The Focus

PADT

A Publication for ANSYS Users

Contents

Feature Articles

- Anti-Symmetry (and Beyond)
- NLDIAG – Plotting Residuals
- Are You an APDL Guru?

On the Web

- Benchmark Your Computer's ANSYS Speed
- PADT Seminar: ANSYS 9.0 Update

Resources

- PADT Support: How can we help?
- Upcoming Training at PADT
- About *The Focus*
  - *The Focus* Library
  - Contributor Information
  - Subscribe / Unsubscribe
  - Legal Disclaimer
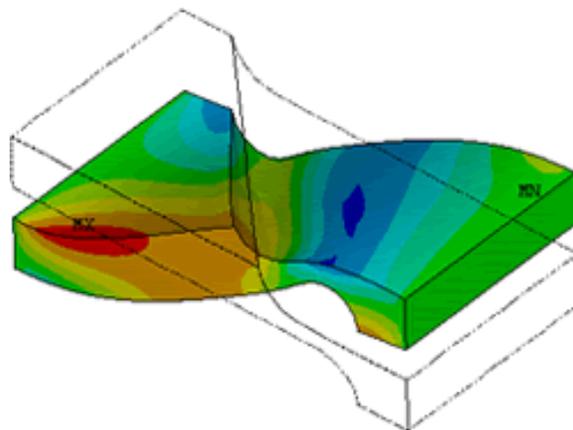
ANSYS
Support Distributor

A Publication for ANSYS Users

Anti-Symmetry (and Beyond)

by Rod Scholl, PADT

So think you know everything about symmetry? How would you take advantage of symmetry here? It has fixed displacment at the ends.
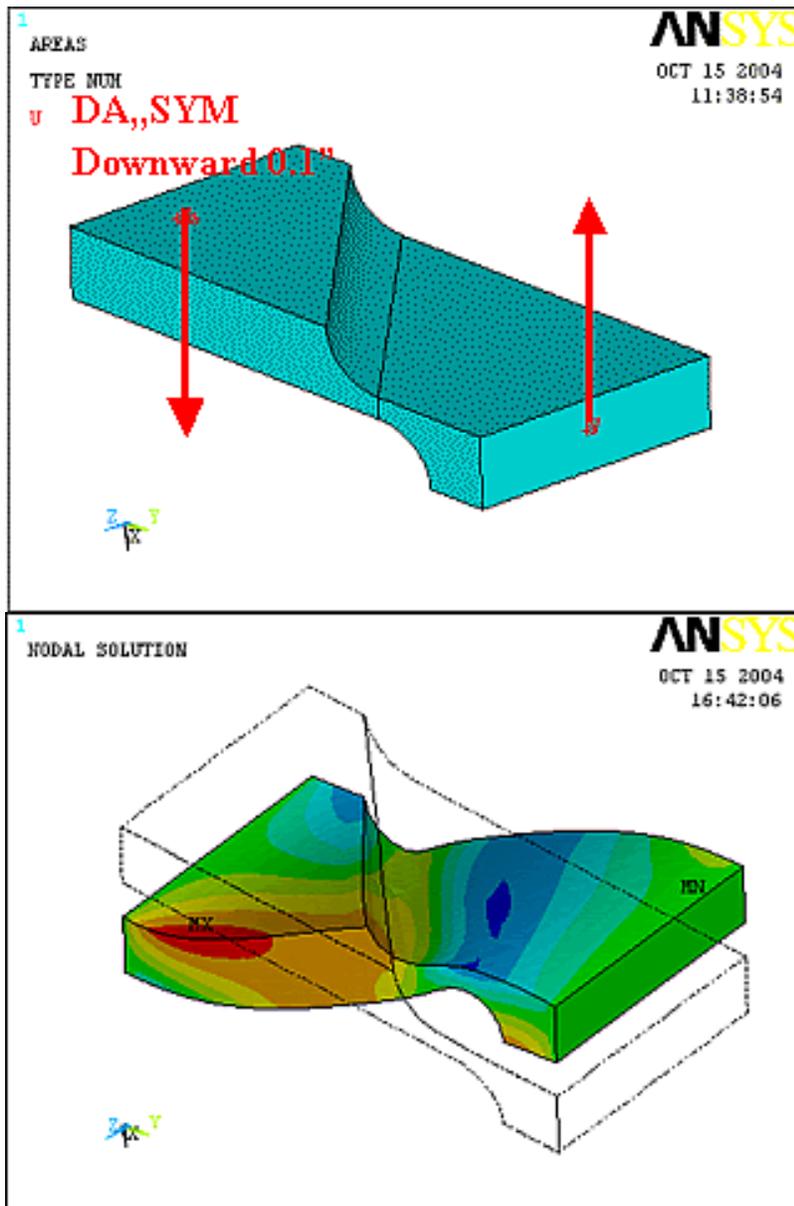
I think it is clear that there is some degree of symmetry here, but the plane of geoemtric similarity is not aligned with the plane of BC similarity. But let's start simple, and build to that case.

