



A Publication for ANSYS Users

Contents

Feature Articles

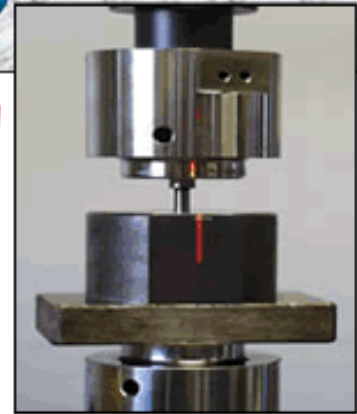
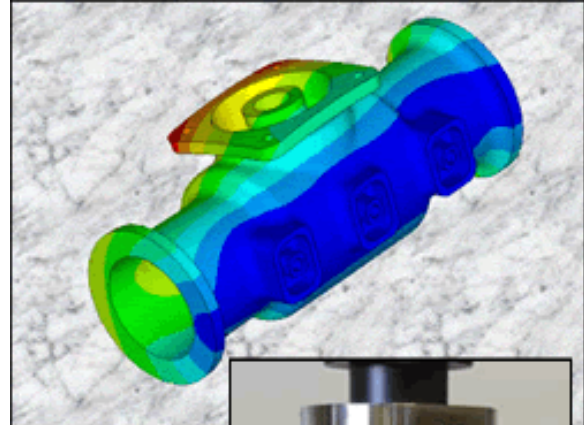
- [Tools of Post-Processing: Plotting to Files](#)
- [ANSYS, Elastomers, and the Physical Test Lab](#)
- [Tcl/Tk in ANSYS](#)

On the Web

- [Ohio CAE's Soft Words Newsletters](#)
- [PADT Seminar: ANSYS 8.1 Update](#)
- [Southwest ANSYS Users Newsletter](#)

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

Tools of Post-Processing: Plotting to Files

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. Spending a little bit of time to learn how to produce more concise and digestible results is a good investment. To this end, we will be covering a couple of key ways to produce higher quality results using POST1 and POST26 including listing, animation, sections, and for this second article, plotting to files for use outside of ANSYS.

The creation of color contour plots is the primary way in which analysts convey result information to other people. In most cases creating those plots with the default settings found in ANSYS is good enough, but with a little extra formatting, the quality and amount of information conveyed in this type of output can be greatly increased. There are many more options than can be covered here, so we will just highlight the most important options with a bias towards quickly creating high-quality output.

Before getting into plot creation, it is important to understand the mechanics of how ANSYS generates a plot. When you issue a plot command to ANSYS, it first creates a 3D set of colored polygons. These are then sent to some sort of post-processor that creates a 2D representation of the data as either an array of pixels (*bitmap*) or as 2D polygons (*vectors*). The algorithms that ANSYS uses to do this conversion, along with the options passed to it, have a huge effect on your plots.

Specifying the Plot Format

The most important command in plotting is /SHOW. It is used to tell ANSYS where and to store plots and some basic options for how to store them. This is true for plotting to the screen as well as to files. There are a ton of options so you should look up this command in the manual and really study it. The basics are:

/SHOW, Fname, Ext, Vect, NCPL

Fname is key to directing where and how the plot is made.

X11, Win32c, etc.

Plot to a terminal. See manual for different options.

TERM

Plot to the terminal devices last specified.

CLOSE

Purges the graphics buffer and closes any open graphics files.

FILE

Plot to the last graphics file specified.

OFF

Don't plot anywhere.

PSCR, HPGL, HPGL2, JPEG,

Plot to a file in the give format. PNG is your best bet for

TIFF, PNG, VRML

size/quality. Puts each plot into a *filenameNNN.ext*, where *ext* is the file extension for the given format and *NNN* goes from 000 to 999.

Filename

In ANSYS's own format to the given filename.

Vect specifies raster (0) or vector (1) mode.

This is not bitmap vs. vector, but rather filled vs. wireframe.

NCPL is the number of planes.

The Focus



A Publication for ANSYS Users

Set to 8 in most cases to get full color.

Use /show to direct your plots to files and to turn on full color. I recommend /show,png,,,8. This plots to the PNG format, which is highly compressed but non lossy like JPEG, and is a format that anyone with a web browser can read. The little fuzzy bits in the JPEG image in Figure 1 show the “lossy” nature of JPEG. The TIFF and PNG look the same but the PNG file is much smaller.

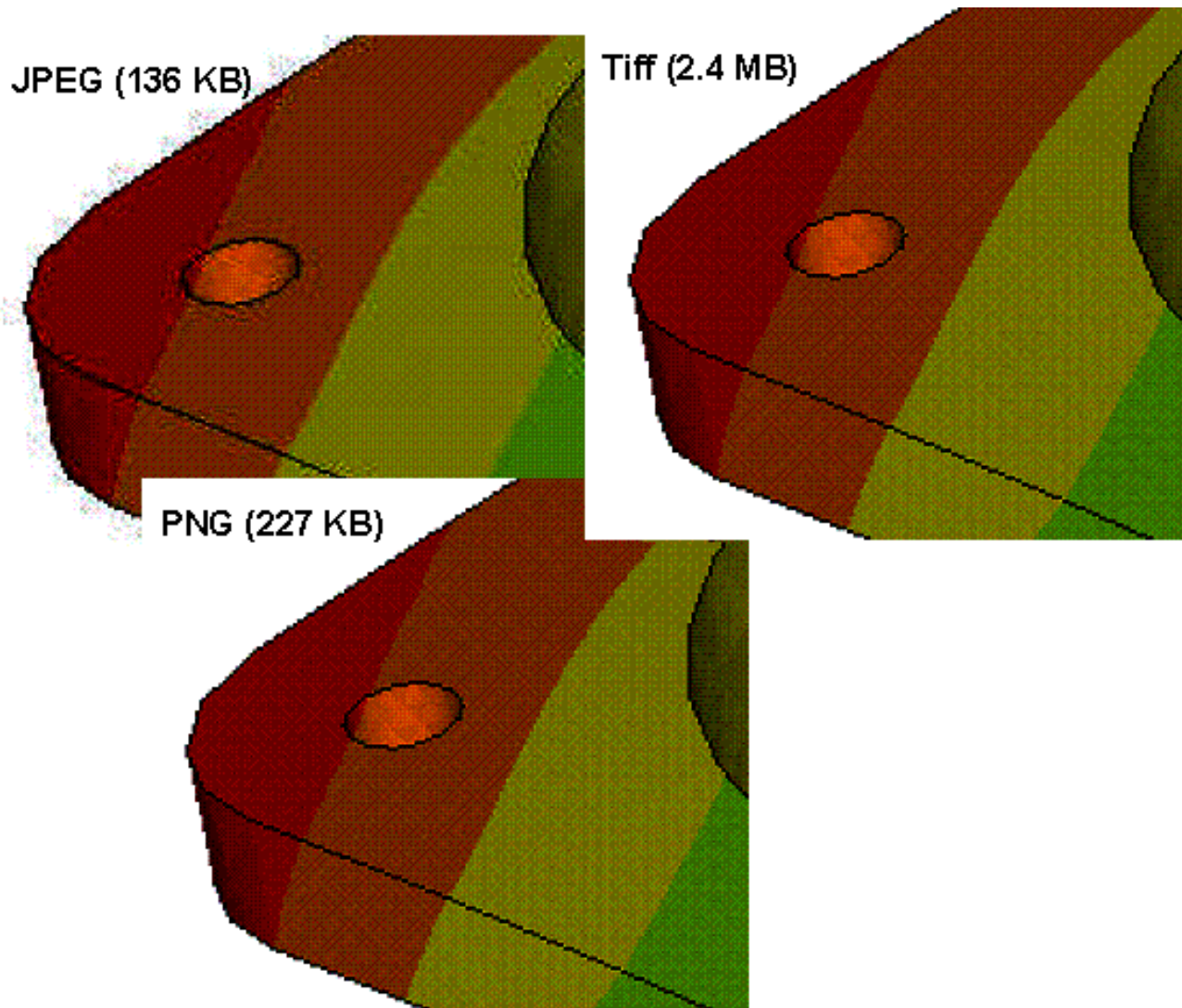


Figure 1. Bitmap file formats in ANSYS.

If you want a vector then try /show,pscr,,,8 and you will get polygons in an Encapsulated Postscript format. You can then use a program like Adobe Illustrator to make very high quality images that can be scaled without loss in quality.

