

The Focus



A Publication for ANSYS Users

Contents

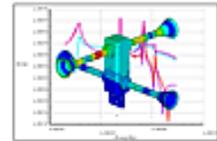
Feature Articles

- [Taming ANSYS 6.1](#)
- [GST Plot: Just What Is That Plot Anyway?](#)
- [xPSD: Fast & Accurate PSD Analysis for ANSYS](#)



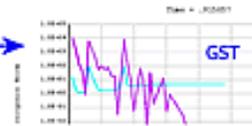
On the Web

- [ANSYS WTC Study](#)
- [PADT: Contact Analysis Seminar](#)
- [PADT: Advanced Analysis Seminar](#)



... Also

- [Enhanced Random Vib's](#)
- [Interpreting This!](#)



Resources

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



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

The Focus



A Publication for ANSYS Users

Taming ANSYS 6.1

by [Rod Scholl](#)



ANSYS release 6.1 included several large changes. We will address a few of these changes and also some of the features that were introduced as a result. The enhancements new to the 6.1 release are as follows. The highlighted topics will be discussed in this article.

- Usability
 - BBOX option on /DEVICE applies to Element plots also
- System Enhancements
 - [New GUI](#)
 - [Memory Manger](#)
- Other New Features
 - ~Heal Module replaces CADfix for ANSYS
 - Cyclic Symmetry documentation enhanced
 - Post26 includes Repsonse Spectrum, and calculate Covariance
- Structural
 - Cyclo-Symmetry with dissimilar meshes
 - Gasket Simulation using INT192-195
 - ADAMS Interface
 - Cast Iron material property added
 - Viscoelasticity added to more elements
 - Meshing order can be based on area size (meshes from smallest to largest)
 - Contact Wizard Toolbar
 - Generalized Plane Strain

The Focus



A Publication for ANSYS Users

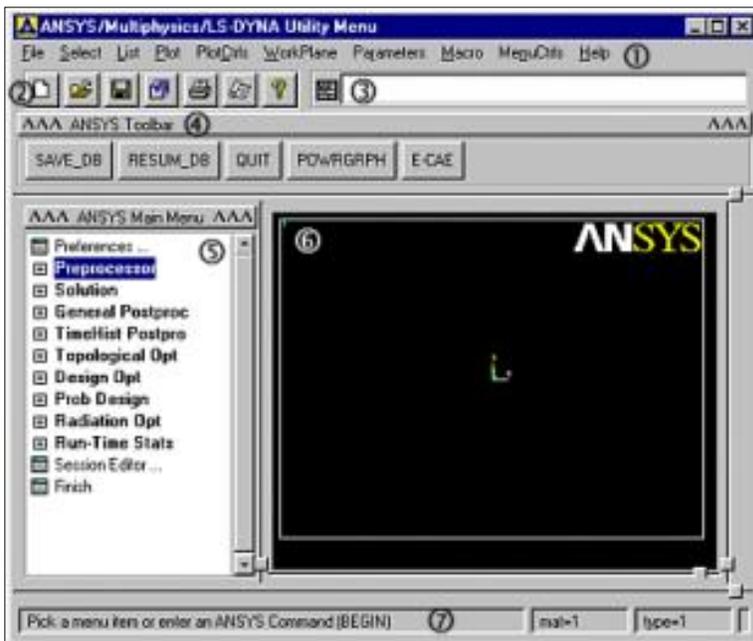
- Coupled Field
 - Fluid-Solid Interaction
 - Piezoelectric-Circuit Analysis
 - 2-D Electromechanical Transducer Element
- Electromagnetics
 - High Frequency Transmission Line Propagation
 - Modal Analysis of Structures using perfectly matched layers
 - LMATRIX macro allows use of iterative/sparse solvers
- Probabilistic Design
 - Equal Interval Spacing of DOE levels using PDDOEL
 - Normalization of input variables more robust in RSFIT
- ANSYS/LS-DYNA
 - Strain-Rate effects added to Plasticity Model
 - Enhanced Failure Criteria for Plasticity Model
 - .RST files can be split with /COFIG
 - Local Coordinate Systems support offset origins
- ANSYS Commands
 - AVRES , FULL valid during /CYCEXPAND
 - /CYCEXPAND is recommended over /EXPAND
 - /PSOLVE , EIGLANB solves modal over multiple load steps

