

The Focus

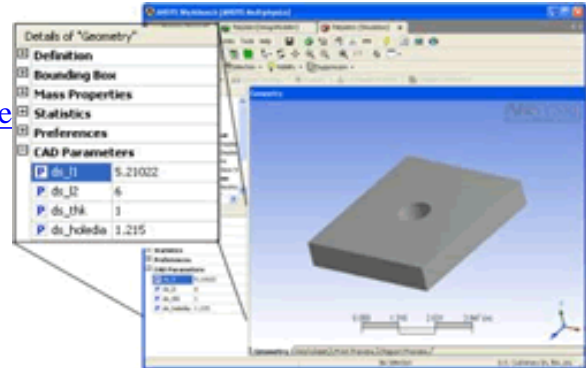


A Publication for ANSYS Users

Contents

Feature Articles

- [Workbench: Leveraging Parametrics through the Workbench Product Line](#)
- [Multiphysics Allows Dissimilar Meshes Between Fields](#)
- [ANSYS 8.0/8.1: Something for Everyone](#)



On the Web

- [Dimension 3D Printer](#)
- [Axel Products Physical Testing Services](#)
- [Free PADT Seminar: Geometry Modeling](#)



Resources

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

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



A Publication for ANSYS Users

Leveraging the Advantages of Workbench: Leveraging Parametrics through the Workbench Product Line

by Eric Miller, PADT

Overview

Over the past several releases, ANSYS Inc. has been rolling out more and more of their Workbench technology. There is already a lot of marketing and technical information available on the various aspects of Workbench, but PADT is still often asked: “Why should I use Workbench?” or “How can Workbench help me?” We will attempt to answer these questions with a series of articles on using these new tools to your advantage.

From the ground up, the tools in the Workbench are written to be parametric. This article will explain how this design focus makes Workbench more powerful than ANSYS in parametric design, and will also explain how to utilize these capabilities to make your design and analysis process much more efficient.

More Replacing Numbers with Variables

Long-time ANSYS users have become dependent upon the fact that almost any number in ANSYS can be replaced with a parameter variable. Workbench takes this one step further in that it allows you to pass those parameters from application to application and easily update a change through the modules that need it. In addition, the ANSYS development team is creating additional applications that specifically take advantage of these associative parameters to offer some powerful tools for parametric studies. Most importantly, you have access to your CAD system’s parameters if you created your geometry in the CAD environment.

The way a user makes a numeric value into a parameter is a little different than in ANSYS. Because the program is interactive, the user flags potential variables using the GUI. Values that can be set to parametric behavior have a small box to the left of them in the Detail View. A user simply clicks the box to make a parameter active. Figure 1 shows several typical Detail Views where parameters have been turned on. In DesignModeler, a D is placed in the box; in DesignSimulation, a P is used.

The Focus



A Publication for ANSYS Users

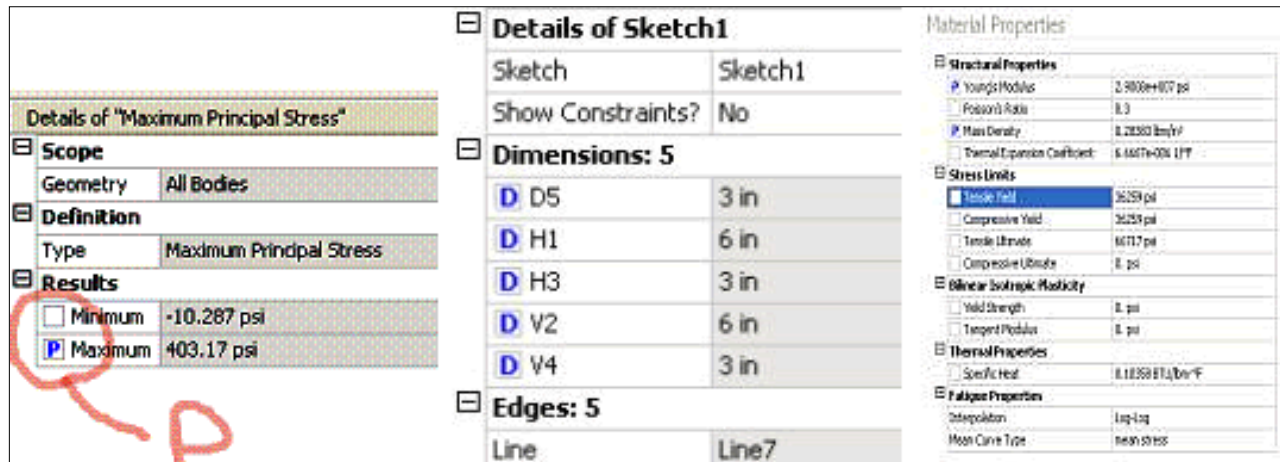


Figure 1. Typical parameter flags.