Multiple Windows

The easiest way to increase the amount of information in a single plot is to use multiple windows with the /ERASE and /NOERASE commands. This allows you to plot elements with stresses, graphs with contours, or different modes/load steps all on one page. The way it works is that you

The Focus



A Publication for ANSYS Users

position and control on/off status of windows with, you guessed it, /WINDOW. By default ANSYS erases the “canvas” every time you issue a plot command, so you use /noerase to turn off that feature and then you can make multiple plots. Figure 2 shows a simple macro and an example of mode plots all on the same page. The bolded lines are the ones that affect the layout of the windows.

```

! Plots first 4 modes
/erase
/win.1.1top @/win.2.2top
/win.3.1bot @/win.4.2bot
/view,all,1,1,1,1
/wup,all,8
/angle,all,0
/dist,all,1.3
/focus,all,.2,.08,.05
/win.2.off @/win.3.off @/win.4.off
/plot,info,0
/annot,delete
/ISPPC, 15,0.700,1,0,0
^dim,axx,.4
^dim,any,.4
^dim,feq,.4
axx(1) - .98, .4, -.98, .4
any(1) - .01, .01, -.92, -.92

^do,i,1,4
set,1,i
^get,feq(i),mode,i,feq
plot,1,u,sum
/win.1.off
/win.1+1.on
/noerase
^enddo
^do,i,1,4
/Title,axx(i),any(i),Mode %i: %feq(i)% Hz
^enddo
/erase
axx
any
feq

```

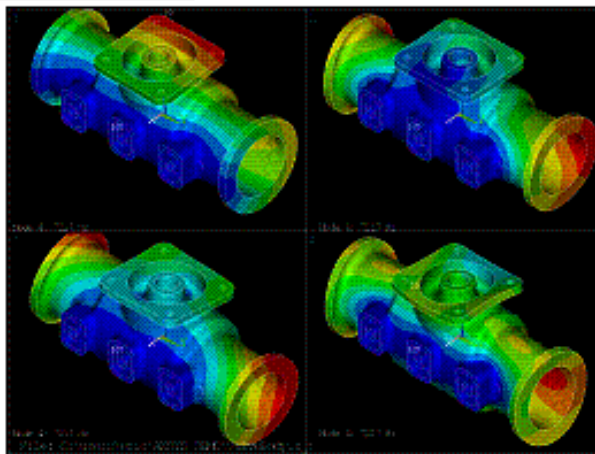


Figure 2. Creating multiple windows.

Reverse Video

As you can see in the previous example, by default ANSYS places a black background in plots and outlines things with white lines. This is preferred for the screen but uses a lot of ink when you print the plots. You can easily swap the black and white using the GUI command:

PlotCtrls -> Style -> Colors -> Reverse Video

The Focus



A Publication for ANSYS Users

If you make your plots using macros, then place the following in you macro:

```
/RGB,INDEX, 100, 100, 100, 0
/RGB,INDEX, 80, 80, 80, 13
/RGB,INDEX, 60, 60, 60, 14
/RGB,INDEX, 0, 0, 0, 15
```

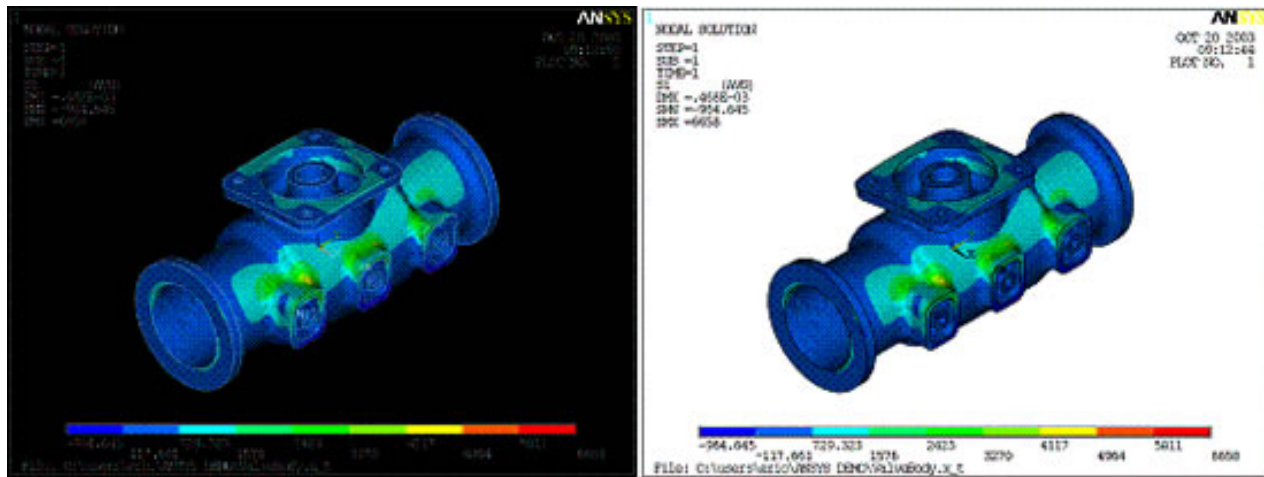


Figure 3. Reverse video.

Like everything in ANSYS, colors are done by numbers. By default, 0 is black and 15 is usually white. 13 and 14 are two shades of grey. The /RGB commands swap black and white as well as the two grey shades.

Quality

The biggest problem I have seen with typical plots is that the quality of the image is usually very low. The defaults for plotting in ANSYS are set up for the screen and often do not translate well into files for use in other applications. However, with a few commands and a little longer plotting times, the quality can be raised.

The Focus



A Publication for ANSYS Users

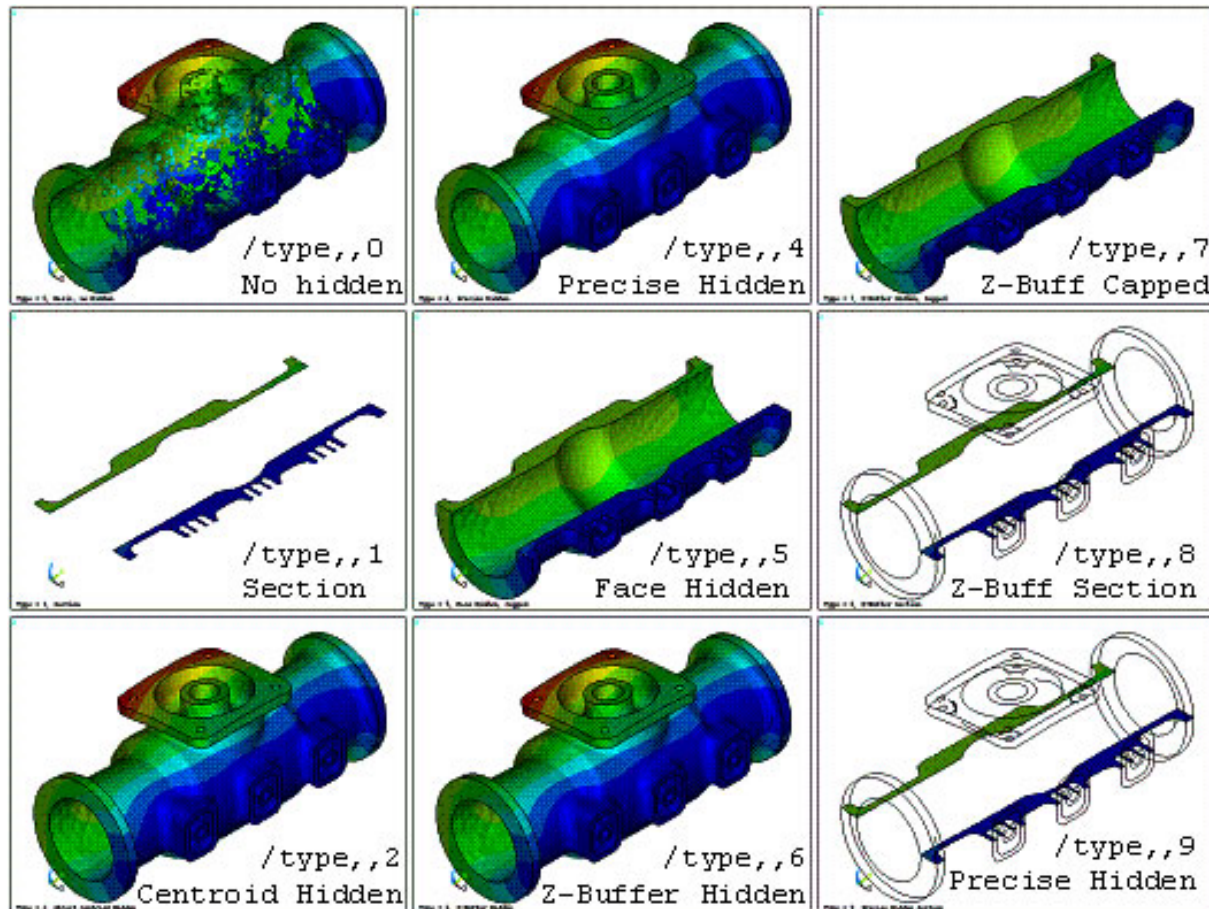


Figure 4. /TYPE options.