The New GUI

The Focus



A Publication for ANSYS Users



ANSYS 6.1 has a new face attached and for good reason. The new GUI is written in a multi-platform language (Tcl/Tk) for greatly enhanced ability to customize. Anyone who has sifted through a .GRN file using UIDL should be pleased. Also, changes are functional on both Windows and Unix platforms! Due to a number of factors the new GUI was released on a very tight schedule. Because ANSYS, Inc. wanted large feedback from the users, it was decided to make the new GUI the default and

provide a command to switch back to the old GUI. Had they done it otherwise, it is likely that no one would have explored the new menus, and they would not have benefited from the mass of recommendations, fixes, etc. which ensued after the release of 6.1.

So our recommendation is to check out the new GUI and if you get too bogged down using it, switch back to the old Menu style by the following steps:

° Switching 6.1 to the Classic Menus

1. In a text editor, open your `start61.ans` file in your ANSYS installation directory /Docu folder. Something like `C:\Program Files\ANSYS_Inc\Docu\start61.ans`.
2. To the bottom, add the command: `/MSTART,UTIL,1`
3. Restart ANSYS

° Getting Started in the New GUI

At some future release, the classic menus will no longer be available so know that at some point you will have to make the leap. Besides a few irritating behaviors, the new GUI is simple to use. There is zero learning curve, because the menu trees are almost identical. There is a couple of extra layers added here or there, which adds a few mouse-clicks that I never like. However, we gain little +/- buttons with right-clickable behavior of which you will want to take advantage. In sum, it should be less clicks, not more.