The Focus

A Publication for ANSYS Users

Using the DA,,SYMM or DA,,ASYM command

First, for the shown 3-D bock with 45° rotation, in order to apply a fixed end BC, you can save some keystrokes by knowing the following are equivalent commands.
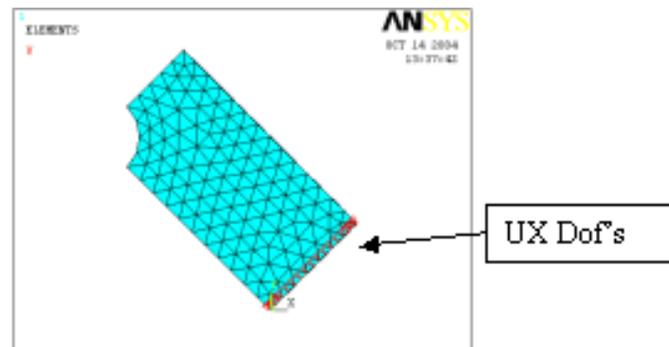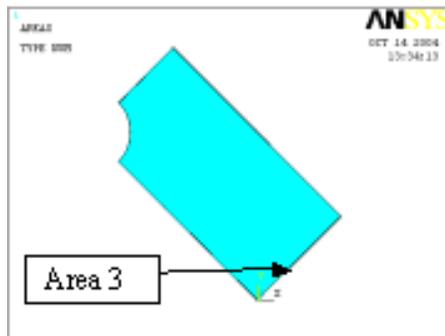
WPROTAT, -45 !Rotate WP
CSYS,4 !Switch to WP CS
ASEL,s,,,3
NSLA,s,1
NROTATE,all
D,all,ux,0
CSYS,0 !Return to global CS
WPROTAT,,,-45 !Return WP

DA,3,SYMM !That's it!

Note that the symmetry BC looks like a bunch of S's around the boundary when plotted. If this doesn't give you enough comfort, try SBCTRAN to move it to the nodes and view the little arrows (I usually do).

But What About That Whole Anti-Symmetry Bit?

Anti-symmetry, via a DA,Area,ASYM command, places the DOF's on everything except the normal direction of the area. (Table below is valid for solid 3-D structural DOF's).

	SYMM	ASYM
Normal	3-D	3-D
X	UX	UY,UZ
Y	UY	UX,UZ
Z	UZ	UX,UY

So, an example:

The Focus

A Publication for ANSYS Users
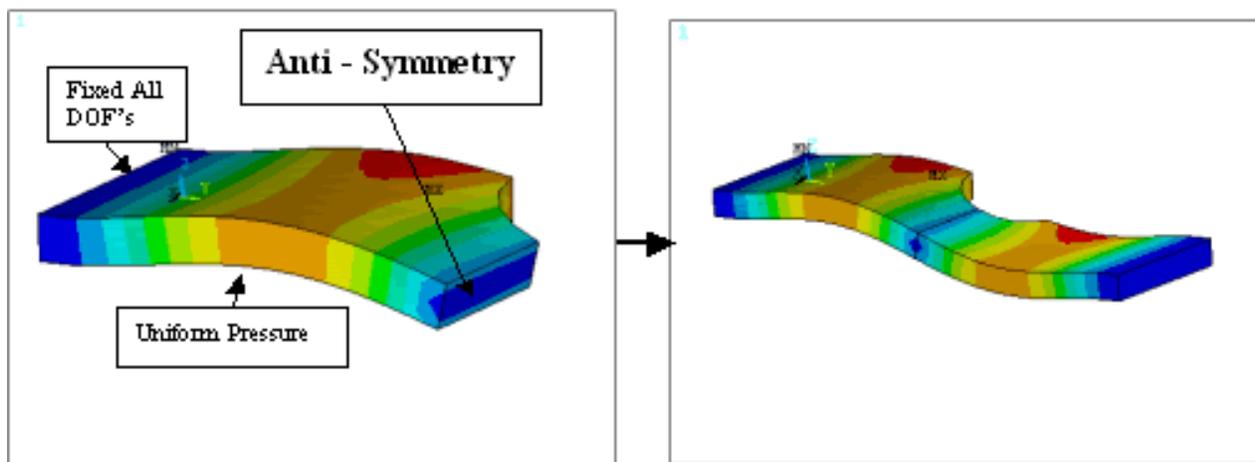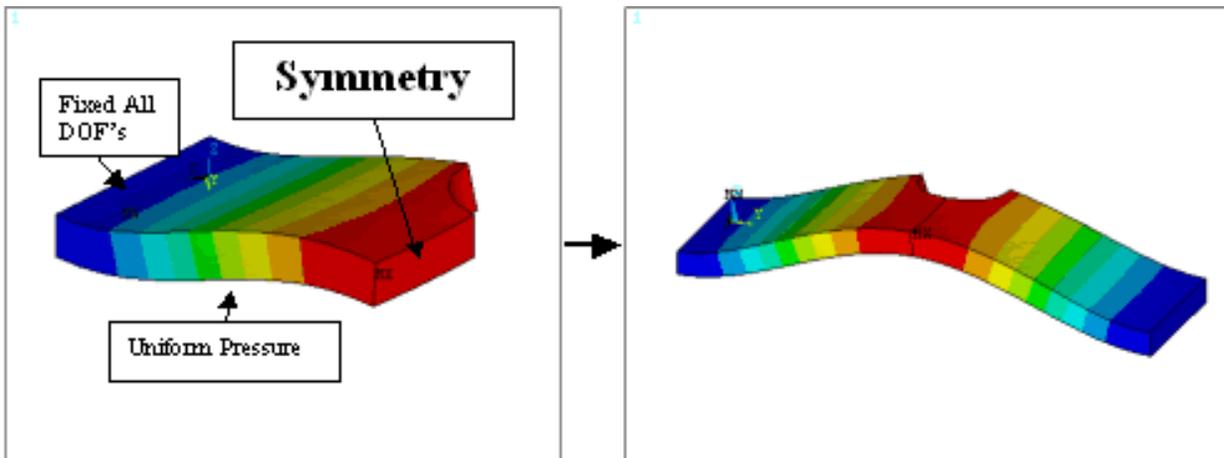
See how the geometry is reflected, but the loads are reversed?
[*Download the macro*: antisym.mac]

Beyond Anti-Symmetry

Occasionally, simplifications are available in not so-straightforward symmetries. For example, how about this case:

The Focus

A Publication for ANSYS Users
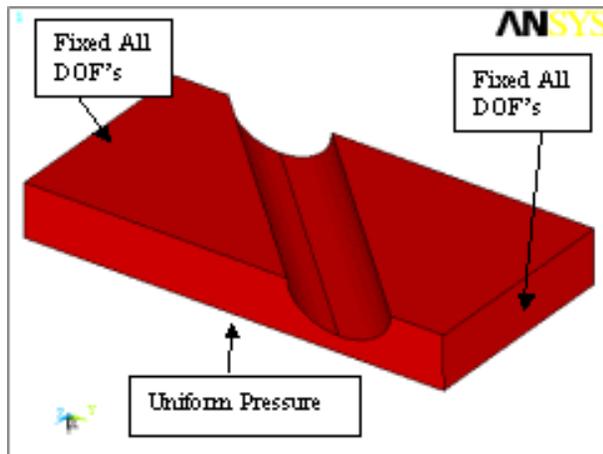
The vertical displacement solution looks like this:
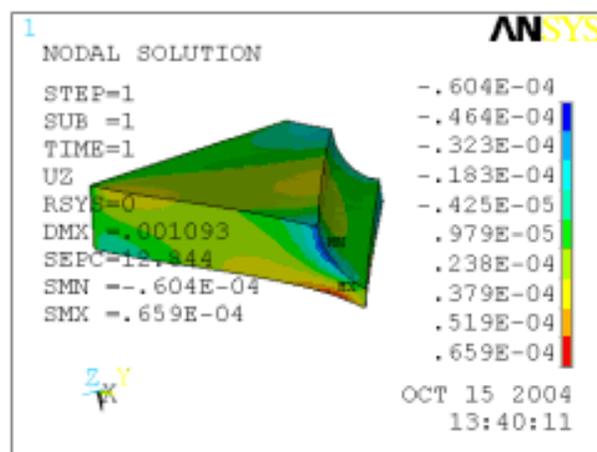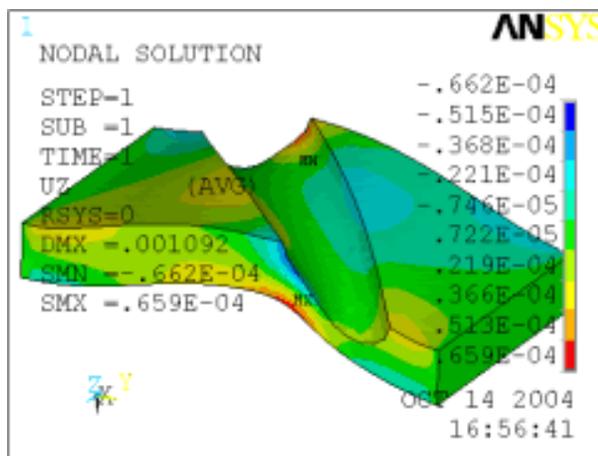
We can definitely see the result's symmtery, but the geometric symmetry doesn't match the BC symmetry.

The symmetry condition can be enforced using many possible cuts, but perhaps it is easiest to visualize if we choose to make our cut down the center of the groove. Next, the nodes on opposite ends are constraint equation'd to its brother, as shown below. Note that deflections in the normal and transverse direction (X and Y in this case) are made to be equal and opposite, whereas the vertical deflections (Z) are held to be equal. The guts of the script is shown for reference.

The Focus

A Publication for ANSYS Users
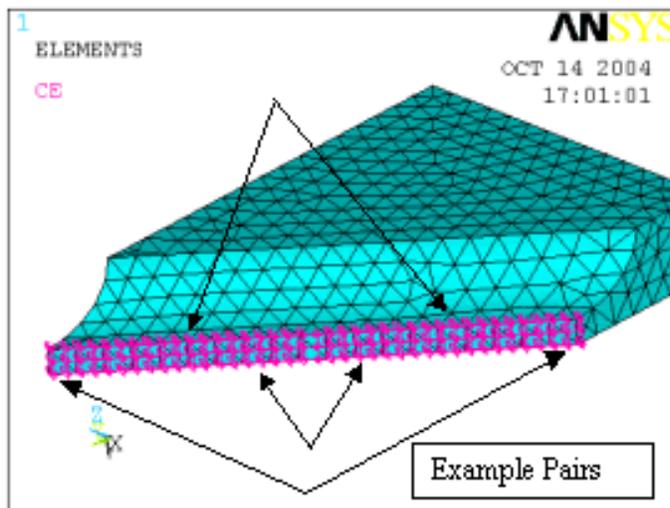
Example Pairs

```
*get,ncount,node,,count
a=0
*do,icount,1,ncount
cmsel,s,n2      !left side node set
a=ndnext(a)
cmsel,s,n1      !left and right side node set
CE,(icount-1)*3+1,0,a,ux,1,node(nx(a),-ny(a),nz(a)),ux,1
CE,(icount-1)*3+2,0,a,uy,1,node(nx(a),-ny(a),nz(a)),uy,1
CE,(icount-1)*3+3,0,a,uz,1,node(nx(a),-ny(a),nz(a)),uz,-1
*enddo
```

