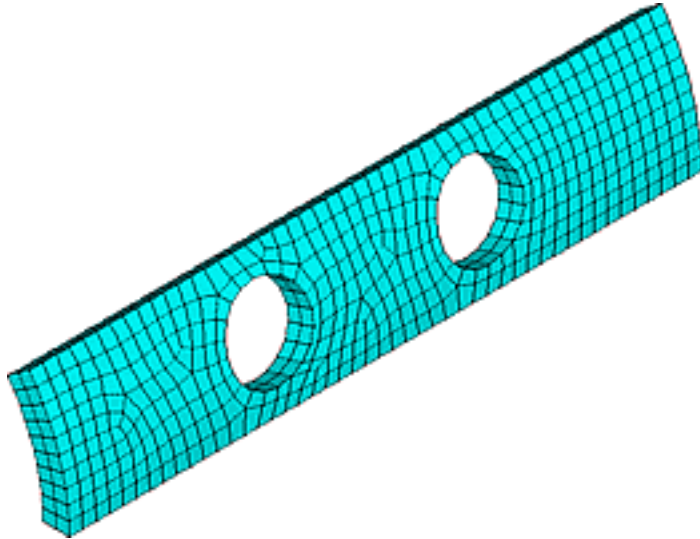


Figure 2. Sector model.

Prestress Solution

In the Solution processor, make sure the analysis type is set to static (Solution > Analysis Type > New Analysis). Turn prestress effects on (Solution > Unabridged Menu; Solution > Analysis Type > Analysis Options).

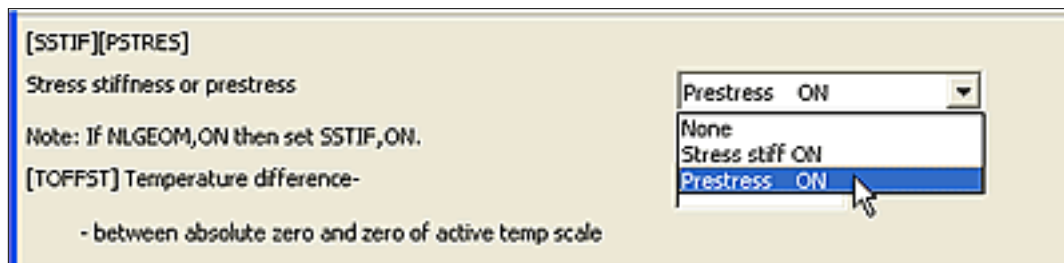


Figure 3. Turn prestress on.

When performing an eigen buckling analysis, the standard procedure is to apply a given load during the prestress run and then multiply it by the load factors calculated during the eigen buckling run to obtain the buckling loads. Obviously, a unit load is preferred since this makes the calculation simple. However, if you're applying point forces, the total force on the sector should be $1/N$ where N is the number of sectors. Also, if you're applying a distributed force, note that the edge nodes should only receive half the value of the forces applied to the interior nodes to account for symmetry (see Figure 4). If you're applying pressure, you don't have to worry about this.