The Focus



A Publication for ANSYS Users

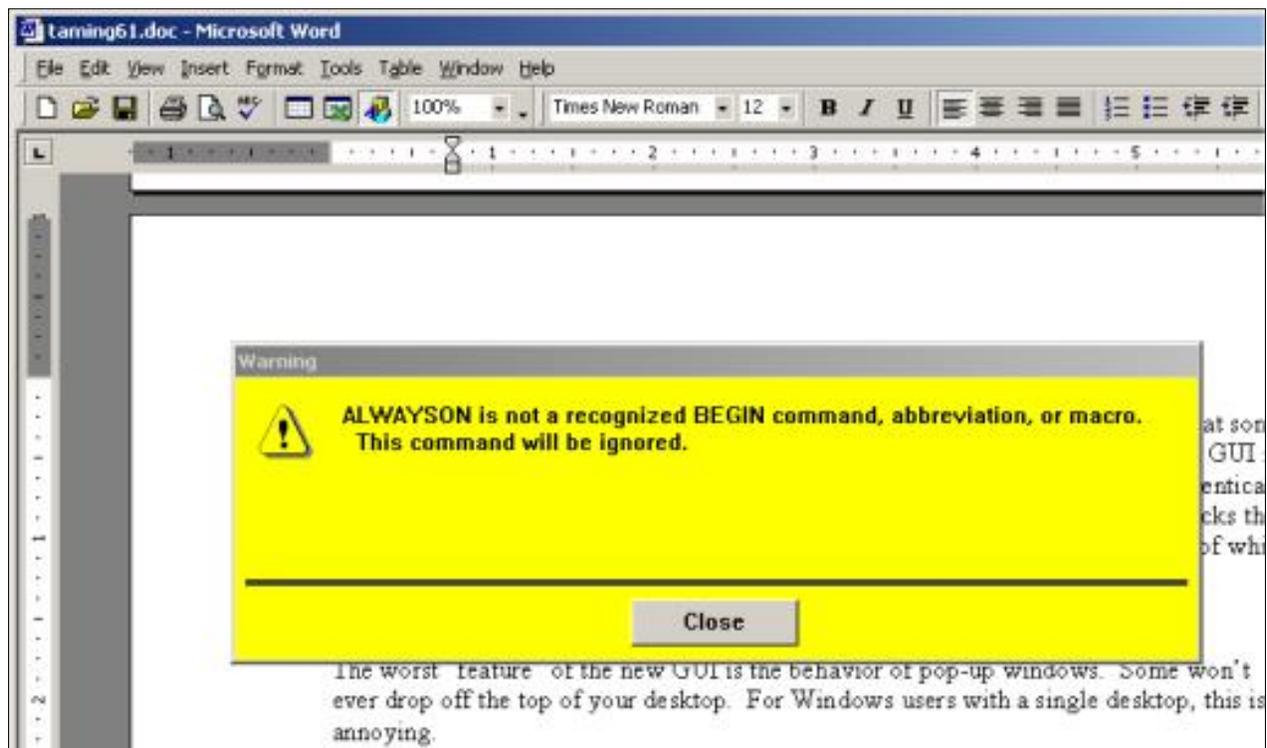
◦ The Worst Part

The worst of the worst is that the command line input loses focus each time a plot command is issued. Thus, after every APLOT, LPLOT, EPLO, one must re-click in the input window to continue issuing commands. The good news is that there is a patch macro for this, which you can download from our website. It can change your life!



- [fix61.mac](#) (817 bytes)

The next worst feature of the new GUI is the behavior of pop-up windows. Some won't ever drop off the top of *any* application! For Windows users with a single desktop, this is annoying.



Other windows get lost *behind* the ANSYS GUI such as list menus and Tcl GUIs. Also, the focus may be stuck in a Tcl GUI underneath the main window. In these cases, you are best to select the hidden Tcl GUI from the application tab on your Start Bar.

All this behavior should be rectified by the next release. There are plenty of additional complaints in the community; everyone has their own preferences. We won't cover them here—we'll let you discover them on your own! You can always submit change requests to your ASC, ASD, or me.

The Focus



A Publication for ANSYS Users

The Memory Manager

There is a new memory manager meaning you do not have to specify a `mem` on startup. ANSYS will dynamically allocate Scratch Space memory as needed. This has a few advantages:

- Avoids needing a contiguous block of memory
- Avoids tying up memory when not needed

And one *big* disadvantage:

- `-db` memory is stuck on 32 MB. It doesn't dynamically allocate.

Thus if you have a `.db` file (or `.db + .rst`, if you are post-processing) that is greater than 32 MB, ANSYS is going to supplement RAM with a `.PAGE` file which is very, very slow.

NOTE: ANSYS still supports the use of `mem` and `db`. *I recommend you continue using them.*

° Running Pre-Stress in the Classic GUI

There are several bugs introduced in 6.1. One we have found particularly beguiling is activating pre-stress. In order to run a pre-stressed modal analysis, for example, one must activate pre-stress during the static stress run. The manual will tell you to seek:

Main Menu > Preprocessor > Loads > Analysis Options

Main Menu > Solution > Analysis Options

However, by default the Analysis Options menu choice does not show up until you perform:

Main Menu > Solution > Analysis Type > Sol'n Controls

In the Advanced NL Tab, click on 'Abridged Menu' to turn on the 'Unabridged Menu', then click on OK. Now the 'Analysis Options' menu pick is available, and you can use the GUI to turn on pre-stress for your static run.

In Conclusion

Go to it, and good luck.