The most important command is /TYPE. It determines the hidden line or slicing algorithm ANSYS uses to make your plots. Figure 4 illustrates the 9 options available. The key issue with /TYPE is to pick an algorithm that does not use Z-buffering. Z-buffering is a very quick way to calculate hidden line removal, but the resolution of your screen is used to make the image. This can result in “chunky” plots. I use Types 4, 5, and 9, which uses Price’s hidden methods for each kind of plot.

Once a /TYPE is chosen, you should make sure Power Graphics are turned on (/graph,power). Many of the newer plotting routines only work with power graphics, so make sure it is enabled. One of the more important features that only works in this mode is greater facet resolution with /EFACET. By default, ANSYS will place flat triangles on element faces. If you set /EFACET higher, the curvature of your parts is shown and the shading is smoother. Figure 5 shows an example of how this option affects curvy parts. One thing to note is that increasing the number facets really slows down plotting because it increases the number of triangles the plotting algorithm must process. If you have midside nodes, you should always use at least /EFACE,2.

The Focus



A Publication for ANSYS Users

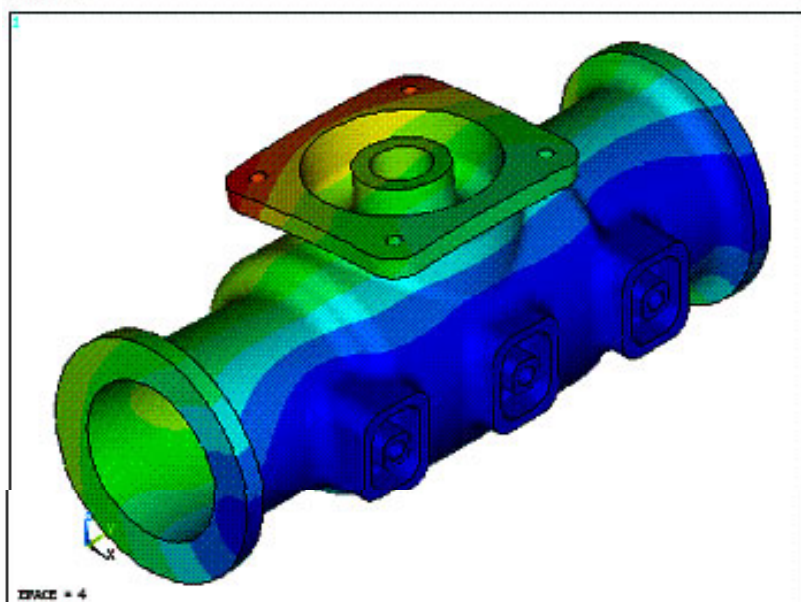
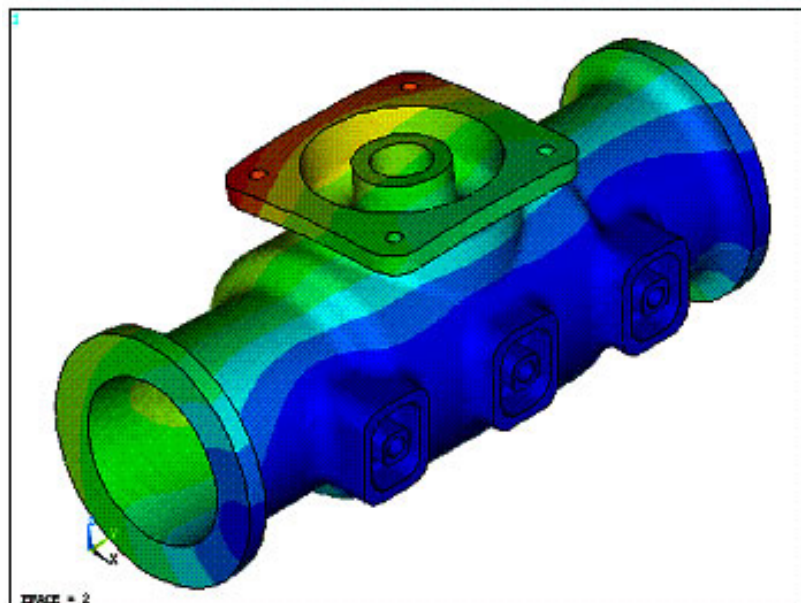
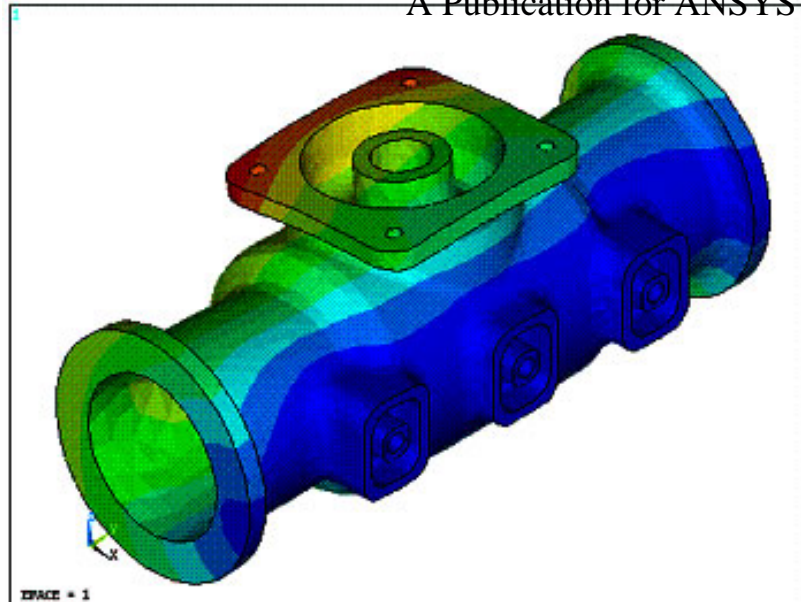


Figure 5./EFACET options.

The Focus



A Publication for ANSYS Users

The last quality knob that is commonly used is the resolution for the bitmap you are creating. By default, ANSYS creates PNG, TIFF, and JPEG files at around 800 pixels tall. Resolution is important because of scaling and aliasing. ANSYS does not get rid of jaggy edges (aliasing) but most MS Office products and image editors do when you scale images down. So the best way to get a nice looking plot is to make it around twice the final display size, and then scale it down. I generally use `/gfile,1200`. Figure 6 illustrates some examples.