The results are identical at a location very near the boundary, give or take 0.3% due to the discretization and constraint equations. Note that peak strains are within 0.07%.
[*Download the macro*: bysym.mac]
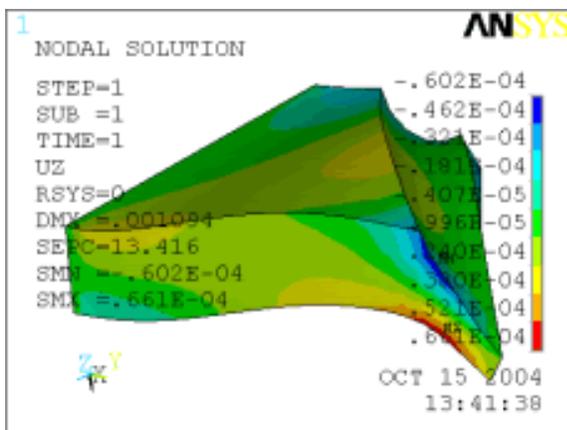
The Focus

Inverted-Reversed Symmetry (or Whatever You Want to Call It)

Once you get the hang of it, the above script requires only changes to some +'s and –'s to model geometries like the one below using symmetry (only works for some BC scenarios).
[*Download the macro*: breversed.mac]
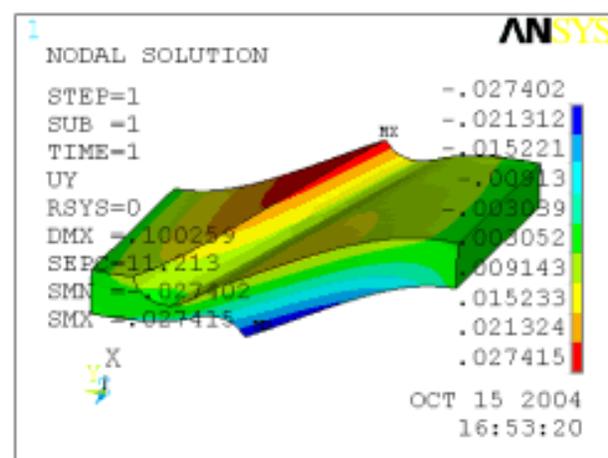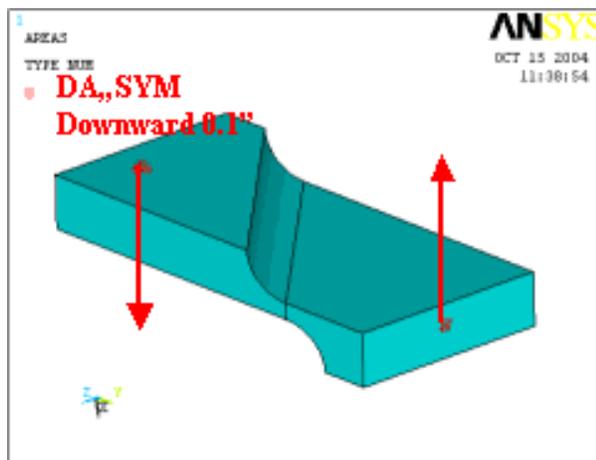
This time, the nodes on opposite ends and oppoiste vertical sides are constraint equation'd to its brother as shown below. Note that deflections to the left and right (X in this case) are made to be equal, as are the deflection in the vertical direction (Y) held equal, whereas the transverse vertical defelctions (Z) are held to be equal and opposite.