The Focus

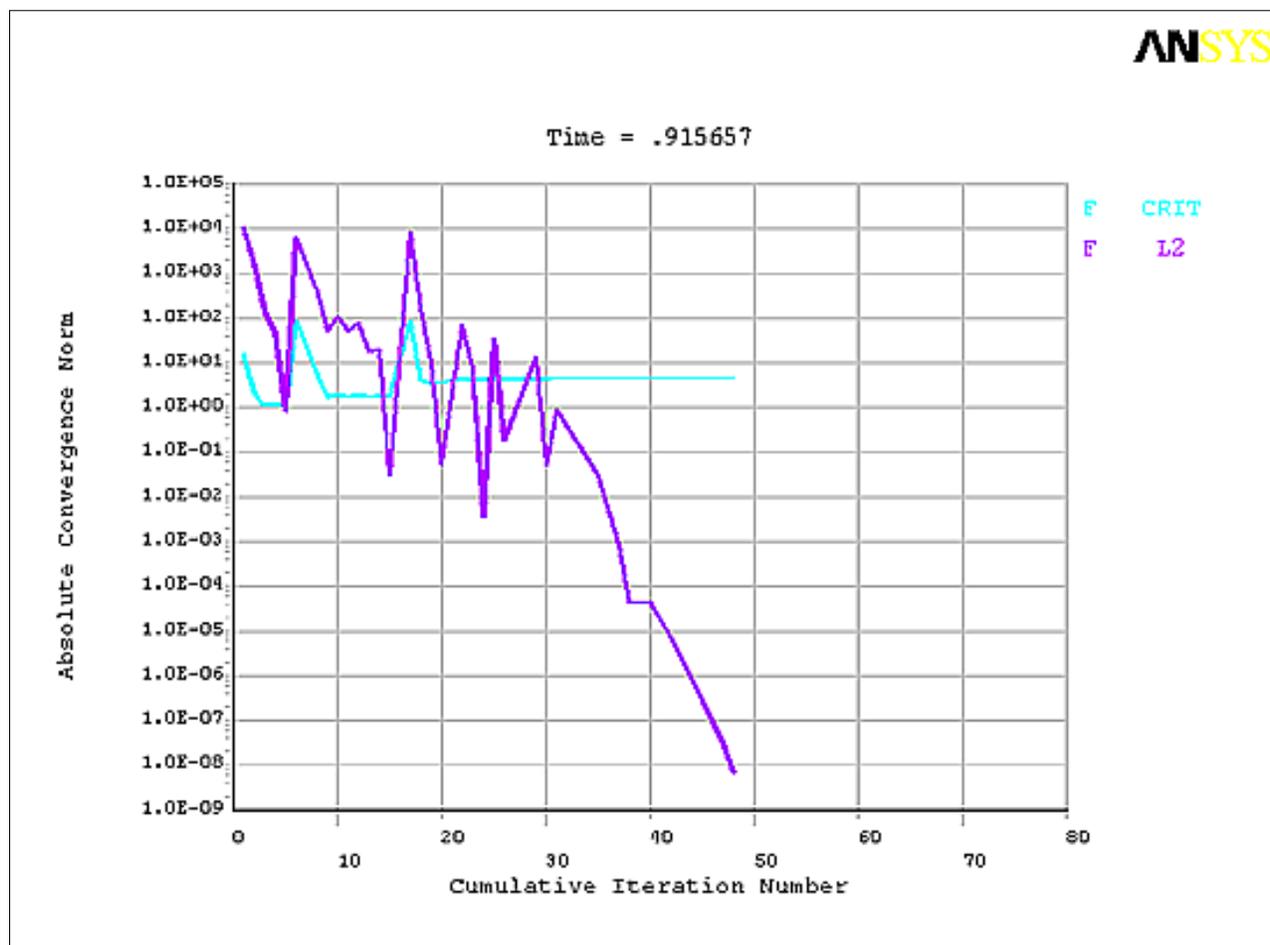


A Publication for ANSYS Users

GST Plot Just What Is That Plot Anyway?

by [Ted Harris](#)

Any time you run ANSYS and it performs a nonlinear analysis, you'll see a Graphical Solution Tracking, or GST plot. Without some explanation, this plot can be confusing.



Why does the GST plot appear? Any time one of the flags is set for nonlinear analysis, ANSYS will display the GST plot when running interactively. So if you see the GST plot during your interactive solution run, you know that ANSYS is performing a nonlinear analysis.

Many times it's obvious that ANSYS will perform a nonlinear analysis. A general contact analysis, for example, is a nonlinear problem since ANSYS must iterate to determine whether the contact element status is open or closed. A plasticity analysis is nonlinear as well, since there is an inherent nonlinear relationship between stress and strain and hence, between deflection and applied load.

In other cases it's not so clear why ANSYS is performing a nonlinear analysis. A coupled field

The Focus



A Publication for ANSYS Users

analysis, for example, which solves a structural and thermal problem at the same time is a nonlinear solution, even though the structural and thermal analyses might both be linear if run separately. Turning on large deflections in a structural analysis also causes ANSYS to do a nonlinear analysis. In this case, the nonlinearity is the change in stiffness with changes in deflection.

The GST plot can provide us with useful information on the performance and success of our nonlinear analysis. The quantities being plotted are always some residual vs. a tolerance. The residual can be thought of as the difference between the applied load and the calculated reaction load. Ideally we want the reaction load to match the applied load. In a linear analysis this is exactly what happens. In a nonlinear analysis, however, the Newton-Raphson technique is utilized to iterate to a solution. Without going into the details of the math, a norm, which is a scalar quantity used to express the reaction load, is compared to a norm of the applied load. When the difference between those two quantities becomes less than the tolerance, our solution is converged.