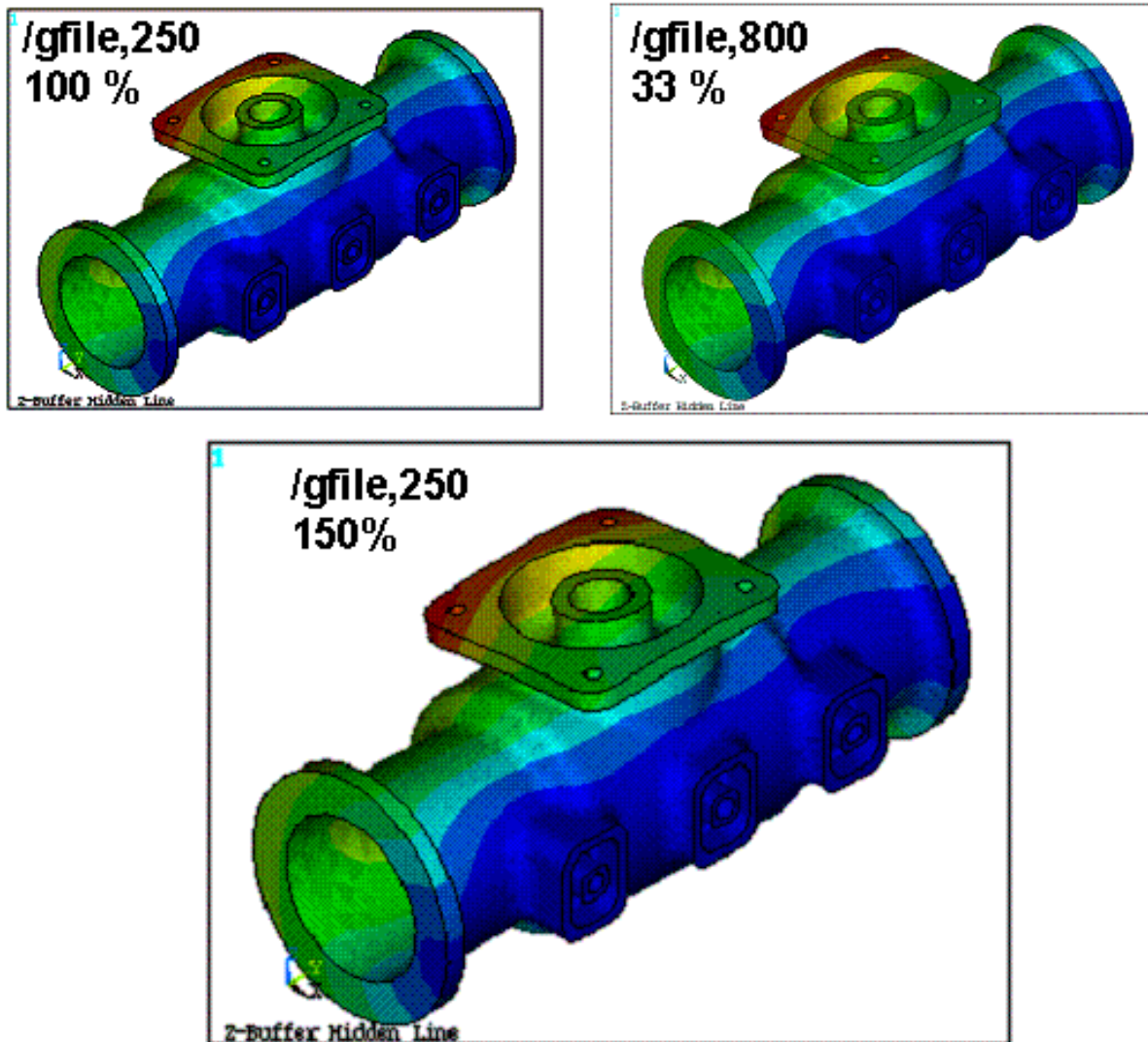


Figure 6. Resolution and scaling.

The Focus



A Publication for ANSYS Users

Decorations

The previous topics have focused on the plotting of results on the geometry. However, a large portion of the information in a plot is contained in the annotations and decorations around the geometry, such as contours, titles, information blocks, and annotations. The proper use of these options can greatly increase the value of the plot, and most importantly, allow you to place critical non-graphical information with the plot.

The standard borders, title, and legends are controlled with the /PLOPT command. There are too many options to list here, but take a look at the command in the manual and play with it. I like to turn off the view information with /plopt,leg2,off, the min-max symbols off with /plopt,minm,off, and turn on the jobname with /plopt,file,on. On a similar note, I usually move the coordinate system to the lower left hand corner with /triad,lbot or just turn it off altogether with /triad,off.

Another useful capability that most people don't use is the ability to add your own bitmaps to a plot. You use annotations to create them. It's a great way to add your own logo or pictures of the real hardware or a test. Figure 7 shows a typical usage of this command.

```
/txtre,file,66,padt_logo,bmp,,
/LSYM,1.39,-.99,0, 66, 0.000,1
/txtre,file,67,ur,png,,
/lSYM, .8,-.8, 0, 67, 0.000,1
```

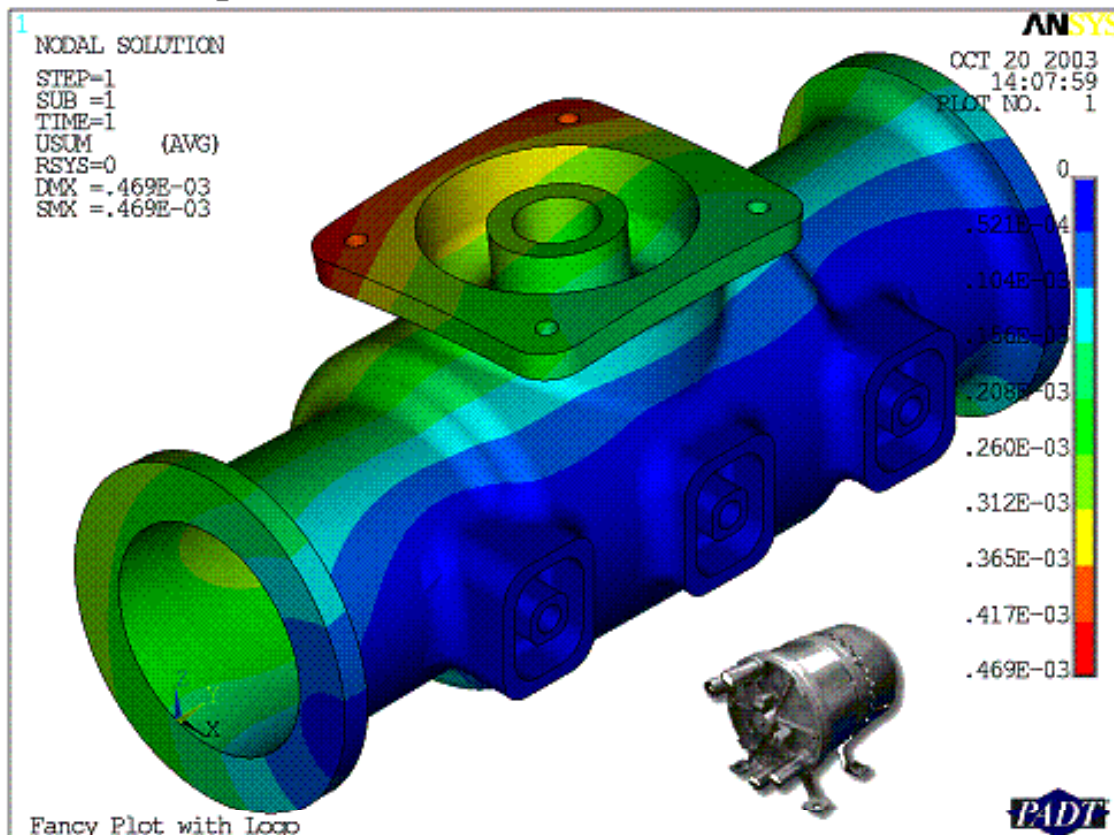


Figure 7. Bitmaps on plots.

The Focus



A Publication for ANSYS Users

Figure 7 is also an example of contours being in the wrong place. Recent versions of ANSYS have the option to not only move the contour, but also to have a unique contour in each window. /PLOPT,info,1 gives you the standard ANSYS contour on the left side of the window (first image in Figure 8) and /PLOT,INFO,2 gives you the new style that you can move around (second image) with /UDOC,WIN,CNTR,LEFT|RIGHT|TOP|BOTTOM. The third image shows two windows with two different contour styles.

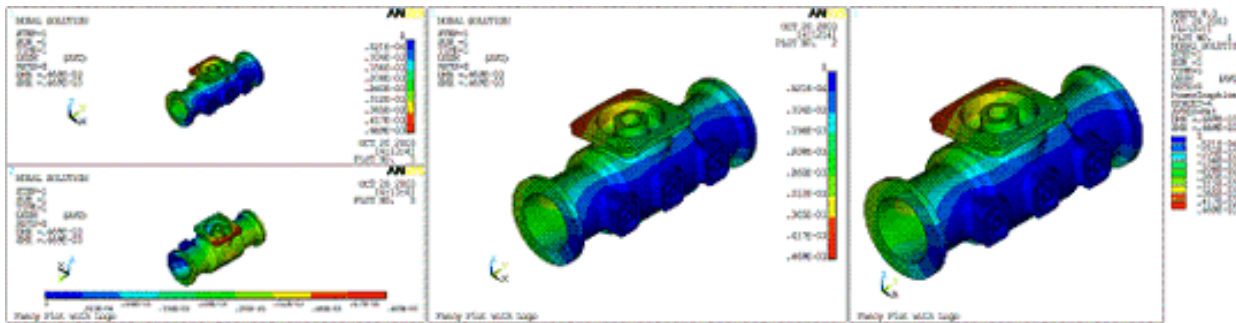


Figure 8. Contour style and location.

Speaking of contours, a lot of times the default ANSYS number format used in the contours aren't so great. With /GFORMAT,Ftype,Nwidth,Nsig you can specify a FORTRAN format to use. Unfortunately there isn't enough room, so the largest you can get is F12.9 or E12.9. I like to use the F format when I don't want scientific notation.

Annotation

ANSYS allows the user to place 2D graphics on top of their plots using the Annotation utility. The best way to use this is to create your annotation using the tool provided in the GUI, and then save the annotation to a file that you can place in a macro. At some point we will try and do a Focus article just on this topic, but it is too involved for this article. Figure 9 shows the most common uses of annotation: to point out a key location on the model with a 3D arrow, and to list loads and company information.