Sharing Values with Associativity

Once a parameter is established, it can flow up and down through related applications. As an example of this, we will turn to the ubiquitous “flat plate with a hole in it”, or FPWAHIT, example. Figure 2 shows where several parameters have been defined in Design Modeler to describe the 2D profile that this extremely complex part was modeled with. Note the “D” next to the parameters in the inset. When you click on the parameter, you are prompted for a name that will be given to the parameter in other applications. For this case we have chosen to use ds_ as a prefix. This will allow us to filter the parameters when we attach the geometry to the analysis tools, so we don’t end up with 250 parameters that we need to sort through. This “filter by string” works for geometry created in DM as well as geometry from external CAD packages.

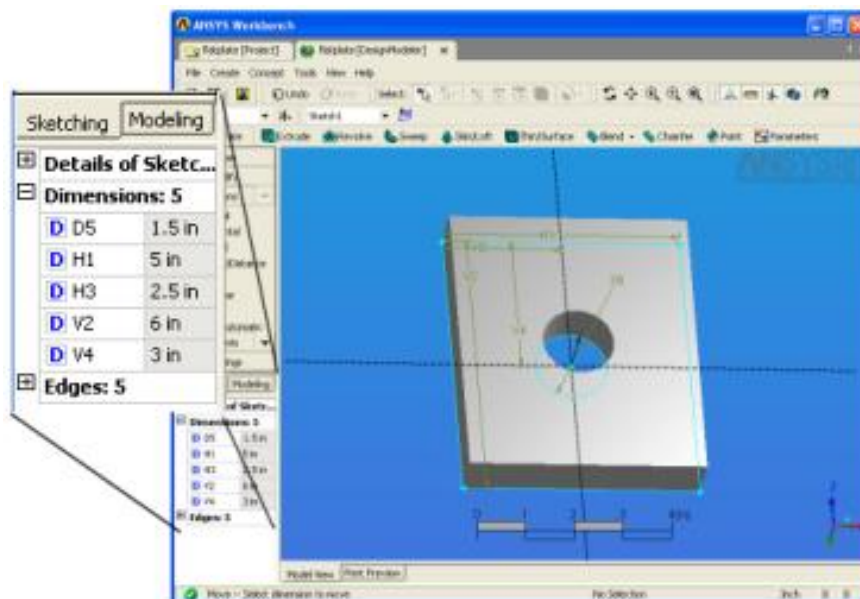


Figure 2. DesignModeler parameters.

The Focus



A Publication for ANSYS Users

Once in DesignSimulation (the module formerly known as the ANSYS Workbench Environment), the parameters are now available in the analysis environment. Figure 3 shows this for our example.

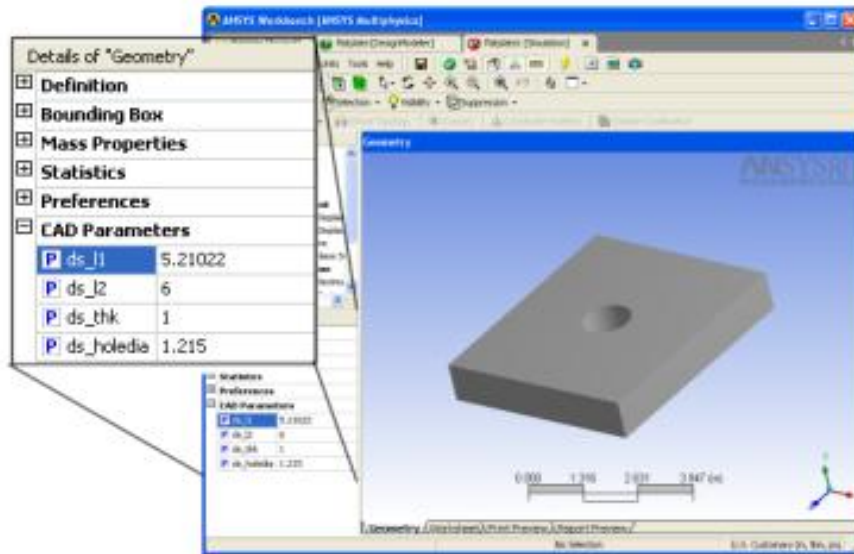


Figure 3. Model in DS with parameters sent over.