The Focus



A Publication for ANSYS Users

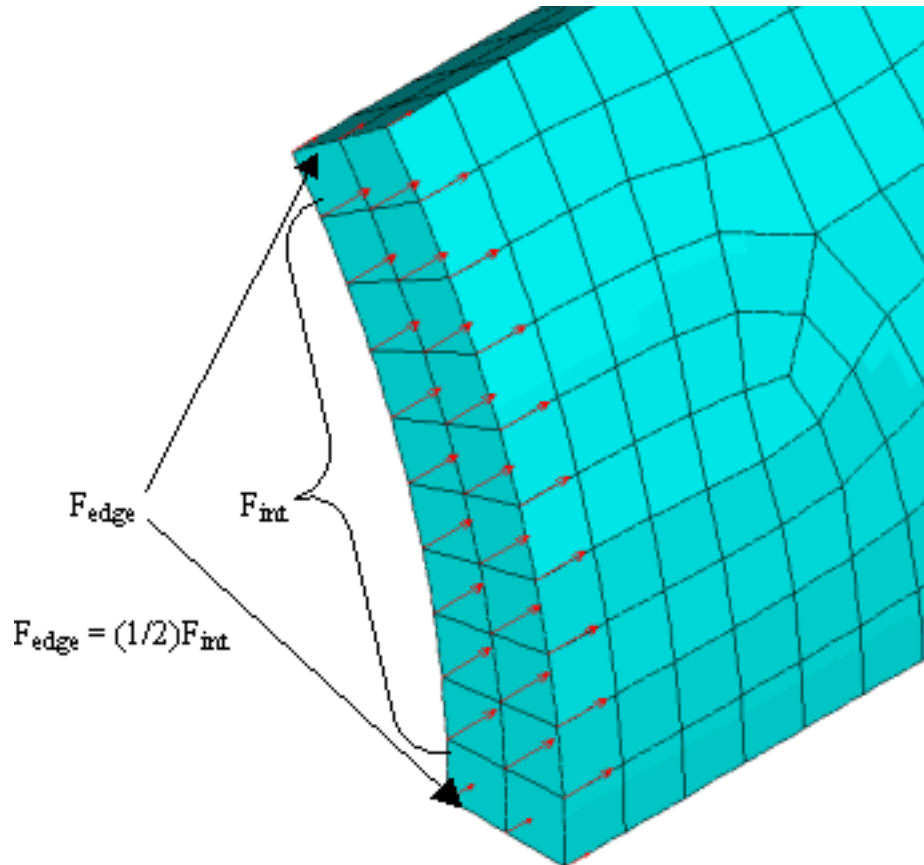


Figure 4. Sector forces.

Solve. Make sure you hold on to the .emat and .esav files.

Eigen Buckling Solution

Exit and re-enter the Solution processor. Change the analysis type to Eigen Buckling (Solution > Analysis Type > New Analysis). Specify the number of eigen buckling modes to extract (Solution > Analysis Type > Analysis Options) and expand the same number of modes (Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes).

Set the harmonic index range. As with a modal cyclic symmetry analysis, you have to specify the harmonic indices for which to perform the cyclic symmetry buckling analysis via CYCOPT,HINDEX (Solution > Solve > Cyclic Options). Harmonic index 0 corresponds to radial or tangential buckling. Bending buckling modes may be found under harmonic index 1. However, you may have a more complex mode shape with a much lower load factor than what you obtain with

The Focus



A Publication for ANSYS Users

harmonic index 0 or 1, so be sure to specify a large enough harmonic index range to capture this response. For example, the cylinder modeled here has a minimum load factor of 533873, which corresponds to a harmonic index of 2 as shown in Figure 5.