Example Pairs
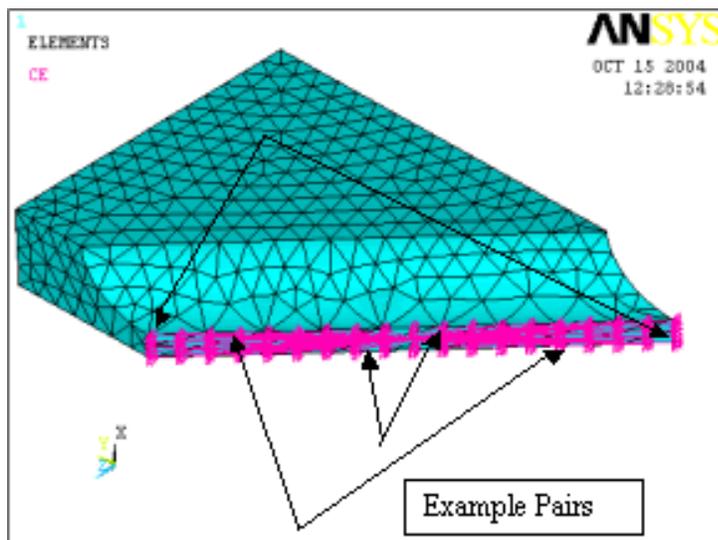
The new script guts look like this:

The Focus

A Publication for ANSYS Users

```
*get,ncount,node,,count
a=0
*do,icount,1,ncount
cmsel,s,n2      !left side node set
a=ndnext(a)
cmsel,s,n1      !left and right side node set
CE,(icount-1)*3+1,0,a,ux,1,node(nx(a),-ny(a),-nz(a)),ux,1
CE,(icount-1)*3+2,0,a,uy,1,node(nx(a),-ny(a),-nz(a)),uy,1
CE,(icount-1)*3+3,0,a,uz,1,node(nx(a),-ny(a),-nz(a)),uz,1
*enddo
```

It's also interesting to note that athough the cut is shown down the diagonal center, cuts in any vertical plane would work as long as they captured half the geometry!!! This can be important because sometimes, as in this case, the peak stress is expected at the center of the groove – right where we *don't* want a bunch of CE's.

```
1                                    ANSYS
  NODAL SOLUTION
  STEP=1                     -.010559
  SUB =1              mx    -.006339
  TIME=1                    -.002119
  UY                         .002101
  RSYS=0                     .006321
  DMX =.100259               .010541
  SEPC=11.683                .014761
  SMN =.010559               .018982
  SMX =.027422               .023202
                             .027422
                       OCT 15 2004
                          16:55:06
```
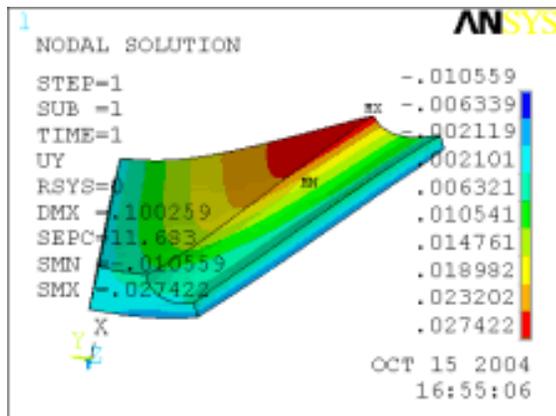
You might encounter symmetries more often than you think!

The Focus

A Publication for ANSYS Users

NLDIAG – Plotting Residuals

by Rod Scholl, PADT

One of the features added in ANSYS 7.0 has almost eliminated the guess-work required on many convergence problems via the NLDIAG command. Many users are still unaware of it.

NLDIAG,NRRES,ON creates a file that stores the Newton-Raphson residuals everywhere within the model (and has to be issued before the SOLVE). Thus, after a failed convergence run, as long as you have one successful iteration (you don't even need a substep or loadstep), you can enter the post-processor and type PLNSOL,NRRES and visually see where the residuals reside. Generally, I get a plot with solid blue except for one hot-spot. This hot-spot is where the convergence is sticking. Now you just have to figure out how to fix it!

Problems?

Since this capability was not built into ANSYS from the ground up, it has been added on top of ANSYS using many of the same customization tools that users have access to. We've been using it around here for a while and have developed a few suggestions that we have found useful.

The most important thing to remember is that these results are not stored in the *.RST file so before you can look at them you need to give ANSYS something to write on top of. Make sure you always issue a SET command before you plot. If you didn't get an *.RST from your run, no problem. Just point to any *.RST file. The values in the *.RST file don't even need to be correct or from the same model, you just need the SET command to create the data structures in memory that ANSYS needs to plot.