Next, consider that in a nonlinear analysis we typically use multiple substeps per load step. Think of load steps as the top level load history control time points. We, as users, specify one or more load steps for our analysis to apply the loads in a manner that makes sense for our simulation. Within each load step, several substeps are usually performed. Since loads are normally ramped from their initial value at the beginning of a load step to their final value at the end of a load step, each substep takes a bigger bite of the load. Thus using multiple substeps allows ANSYS to tackle a smaller bite of the solution at a time rather than attempting to apply the full load of a load step all at once. This generally enhances the convergence behavior, allowing us to more accurately model the load and response history, and sometimes allowing us to get a converged solution that we couldn't otherwise obtain without doing multiple substeps.

Within each substep, ANSYS will typically perform several equilibrium iterations, using the Newton-Raphson technique, to balance the reaction load with the applied load. The displacement increment is adjusted in these iterations until convergence on load is achieved.

As users, we control the load steps and to a large degree the substeps, but we don't have direct control over the number of equilibrium iterations needed to obtain convergence.

Getting back to the GST plot, the F_{L2} in our example plot represents the L2 norm of the force residual, and F_{Crit} is the force criterion. Our goal is for the norm to be less than the criterion, at which point our solution is in most cases converged for the current substep. ANSYS will then continue with the next substep and its corresponding load increment, or will report that the solution is done if that was the last substep.

The next issue to address is the convergence criterion. It's all well and good to say we have a converged solution, but we need to know to what criterion we have converged. By default, ANSYS calculates a convergence criterion based on the norm of the applied load. A factor, which defaults to 0.005, is applied to the norm of the applied load in a structural analysis and the result is the convergence criterion. Thus, as the incremental load changes through the course of our analysis, the convergence criterion will change as well.

The type of analysis and the degree of freedom set determine the convergence quantities. For example, force, moment, and displacement criteria might be tracked in a six degree of freedom

The Focus



A Publication for ANSYS Users

structural analysis, while a temperature criterion is likely the sole criterion in a basic heat transfer analysis. The ANSYS documentation provides details on the default criteria used for each analysis type.

As our analysis progresses, we want the norm curve (L2 norm by default) on the GST plot to drop below the criterion curve. If it does, in most cases the substep will be converged and the analysis will continue with the next substep, if any. If the norm curve does not drop below the criterion curve, or does so slowly, we need to make some adjustments to our analysis to improve its performance or at least to get it to converge.

Thus, the GST plot gives us a history of the convergence behavior of our nonlinear analysis. We can use this information to track how well or how poorly our solution is converging, in hopes that we can improve the performance if necessary. The knobs to turn to improve our convergence behavior will be the topic of a future article.

The Focus

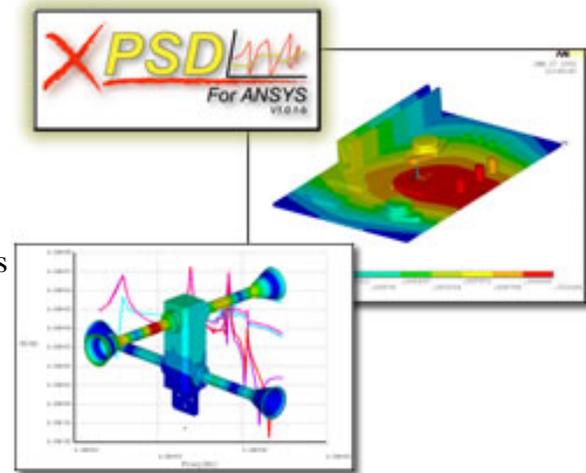


A Publication for ANSYS Users

xPSD Fast & Accurate PSD Analysis for ANSYS

Why xPSD?

- Standard PSD Method in ANSYS Offers a robust and comprehensive solution for Random Vibration Analysis
- However...
 - It uses a huge amount of disk space
 - It can be very slow
 - It gives RMS results for invariant values
- Users who have large static structures (aerospace, automotive) find the standard ANSYS approach ineffective
- PADT developed xPSD in direct response to requests of Raytheon Missiles in Tucson



What Is xPSD?