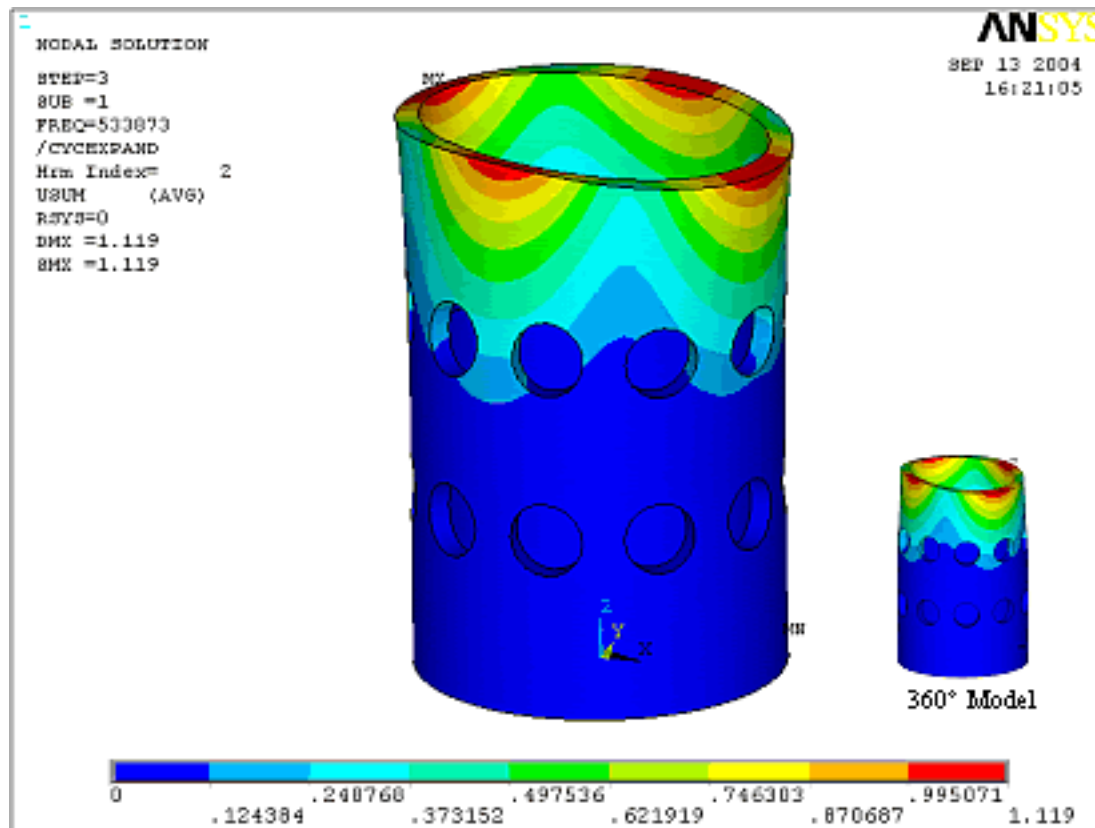


Figure 5. Minimum eigen buckling load factor = 533873 (360° model inset).

Postprocessing

As with modal cyclic symmetry, you'll want to expand the results to a full 360° display using /CYCEXPAND,ON (General Postproc > Cyclic Analysis > Cyc Expansion). You can list and read results grouped by harmonic index (General Postproc > Read Results > By Pick) or in order of load factor (General Postproc > Read Results > By Pick (Freq ordered)). My preference is to list results in order of load factor, so that I can pinpoint the minimum value without too much trouble.

Figures 6 through 8 compare cyclic symmetry versus full 360° results. Note that the calculated load factors are identical. The only difference in some instances is where the structure buckles "this way" for a cyclic symmetry model versus "that

The Focus



A Publication for ANSYS Users

way” for a full 360° model.