On a related note, ANSYS always scales by the active set of displacements when you issue any PLNSOL command. Things can look pretty strange if you plot residuals on top of a deflected object, especially if the results are from an un-converged solution or a dummy *.RST file, so it is a good idea to do a /DSCALE,1, OFF first.

Lastly, if you do use a bogus or dummy *.RST file, don't issue a SAVE command. This will write the bogus information to your *.DB and you may not like the results.

The Focus

A Publication for ANSYS Users

And just in case you don't know what those residuals are, or what I'm going on about:

When ANSYS is solving iteratively because of a non-linearity, such as stiffness matrix updates for plastic material properties, it checks how equal one side is to the other in the equation F=KX (for example). The residuals are how "unequal" each term in F is to each term in KX. These "residuals" are used as the convergence criteria (usually related to sum of all the squared residuals). They are what is shown in the GST (grahpical solution tracking) screen during a solve (see the GST Plot article from an earlier issue of *The Focus*). When the sum of errors is deemed small enough, the F=KX is considered balanced (enough) and the solution considered converged (enough).

So, give this great little utility a try next time you are having convergence problems. Once you understand how it works (overlays results on existing results), it is a pretty simple tool to use and will save you hours of trying to track down a problem.

The Focus

Are You an APDL Guru?

Last month's puzzle presented a series of open ended problems related to creating deformed geometry in ANSYS; that is, geometry that conforms to the deformed shape of a structure subjected to a given set of loads. No one questions the fact that this is a notoriously difficult problem, which is precisely why it was posed to you the reader as an opportunity to flex your APDL skills. Sadly, the email inbox remained empty…
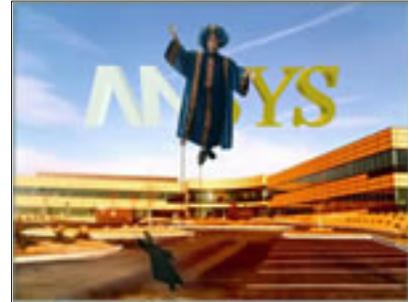
However, included in this month's edition is a macro that creates three-dimensional line geometry from a deformed solution. The macro has been tested on a few different geometries with acceptable performance; however, it has not undergone rigorous testing. So, use it at your own risk. Hopefully it will serve as an encouragement or perhaps inspiration for some intrepid APDL guru to expand upon and provide a robust solution to the community at large. If you do expand upon the macro, let us know and we'll be sure to give you special recognition, if your contribution is an enhancement. Also include is a driver macro as well as a link to a small ANSYS neutral file that contains test geometry.

Downloads: df_test.mac • dfgeo.mac • test.anf

This Month's Puzzle

This month we continue with the geometry creation and modification theme. A macro that performs one or more of the following operations will be considered for acceptance as the work of a true APDL guru.

Here is the scenario. I have a number of straight lines in the XY plane. I would like a macro that does the following:

1. Allows me to select any two lines, but only two.

2. If the two lines intersect, they are both divided at the intersection point and the resulting four lines are connected such that they share a common keypoint at the intersection point.

3. If the lines do not intersect, the first line is extended until it is tangent to the second line.

The Focus

The macro should be capable of handling the special case in which the second line needs to be extended in order for intersection to occur. If this case should occur, the macro should extend both lines to a common intersection point and then join the two lines together. However, if the second line does not require extension, then the macro should divide the second line at the intersection point and join the two pieces of the original second line together with the first line, which has been extended.

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

The Focus

PADT

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
Oct '04	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
Jan '05	1/10	1/12	101	Introduction to ANSYS, Part I	Tempe, AZ
	1/17	1/18	104	ANSYS Workbench Simulation - Intro	Irvine, CA
	1/19	1/19	105	ANSYS Workbench Simulation - Advanced	Irvine, CA
	1/20	1/21	801	Introduction to APDL	Tempe, AZ

The Focus

A Publication for ANSYS Users

Whether it s one of our regularly-scheduled classes at our training facility, or a customized class tailored to your specifications and location, you can be assured that the training you receive will have immediate, positive results on your capabilities in design, analysis, and product and process improvement.

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.