The Focus



A Publication for ANSYS Users

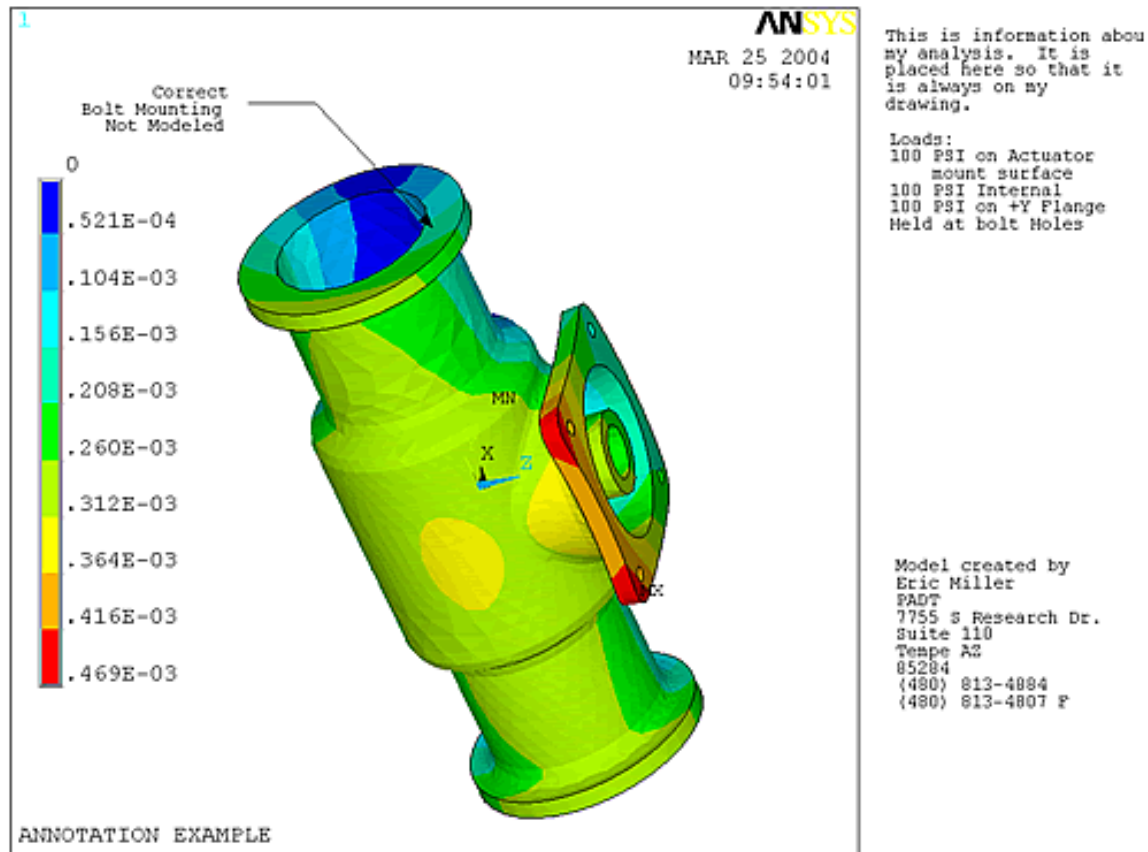


Figure 9. Annotation.

Fonts

The last topic for this article is the use of different fonts on your plots. Spending time on fonts is one way to get ridiculed for wasting time on making things pretty. However, sometimes the default Courier font is hard to read when scaled or copied, and something like the Arial font would be more readable. Use the GUI command PlotCtrls -> Font Controls -> Anno/Graph Font to specify your fonts. If you want it in a macro, then copy the /dev command from the log file. The tricky bit is getting the font description correct, so use the log file. The example in Figure 10 uses:

```
/dev,font,1,Comic*Sans*MS,700,0,-23,0,1,,
```



A Publication for ANSYS Users

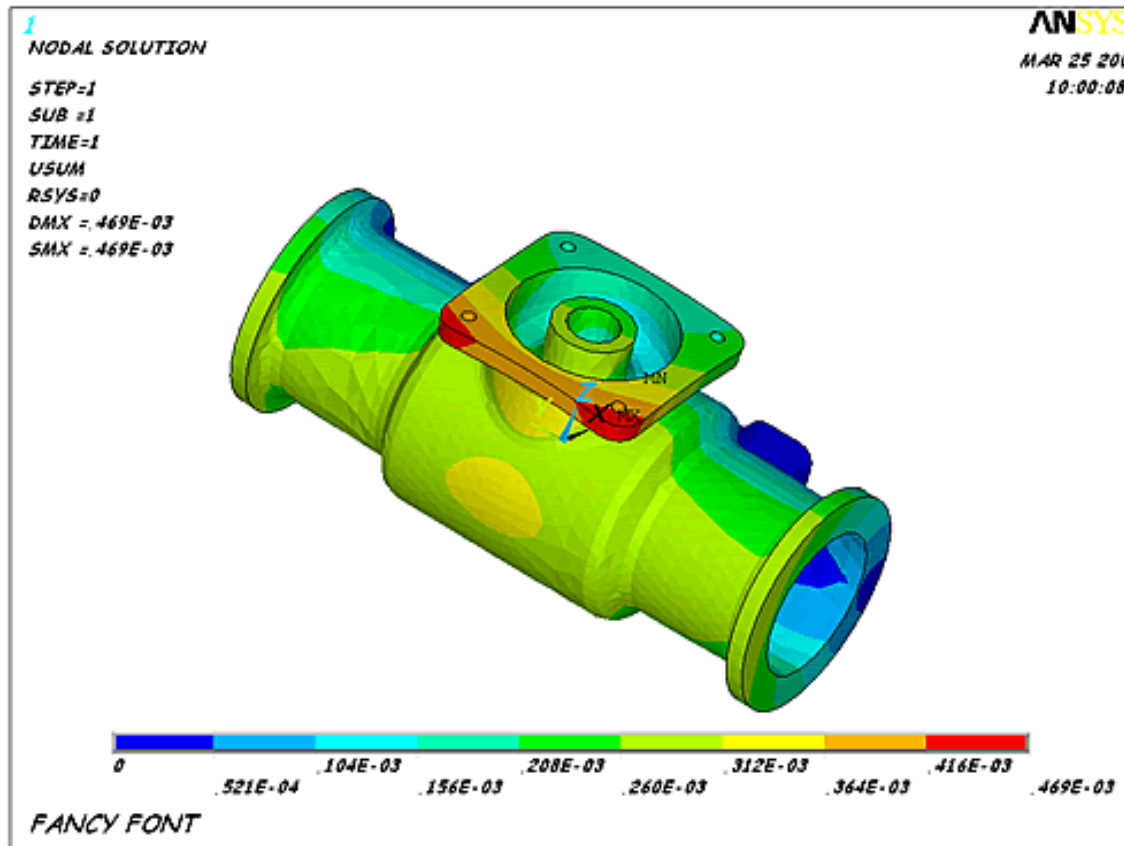


Figure 10. Fonts.

Making Cool Contour Plots for Web and Marketing

A reality of the world is that sometimes you need a plot just to have a cool picture for something. If you take a standard bitmap file and try and delete all the text, contours and such you can spend a lot of time. To make these types of images I use:

```
/plopt,info,off
/plopt,frame,off
/plopt,title,off
/plopt,minm,off
/traid,off
```

This turns everything off, and then I plot to PNG (don't use JPEG because of the fuzzies!). Once the image is imported, you can set the background to transparent and you get something like Figure 11.

The Focus



A Publication for ANSYS Users

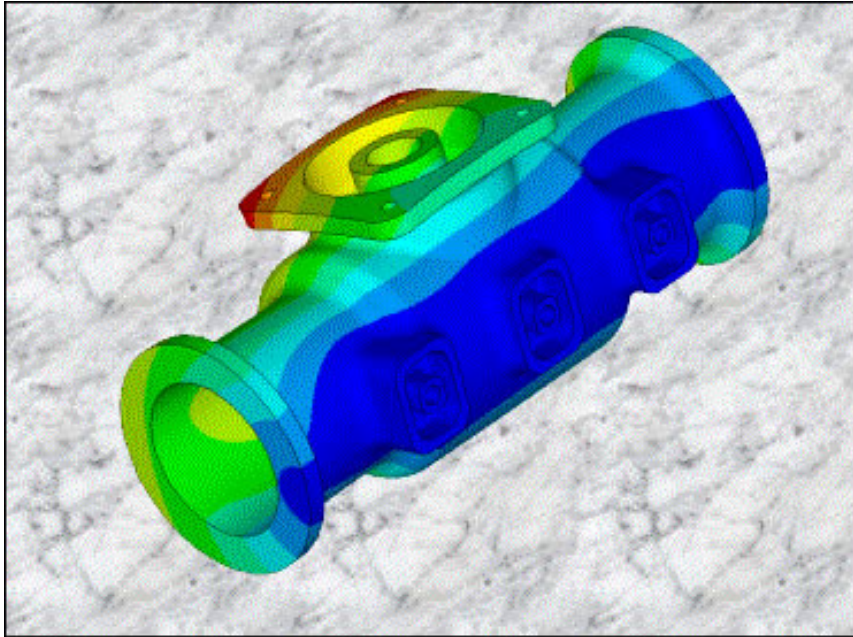


Figure 11. Cool plot.