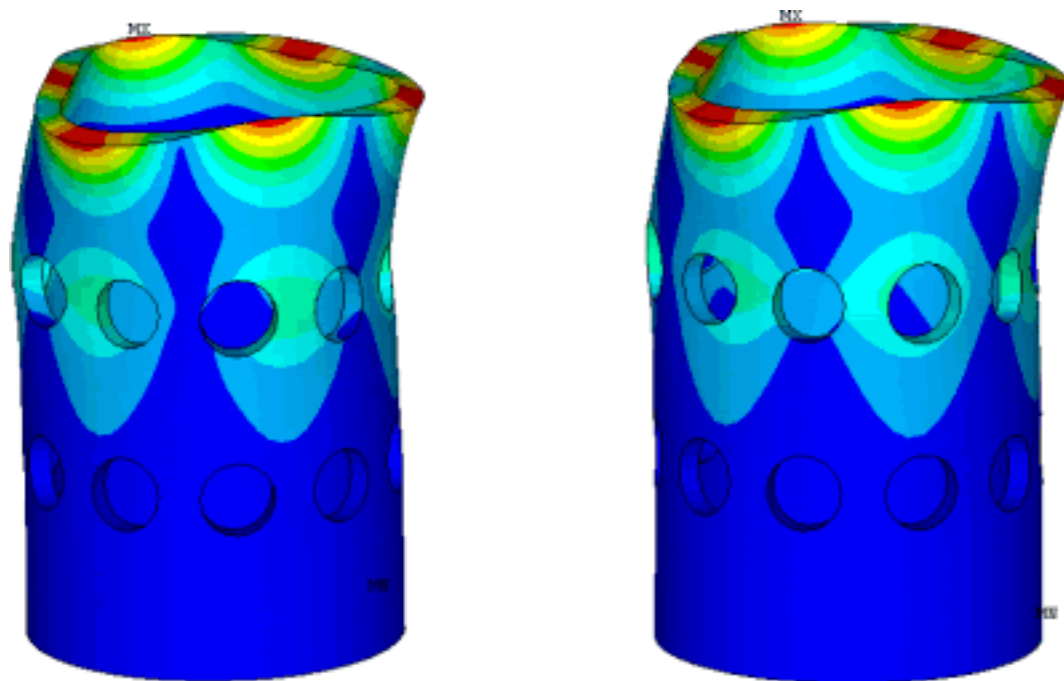


Figure 6. Load factor = 820225 (harmonic index = 3).

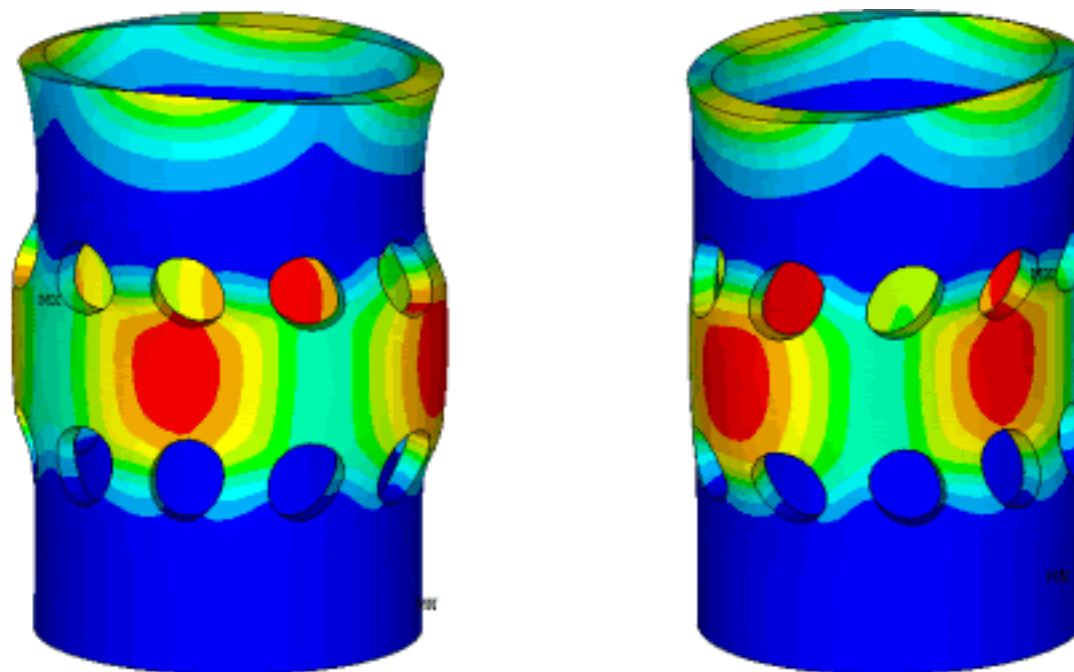


Figure 7. Load factor = 887618 (harmonic index = 2).