Now, to drive the CAD from within the analysis tool, simply change the parameter value then tell the program to update the CAD. Figure 4 shows this with the thickness changed from 1 to 5.

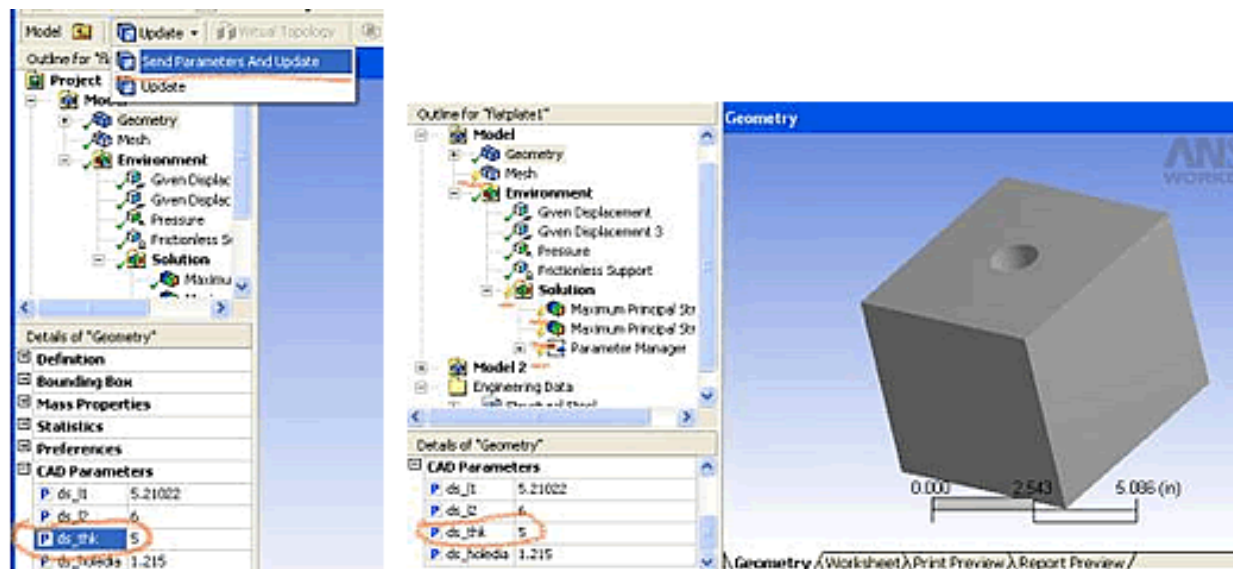


Figure 4. Changing CAD parameter in DS.

Also, do not forget that your result values can be turned in to parameters. Instead of comparing plots, it is better to insert a result item into the Solution branch using the Scope-Geometry option in the Detail View. In our highly illustrative example, selecting the surface that defines the inside of the hole creates a result set that gives the maximum hole stress, as is shown in

The Focus



A Publication for ANSYS Users

Figure 5.