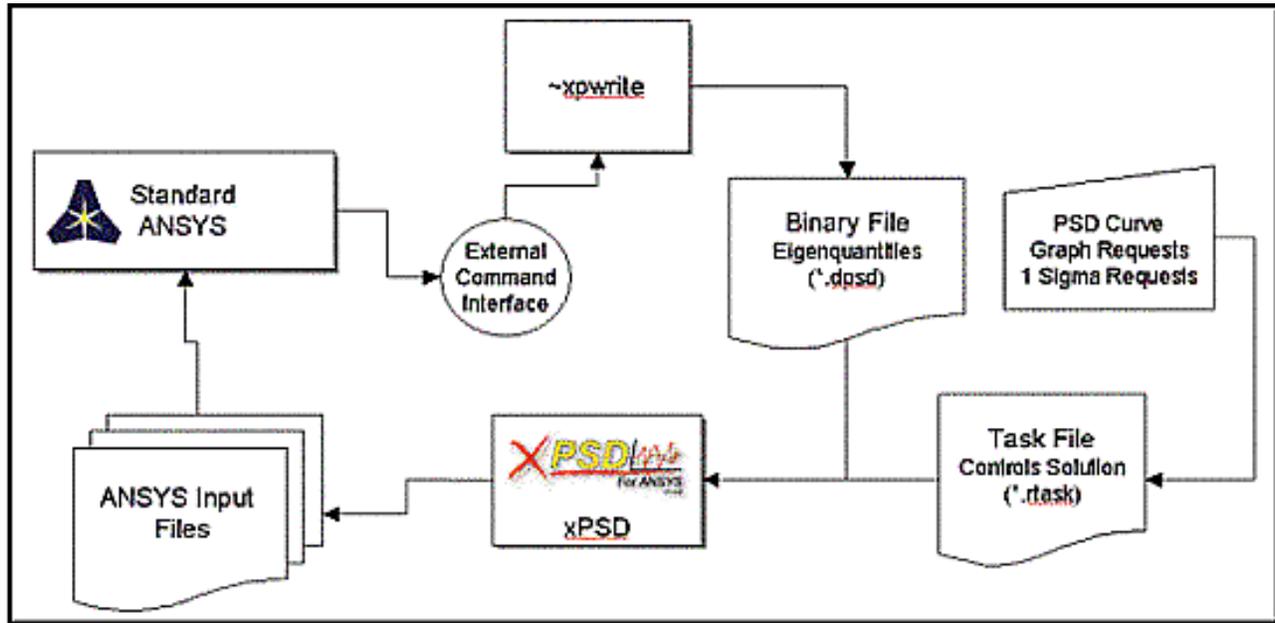
- xPSD is an implementation of the "Large Mass Method" for simulating the effect of a random vibration input on a structure
 - Input is in the form of a **Power Spectral Density** curve
- Contains:
 - One external ANSYS command: `~xpw`
 - One external solver: `xpsolve.exe`
 - Several utility APDL macros

xPSD Procedure

The Focus



A Publication for ANSYS Users



~xpw

- ANSYS external command
- Extracts results from a modal solution and places them in a compact binary result file
 1. Run the modal analysis and save results to compact file (avoids large .RST file)
 2. Run expansion only (avoids large .RST file)
 3. Extract results from existing .RST file
- Resulting DPSD file is usually one-tenth the size of an .RST file
- Much faster I/O than ANSYS .RST file

xpsolve

- Actual PSD solution is done in external program called xpsolve
- User inputs one or more PSD input curves
 - Different dimensions allowed
- User specifies requested output
 - 1-Sigma response on component displacement/stress/strain values
 - PSD Response
 - Statistically Significant Equivalent Stresses
- Placed in a batch file: RTASK
- Very fast: 10 to 10,000 times faster than an ANSYS solution

The Focus



A Publication for ANSYS Users

RTASK File

```
*load,1,disp,log,0,0,1
20,0.0045
30,0.0045
50,0.0057
70,0.0057
90,0.0225
100,0.030
150,0.030
160,0.015

*Plot,1,Node 81 and 43 vs Input Load1,disp,log,500
1,81,1,uz
2,43,1,uz
3,122,1,uz

*sig1,1,sxy for All Nodes,sxy,1,disp,,
*sig1,2,sy for all nodes,sy,1,disp,,
*sig1,3,uz for all nodes,uz,1,disp,,
*sig1,4,sxy for component nodes,sxy,1,disp,,test.comp
*sig1,5,s1 for all nodes,s1,1,disp,,
*sig1,6,seqv for all nodes,seqv,1,disp,,

*job,test

*Damp,.01

*G,386.4
```