The Focus



A Publication for ANSYS Users

ANSYS, Elastomers, and the Physical Test Lab

by [Kurt Miller](#), Axel Products

Analysts are often required to simultaneously perform the roles of designer, engineer, material scientist, and experimentalist. This is especially true for simulation projects involving nonlinear materials and complicated loading conditions. This can be a difficult challenge because the situation may require reasonable assumptions and good judgment when the analyst doesn't have the experience in all of these disciplines.

Analysis of elastomeric parts is just one of many areas that nearly always has complicated material issues, modeling difficulties, and requires good judgment from the analyst. A new course, [Experimental Elastomer Analysis with ANSYS](#), has been developed to bring together the disciplines needed to be successful in this area.

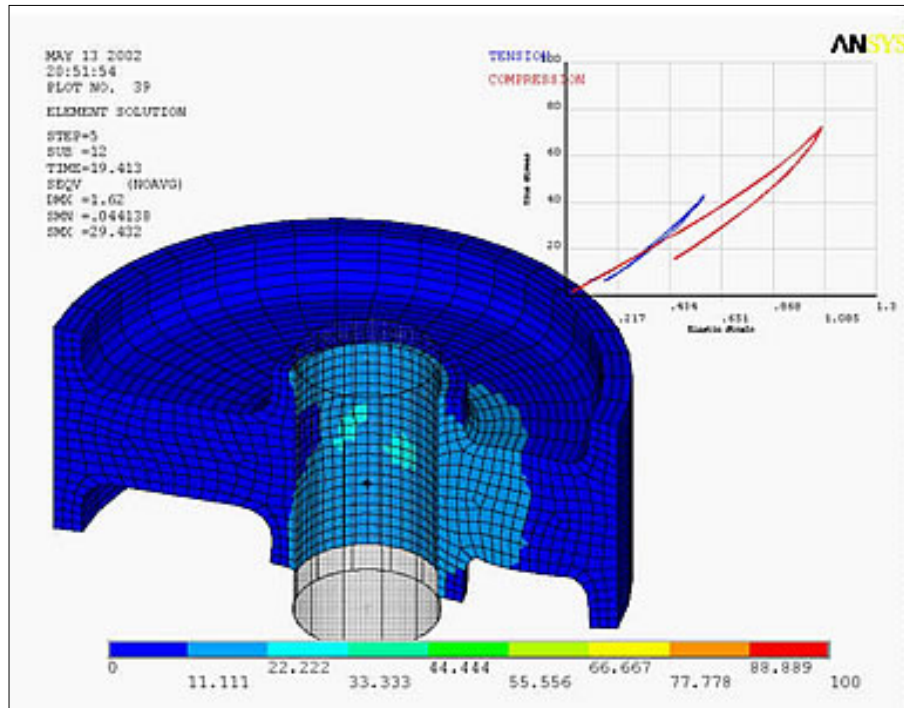


This is a hands-on workshop covering material testing, material modeling, and finite element analysis of elastomers. Instructors from [ANSYS Inc.](#) and [Axel Products, Inc.](#) present an integrated testing and analysis workshop using the experimental facilities of Axcel Products, Inc. and ANSYS software. Attendees perform elastomer experiments using laboratory instruments to create data appropriate for use in building elastomer material models in FEA. Material models are then developed and examined on workstations running ANSYS.

The Focus



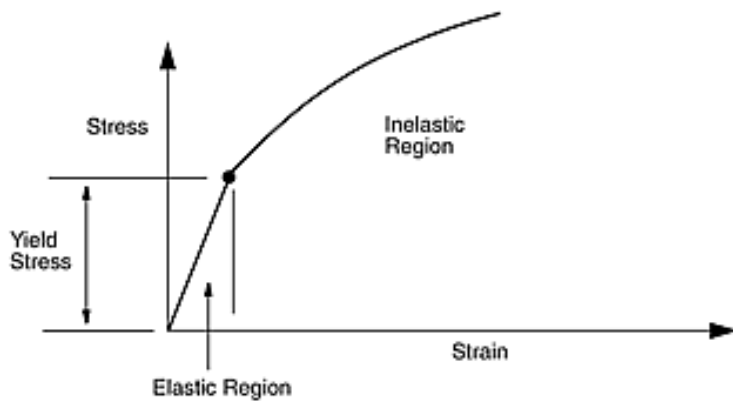
A Publication for ANSYS Users



The basic idea here is attain specific knowledge and to accelerate the development of judgment. A few of the areas that the course covers follow:

When simple things aren't simple.

Lets imagine that the part of interest is soft, elastic and used in compression so getting a modulus of elasticity and a yield point for the material might seem like a good plan. We could use an elastic-plastic model and get this job done quickly.



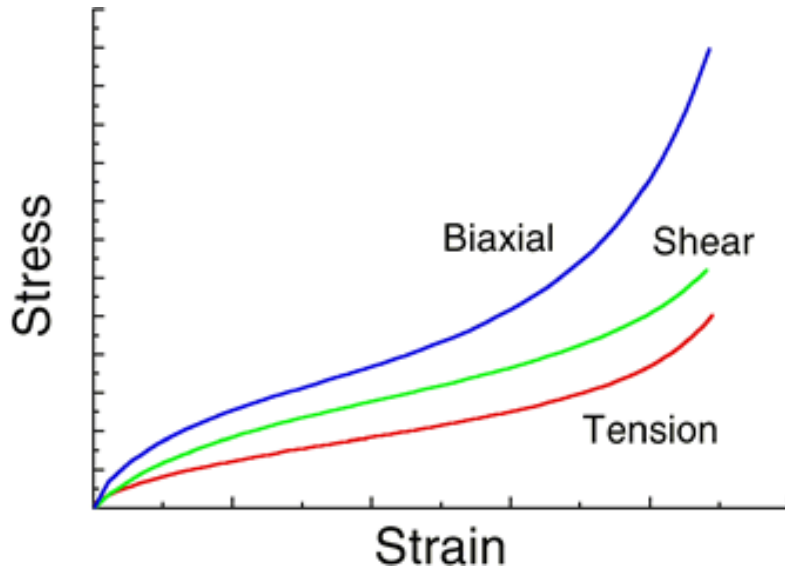
This scenario would likely lead to failure. Elastomers are typically nearly incompressible. As such, compression in one direction leads to significant strains and stresses in all directions. It is important that the incompressibility is taken in account. Special elements are available in ANSYS for this.

Furthermore, most elastomers do not exhibit a linear region from which a modulus of elasticity can be measured. The stress-strain curves tend to be significantly non-linear.



A Publication for ANSYS Users

Also, in large part due to incompressibility, most material models needed to define the static structural properties of elastomers require one or more stress-strain data sets which are then “fitted” in ANSYS to define the material model. These may include tension, pure shear, and equal biaxial experiments.