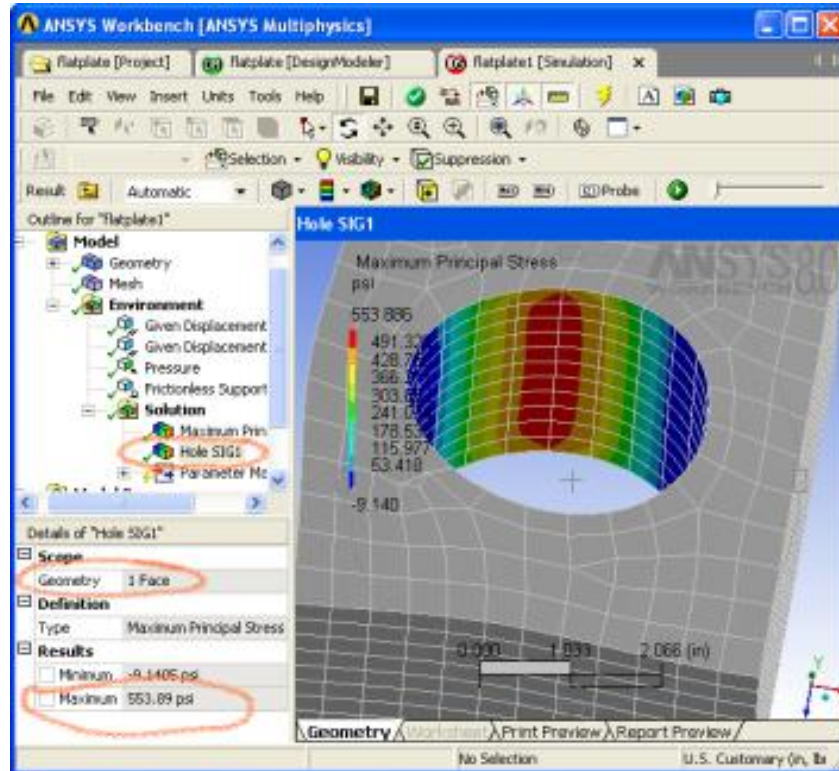


Figure 5. Result parameter scoping.

This shows how simple it is to do “what if” studies by changing parameters, hitting the lightning bolt to solve, and then reviewing the answers. One suggestion, though: if you want a record of the changes you made, duplicate the model in the tree and modify the copy. Now your HTML report will have the same plots and listings for each parameter set.

The previous example utilized the ANSYS geometry generation tool, Design Modeler, but the backwards associativity is valid for all major commercial CAD Packages, including SolidWorks, SolidEdge, UG, Pro/Engineer, and CATIA V5.

Exploring the Design Space

When you need to look at the effect of several design parameters on analysis results, the Workbench offers several tools. The first is built into Design Space and is called the Parameter Manager. To use it, simply insert a Parameter Manager object into your Solution tree. Any parameters you created (by clicking the box next to them) will show up in the table created by the Parameter Manager. Your current set of parameter values is already entered as the first row in the table. To look at other parameter sets, right click on the rows and “Add” another one. Now you can change the values. Once your table is populated, solve your model again and the result parameters will be updated. What is nice about this is that Design Space does not save all those results files, just the parameters you asked for. In this way you can do a simple DOE quickly. As with any table or listing in ANSYS, just right click on the Parameter Manager tree

The Focus



A Publication for ANSYS Users

leaf and choose “Export” to send the data to Excel for further processing. Figure 6 shows our sophisticated DOE study.

Parameter Manager

Definitions

The following table lists all parameters currently available in this branch of the Outline. To add a parameter, choose an object within this branch of the Outline, then check an item in its Details. Give each parameter a meaningful name.

Name	Refers To	Current Value
Geometry ds_thk	ds_thk,Geometry	1.
Geometry ds_holedia	ds_holedia,Geometry	1.5
Structural Steel Young's Modulus	Young's Modulus,Structural Steel	2.9008e+007 psi
Solid Mass	Mass,Solid	8.3713 lbm
Maximum Principal Stress Maximum	Maximum,Maximum Principal Stress	366.26 psi

Scenarios

The following table lists scenarios in which input parameters are varied to study the effect on output parameters. Right click on the grid to add, modify and delete a scenario. The Solve command calculates results for the current branch in addition to these scenarios. Solving many scenarios may take a lot of time. Uncheck any scenario that you do not want to include in the solution. A scenario in blue indicates that all input parameters match the values in the current branch in the Outline.

Run	Geometry ds_thk	Geometry ds_holedia	Structural Steel Young's Modulus psi	Solid Mass lbm	Maximum ...	Status	Comments
<input type="checkbox"/> 1	1.	1.5	2.9008e+007	8.3713	366.26	Done	
<input type="checkbox"/> 2	1.	1.5	2.9008e+007	8.3713	366.26	Done	
<input type="checkbox"/> 3	0.25	1.5	2.9008e+007	2.0928	352.11	Done	
<input type="checkbox"/> 4	1.	2.	2.9008e+007	7.9812	393.4	Done	
<input type="checkbox"/> 5	1.	1.5	2.8e+007	8.3713	366.26	Done	
<input type="checkbox"/> 6	1.	1.5	2.9008e+007	8.3713	366.26	Done	

Figure 6. Parameter Manager DOE study.