Results

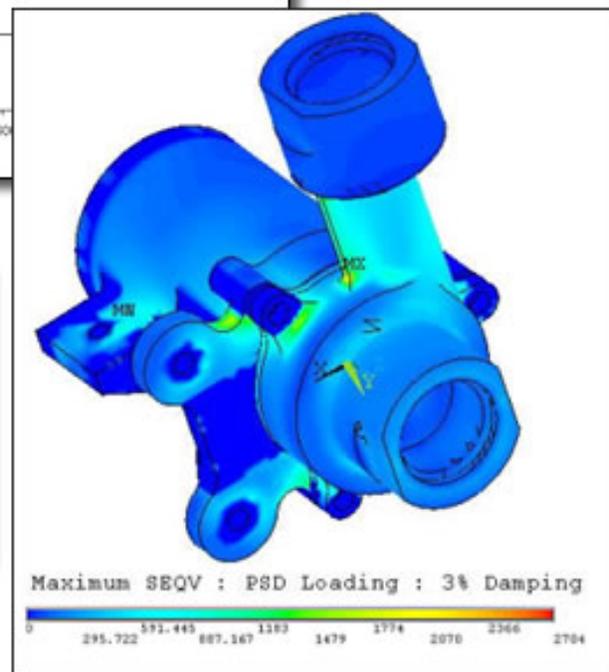
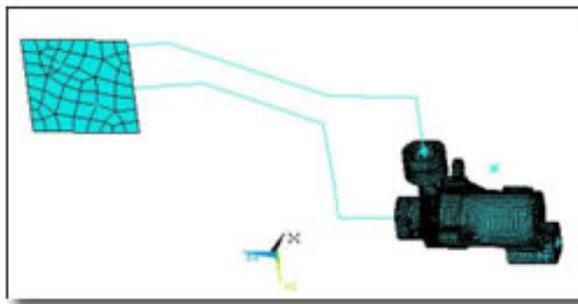
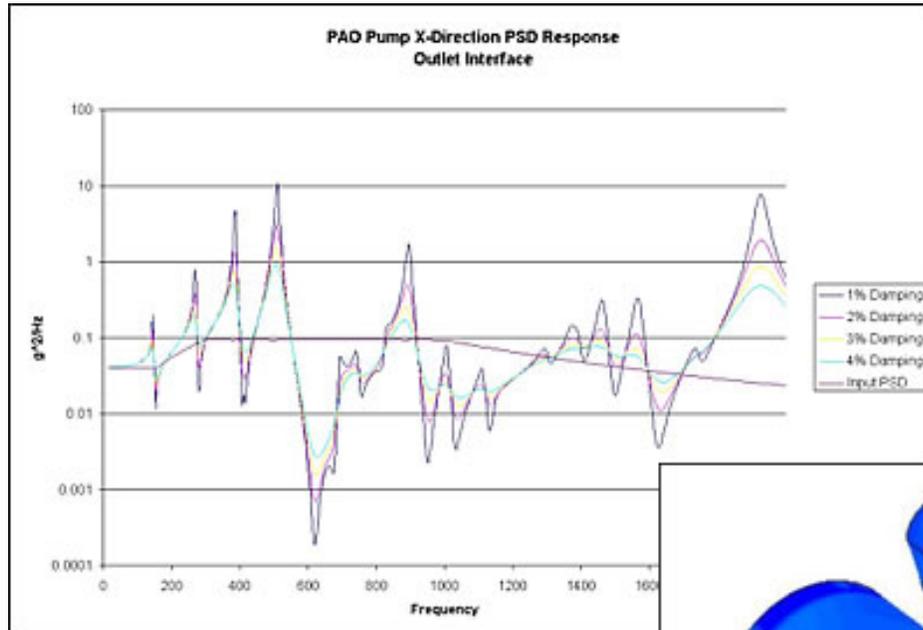
- 1-sigma results are placed in ANSYS vectors and *vput onto model
- Response curves are output to ANSYS tables for *vplot, or as comma-delimited files for Excel