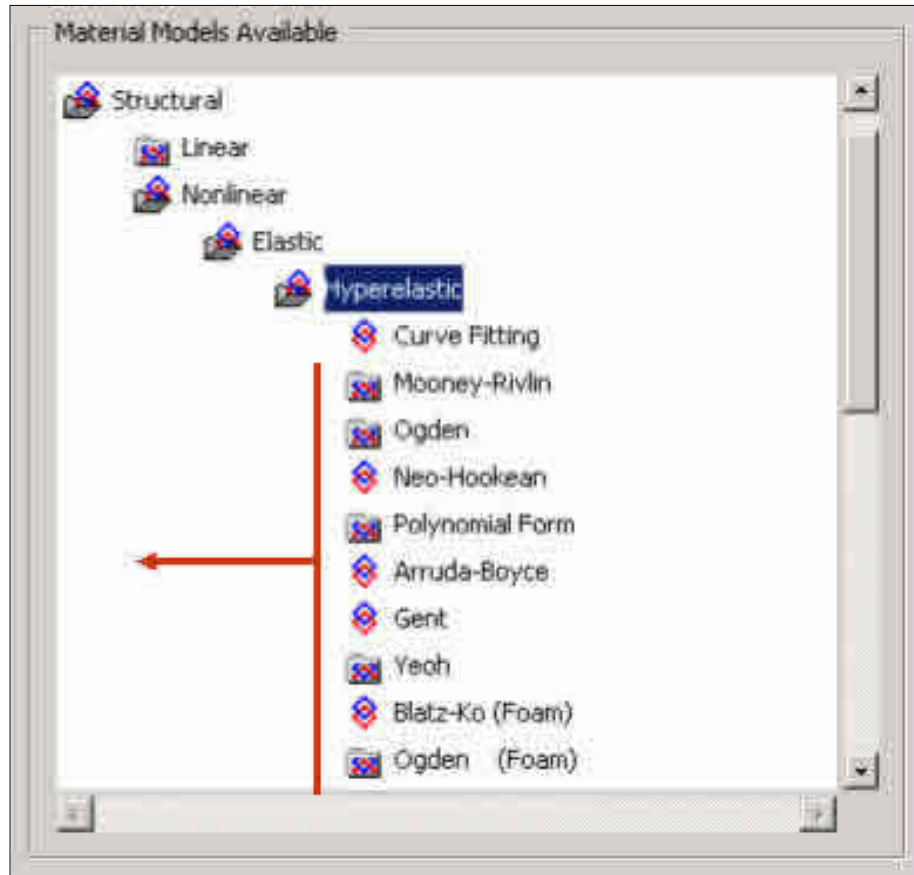
Why are there so many material models for elastomers?

We do have many choices for material models in ANSYS for elastomers and this is a good thing. There are only a few general “forms” and many small variations. Some of the models are very general and very capable but typically require complete experimental data sets to define. Other models simplify the material behavior, and perhaps make it less accurate, but reduce the likelihood of generating physically non-real outputs.

The Focus



A Publication for ANSYS Users



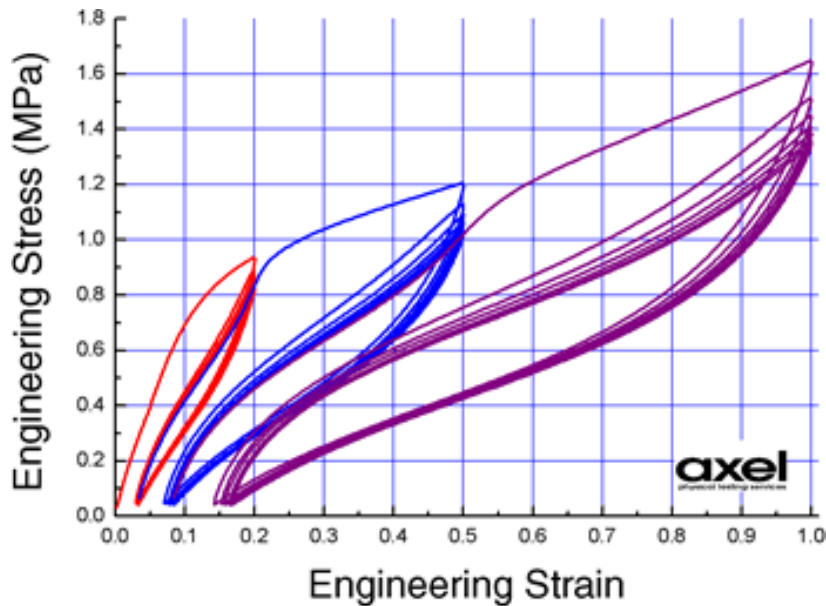
Our supplier provided Mooney-Rivlin constants. Can I use these in ANSYS?

Maybe. Any one set of constants defines a hyperelastic material model for one loading condition. Most elastomers soften with repeated loadings and soften more significantly with large strains than with small strains. Sometimes elastomers exhibit some plastic deformation. The critical issue here is that the analyst understands the exact conditions under which the physical test data and subsequent curve fitting was performed so that the applicability of the resulting Mooney-Rivlin constants to the current application may be examined.

The Focus



A Publication for ANSYS Users



While the Mooney-Rivlin constants tend to be simple in form, other hyperelastic models such as Ogden, are sometimes implemented differently in different FEA codes. As such, constants developed in ABAQUS or taken from the literature may not work in ANSYS.

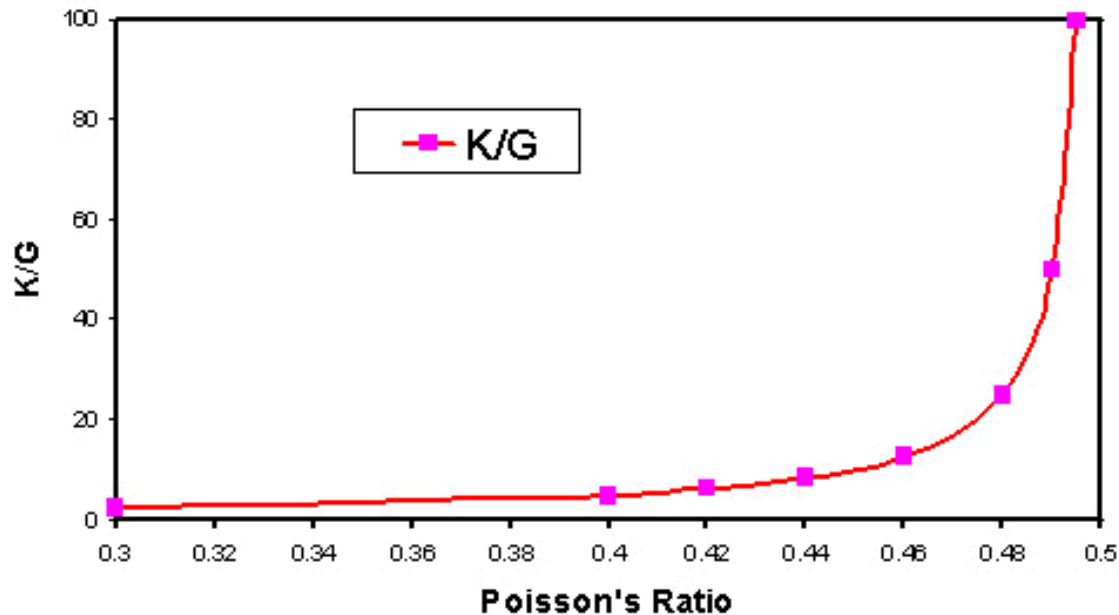
What about Poisson's ratio?

The Poisson's ratio for elastomeric materials is said to be near to 0.5. Poisson's ratio is a measure of a material's compressibility. Exactly 0.5 would indicate that the material is perfectly incompressible. This is a very interesting point and the idea of a Poisson's ratio is valuable but it is not practical to measure this property for elastomers. Very small changes in a material's Poisson's ratio, say 0.47 to 0.49 could have significant compressibility ramifications in an application where an elastomer is highly constrained. It can be very difficult to measure Poisson's ratio in the laboratory due to the level of precision required, especially on soft materials where extensometers can't be attached to the material.



A Publication for ANSYS Users

K/G Relationship to Poisson's Ratio



However, the bulk modulus, which is also a measure of compressibility, may be determined using a volumetric compression experiment. The bulk modulus may be measured with sufficient high resolution for elastomer applications. In general, Poisson's ratio is measured for plastic applications and bulk modulus is measured for elastomeric applications.