If you have a more complex model where the relationship between parameters is more complex, then DesignXplorer and DesignXplorer VT are for you. They are DOE/Probabilistic tools designed for extensive parameter studies. And since they are built into the Workbench, all of your parameters flow over seamlessly.

We could, and most likely will at some time, spend an entire article on these products. But since this is already too long, briefly:

DX and DX/VT present your input and output parameters in the context of a probabilistic study. It allows you to define parameters to work with and ranges to explore then calculates the response surface for that parameter set. The difference between DX and DX/VT is that DX does a solve for each parameter value combination (25 for 4 parameters) whereas DX/VT only does *one* solve and uses mesh morphing and ANSYS' SeriesXpansion technology to create the response surface. If you do a lot of DOE and probabilistic studies, then DX/VT is something you should really look at. Figure 7 shows the DX/VT interface and some typical results. Look for a more detailed discussion on these tools after version 8.1 is released.

The Focus



A Publication for ANSYS Users

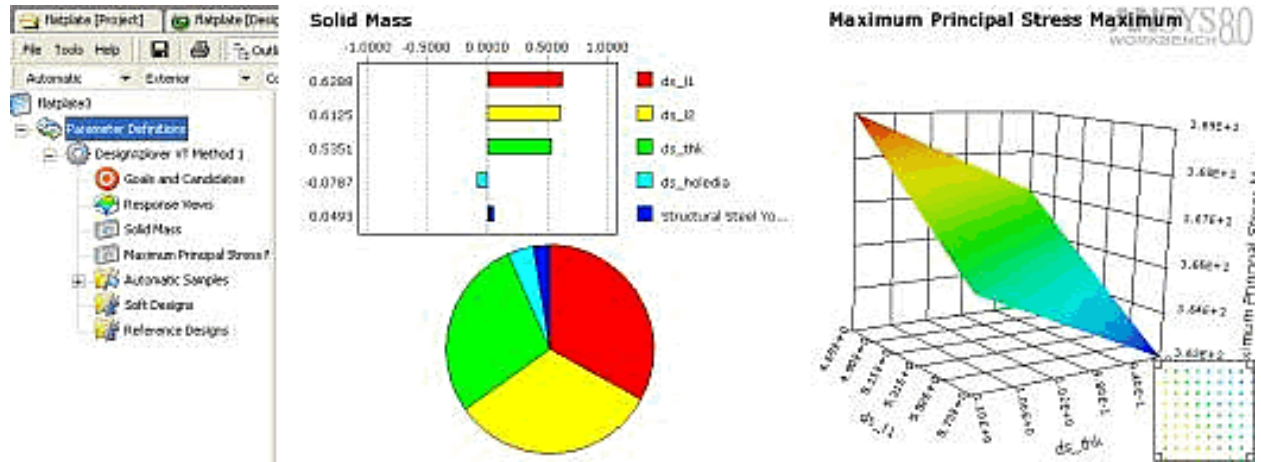
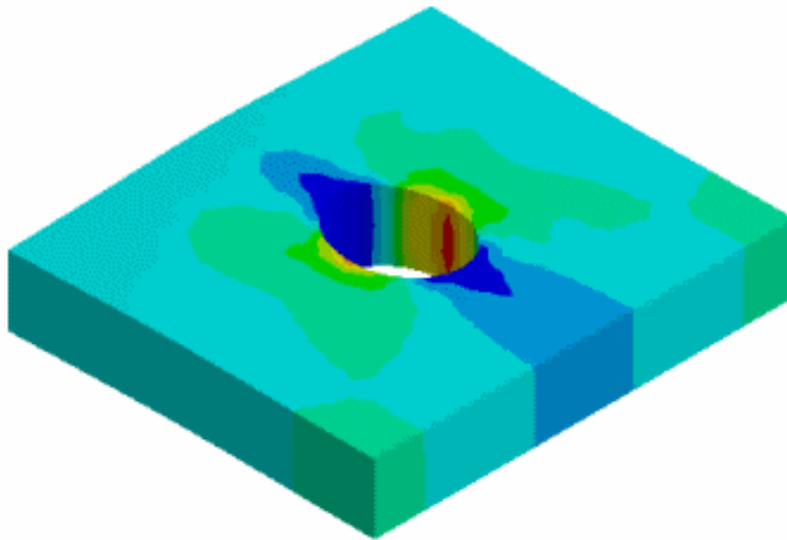


Figure 7. DX/VT interface and results.