The Focus



A Publication for ANSYS Users

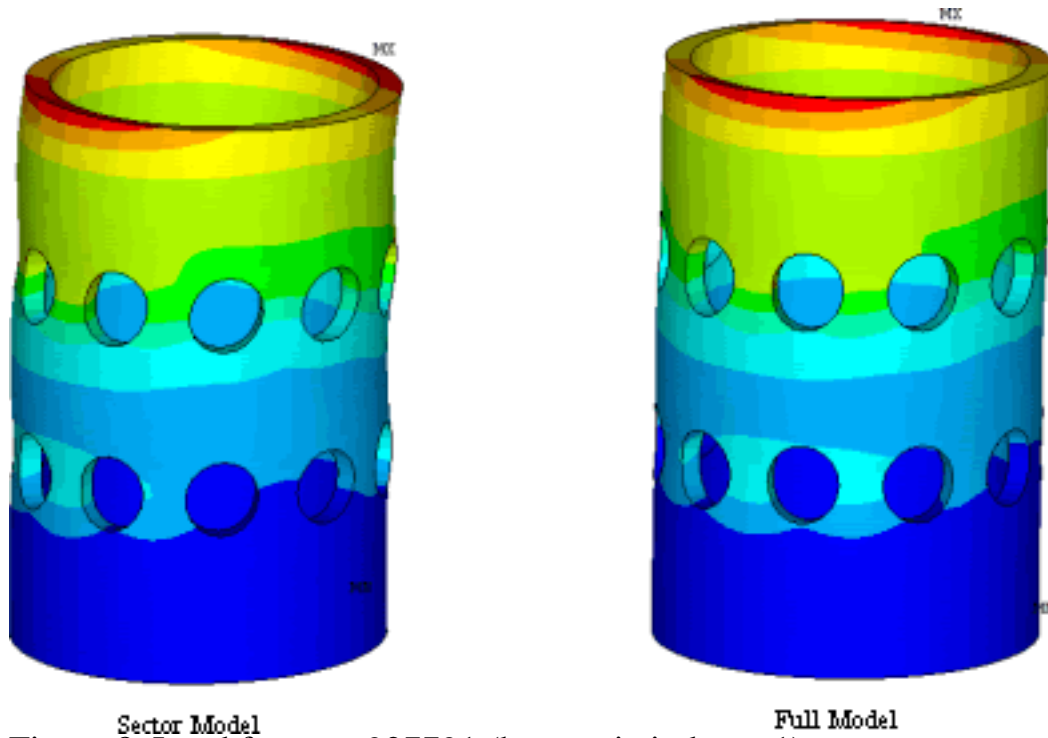


Figure 8. Load factor = 927791 (harmonic index = 1).

Conclusion

Tests indicate that eigen buckling cyclic symmetry is a valid approach as long as the geometry, material, and load requirements have been met. One must take care, however, to specify a large enough harmonic index range so that the minimum load factor for the full 360° structure is extracted.

[Cylbuckle.mac](#) [right click to download] was used for eigen buckling symmetry testing and to create the figures in this article. If you wish to execute this script after downloading, execute `cylbuckle` (no arguments) to perform an eigen buckling cyclic symmetry analysis on a single wedge of the structure. Execute `cylbuckle,1` to perform an eigen buckling analysis on the full 360° structure.

The Focus



A Publication for ANSYS Users

Are You an APDL Guru?

The last installment of the APDL guru certainly elicited more response from you, the reader, than did the first. Perhaps it was the extraordinarily corny rhyme that piqued your interest. At any rate, the submissions were quite ingenious and many of you succeeded in creating a helical spring in less than ten lines of APDL. Most solutions resorted to using beam elements meshed along a helical line to approximate the spring, which certainly is a valid technique.



Patrick Tibbits; however, garners the honor this month of being crowned APDL Guru of the month. Patrick supplied, not one, not two, but rather three solutions each of successively shorter length to the problem. Each of the three solutions Patrick provided relied on a different element topology. He supplied a solution using solid elements, shell element and, of course, the aforementioned beam elements. The most compact solution Patrick provided, which used beam elements meshed along a line, weighed in at a paltry 5 lines of APDL. Kudos to Patrick!

```
csys, 1
k, (1:6), 10, (0:450:90), (0:25:5)
l, (1:5), (2:6)
et, 1, 3
lmesh,all
```

This month marks a slight course correction in the APDL Guru series. Gone are the lines of poetry along with rather impractical puzzles. From this point forward, we will endeavor to supply more open-ended problems that hopefully have at least a twinge of practicality. Our hope is that we can collect novel, robust APDL scripts that demonstrate techniques for solving real-world modeling problems that we can then provide to the community as a whole.

To this end, we present the following puzzle for your enjoyment.