Example: PAO Pump

The Focus



A Publication for ANSYS Users

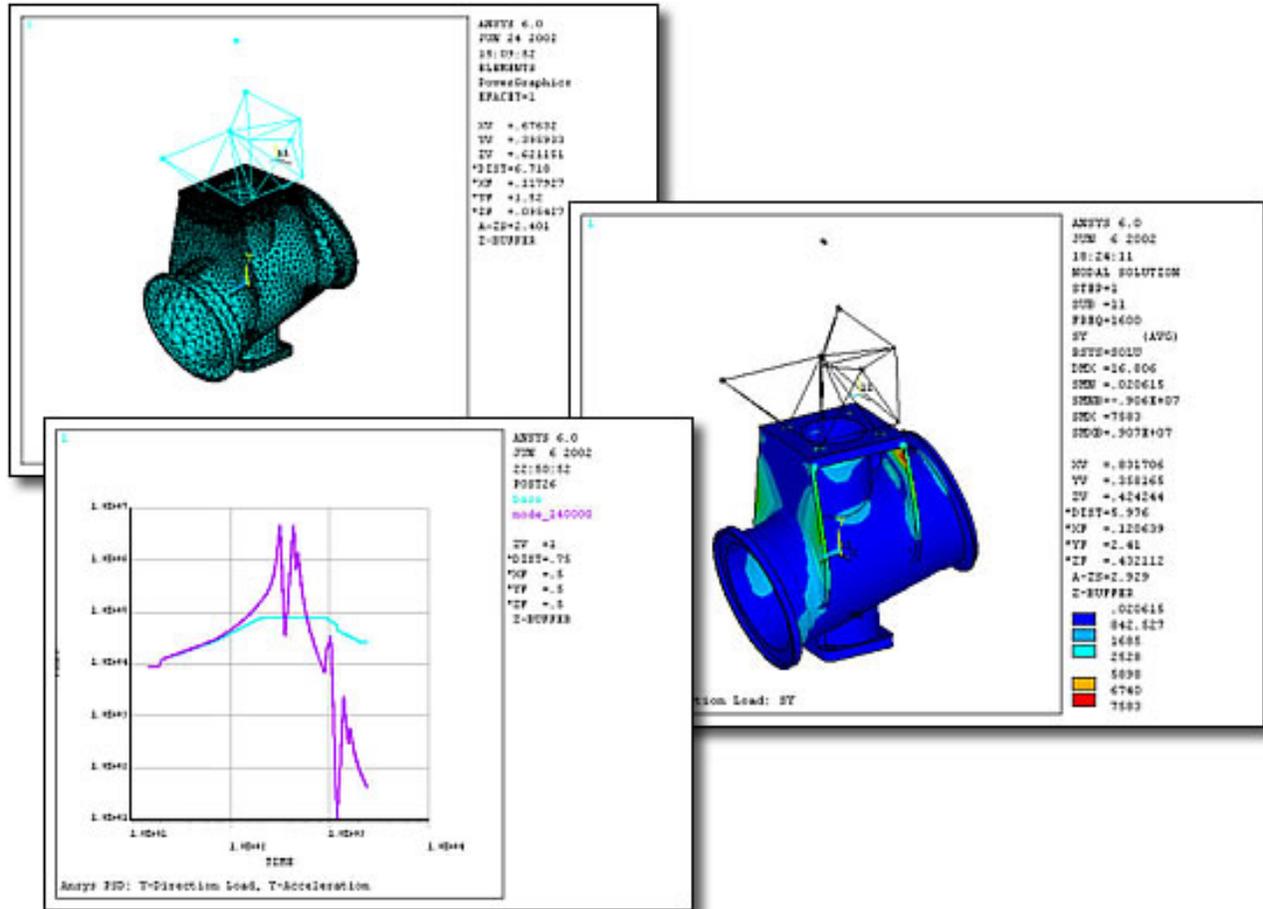


Example: Tempe Valve Body

The Focus



A Publication for ANSYS Users



A Word on Equivalent Stresses

- When requesting 1-sigma SIGE values, ANSYS actually gives RMS SIGE values
 - Documentation change request has been submitted
- There are two known ways to get SIGE values that can be effectively used in life analysis:
 - Monte Carlo
 - Well-known and robust
 - Very slow, lots of calculations per node
 - Siegelman-Reace
 - New method from SANDIA
 - Very fast and robust
- PADT has prototypes for both methods

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.