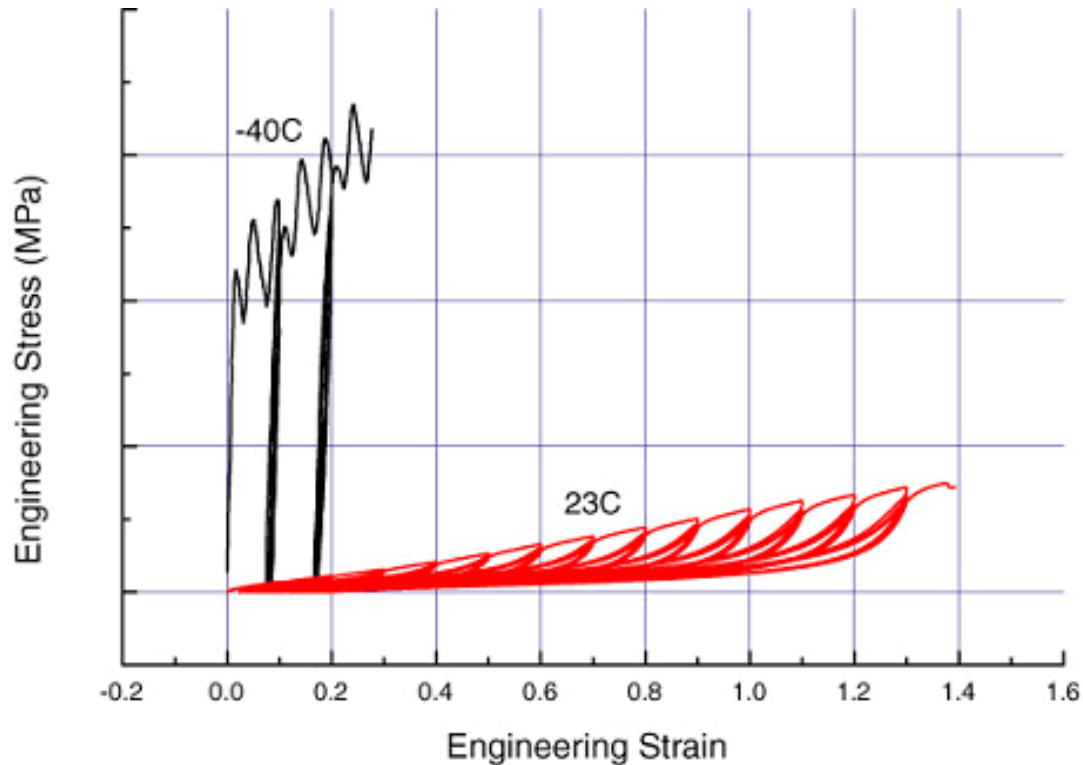
When rubber isn't rubber.

For the purpose of material modeling in ANSYS, the important aspect of a material is how it responds to thermal and mechanical events, not the trade name. Periodically a material with an elastomer "name" is modified to be stiffer or it is used a temperature below its glass transition temperature. A traditional stress-strain curve from a tensile test may not indicate that the material is not elastic unless the unloading behavior is examined as well.

The Focus



A Publication for ANSYS Users



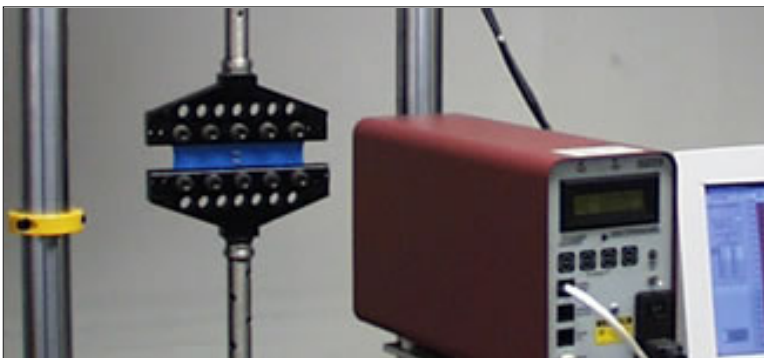
Are we there yet?

There are many effects and sensitivities that are somewhat unique to elastomers. Strain rate sensitivity, temperature sensitivity, viscoelasticity, softening, and thermal expansion to name a few. When do we care and perhaps more importantly, when do I quit? What is important in my particular application right now? The analysis of elastomeric parts is all about compromise.

Why run an ANSYS course in a laboratory?

The above topics describe a few of the practical aspects covered in the new Experimental Elastomer Analysis with ANSYS course. Each concept taken by itself is simple and easy to demonstrate in the laboratory. The ideas become real when observed during testing. Hands on experiments on elastomeric materials and curve fitting in the ANSYS computer workshops provides a solid grounding in the real issues surrounding the successful implementation of elastomer part simulation.

To learn more about this course, please see the [Course Description](#).





A Publication for ANSYS Users

Tcl/Tk in ANSYS

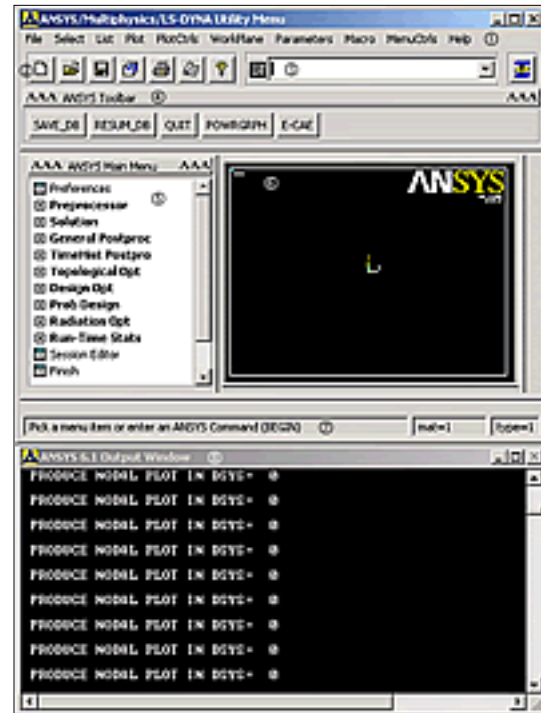
by [Matt Sutton](#), PADT

If you have used ANSYS since version 6.1, you probably have noticed the new look and feel of the default GUI. Gone are the teal green cascading menus and multiple windows. Now, the GUI has a more modern Windows look in which everything by default is contained in one window. Furthermore, the complex cascading menus have been replaced by a more hierarchical tree control in which the logical grouping of commands is more obvious.

Now, if you are a seasoned ANSYS veteran, you may be snarling saying “Why didn’t they leave good enough alone!!” I’m sure there were similar remarks made when punch cards were replaced by magnetic tapes, but alas, change happens. However, I for one think the new look and feel of the ANSYS interface is more conducive to increased productivity. Even the command line has been reworked to provide added hints and increased flexibility.

Perhaps while waiting for a contact problem to converge, you let your mind wander and asked yourself, “I wonder exactly how ANSYS created this new and improved GUI, and how did they do it so fast.” Would you believe that primarily one person wrote the vast majority of it in the interim between two releases? How did he do that?

Well, he put forth a good deal of effort, but in the end it was feasible because of the power and flexibility of the tcl/tk scripting language. Tcl is often heralded as the most underrated scripting language around. It often takes a back seat in scripting parlance to more fashionable languages like Python. However, tcl’s flexibility and availability have historically made it the primary choice in implementing platform independent user interfaces.



The Focus



A Publication for ANSYS Users



Tcl traces its ancestry to a man by the name of John Ousterhout, who was a professor of Electrical Engineering at the University of California at Berkeley. Dr. Ousterhout developed the language to assist his research into chip design. However, new and more ubiquitous uses for the little scripting language soon became apparent, especially after the tk GUI widget toolkit came to fruition. ANSYS began integrating tcl into the ANSYS system around version 5.6. Since that time, more and more functionality in ANSYS has transitioned from the traditional UIDL implementation to a tcl/tk implementation.

Great! “So what,” you say. Well, the “so what” is that all of the tcl functionality used to build the current ANSYS GUI is available to you, the user, right out of the box! You don’t need expensive, specialized compilers. You don’t need to understand software build processes or complicated make files. All you need is a text editor and your imagination to take your APDL scripting to the next level in user functionality.

Imagine building a sophisticated, intelligent wizard to step users in your organization through a complicated simulation process. With tcl integrated into ANSYS, the wizard can interact with the ANSYS database and dynamically change its appearance as the user progresses through the steps. Just look at the contact wizard as an example. You can create similar functionality for your own simulation processes.

Perhaps your aim is a bit less lofty. Perhaps you just want to dump some precisely formatted data from ANSYS out to a file, and you’re tired of trying to figure out the right combination of *vwrites, etc. Use tcl and lower your blood pressure! Manipulating strings and text files is a snap in tcl.

Furthermore, we can show you how. We have a good deal of experience writing applications in tcl, as well as customizing the ANSYS GUI. We really enjoy helping others learn how to do the same, and consequently we have developed a [Tcl/Tk in ANSYS](#) class that will give you all the tools necessary to write killer apps in ANSYS. Just look at our [training schedule](#) or contact us to find out more!





A Publication for ANSYS Users

A great way to get started using Tcl/Tk is to take a look at the [first article](#) from the November 25, 2002 issue of *The Focus*. For some other online resources, please refer to the following:

Books

- *Tcl and the Tk ToolKit*, by John Ousterhout
- *Effective Tcl/Tk Programming*, by Mark Harrison and Michael McLennan

Links

- [Tcl Developer Exchange](#)
- [PADT's Tcl/Tk in ANSYS Resource Page](#)

Downloads

- [Software downloads for various platforms](#)



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.