The Focus



A Publication for ANSYS Users

This Month's Puzzle

Occasionally in the life of an analyst, there are times when one would like to have available for later use deformed geometry that is obtained from the deflected shape of a structure subjected to some loading scenario. We know that obtaining displacement data for nodes within a solved FE model is a trivial exercise. However, it is somewhat more complex to actually generate new ANSYS geometry that conforms to the deformed shape of a structure.

Therefore, we present the following open-ended problems related to geometry creation from nodal displacement or location data. The problems are sorted in increasing order according to their perceived level of difficulty:

1. Given an ANSYS model consisting of multiple arbitrary 2D planar areas (assume the model can contain various regions in contact), construct the analogous deformed geometry areas from displacement data obtained from a particular solved load step. You can use the original geometry for connectivity/topology information. Does your solution require that you make any assumptions about the original geometry? Can you properly handle curved line segments using splines?
2. Given an ANSYS model consisting of 3D volumes, construct the analogous deformed line geometry from a particular solved load step. Again, you can use the original geometry for topology reference.
 - a. Constructing arbitrary, three-dimensional non-planar surfaces in ANSYS is notoriously difficult if not often impossible. As an added bonus, provide a flag to your macro that will construct any surface of the deformed geometry that is bounded by 3 or 4 lines in the original geometry. The idea is that any partial geometry created by this macro could be exported to a separate package for final processing. If you do create such surfaces, can you provide any qualification as to how well they conform to the deformed shape?
3. Given an ANSYS model consisting of multiple, arbitrary 2D element regions, with no associated geometry (assume the mesh is detached and the geometry has been deleted) construct ANSYS geometry that conforms to the mesh. Try not to rely on separate "parts" within the mesh containing unique element types, material property data, etc. Rather, use the mesh connectivity to determine unique parts.



A Publication for ANSYS Users

- a. Provide as input to your macro a tangency tolerance that is used to determine what constitutes a “sharp corner” in the mesh. You can use this as a means for determining where you should terminate lines.
- b. See if you can construct an algorithm for detecting fillets and other features that might not be properly handled by the “tangency tolerance” mentioned above.

Send your answers to apdl.guru@padtinc.com.

The Focus



A Publication for ANSYS Users

Upcoming Training at PADT

A complete schedule of the training courses for the next three months is shown below. [Learn more](#) about how the **Training Services** offered by PADT can save you time and money. Or, feel free to drop an e-mail to our training coordinator, [Ted Harris!](#)

Month	Start	End	Course	Description	Location
Sep '04	9/30	9/30	108	Paramesh	Tempe, AZ
Oct '04	10/4	10/6	101	Introduction to ANSYS, Part I	Albuquerque, NM
	10/7	10/8	100	Engineering with Finite Element Analysis	Tempe, AZ
	10/14	10/15	203	Dynamics	Irvine, CA
	10/18	10/19	104	ANSYS Workbench Simulation - Intro	Albuquerque, NM
	10/25	10/26	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	10/25	10/27	902	Multiphysics Simulation for MEMS	Tempe, AZ
	10/27	10/28	604	Introduction to CFX	Tempe, AZ
	10/28	10/29	107	ANSYS Workbench DesignModeler	Tempe, AZ
Nov '04	11/1	11/3	101	Introduction to ANSYS, Part I	Albuquerque, NM
	11/4	11/5	301	Heat Transfer	Irvine, CA
	11/8	11/10	401	Introduction to Electromagnetics	Tempe, AZ
	11/11	11/12	202	Advanced Structural Nonlinearities	Tempe, AZ
	11/18	11/19	102	Introduction to ANSYS, Part II	Tempe, AZ
	11/30	11/30	106	ANSYS Workbench DesignXplorer	Tempe, AZ
Dec '04	12/6	12/8	101	Introduction to ANSYS, Part I	Irvine, CA
	12/13	12/14	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	12/15	12/15	105	ANSYS Workbench Simulation - Advanced	Tempe, AZ
	12/16	12/17	604	Introduction to CFX	Tempe, AZ
	12/16	12/17	801	Introduction to APDL	Irvine, CA

The Focus



A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.