Being Successful with Workbench Parameters

The biggest step in using this technology is changing the way you approach problems. Those of us raised on *get's and *vwrite's take a little time to get used to how simple and automatic this stuff is in the Workbench. Once you understand how parameters flow and update, you understand how simple things can be. As always, give it a try.



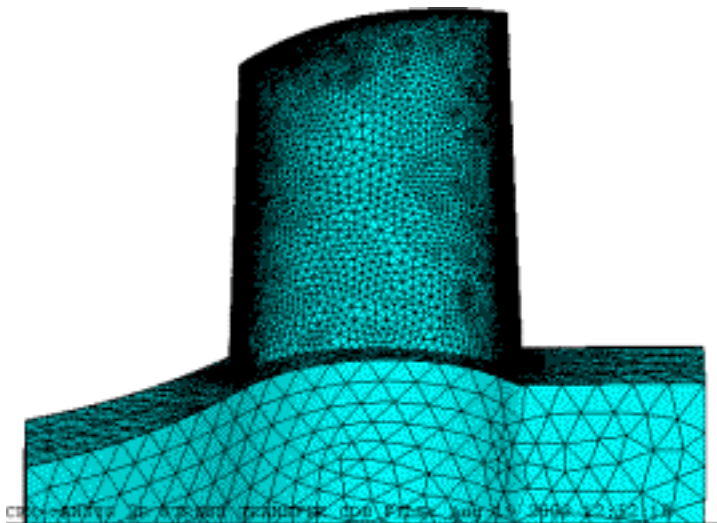


A Publication for ANSYS Users

Multiphysics Allows Dissimilar Meshes Between Fields

The 8.0 release of ANSYS is packed full of significant advances in all areas of the program, but one enhancement in the Multiphysics package is so significant that it deserves to be pointed out on its own before we talk about the other changes. This improvement is the ability in a Multiphysics analysis to transfer loads between two regions with totally different meshes. This addition puts the ANSYS Multiphysics offering even farther ahead of any other solution on the market today and makes this type of simulation easier to set up, computationally more efficient, and significantly more accurate.

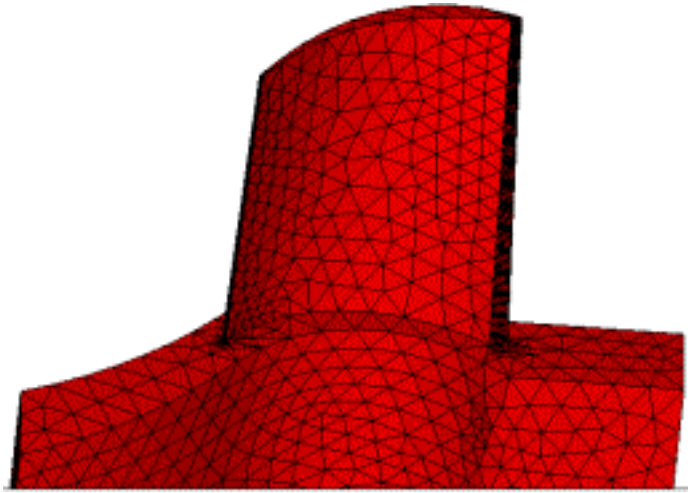
Anyone who has done a serious Multiphysics analysis knows that one of the more annoying aspects is trying to get a “compromise” mesh between two different regions. For example, MEMS devices often have a structural region and an electrostatic “air” region. Usually, you have to give up accuracy in your “air” region to allow for transition into a reasonable structural mesh. The mesh required for accuracy is totally different for the two areas, and when the structure starts to distort, the “air” mesh can often become distorted to the point where it loses accuracy. This capability was developed several releases ago for the FSI capability and has been generalized for all coupled-load solutions available in the Multiphysics product. Many users have been asking for a solution to the problem for some time.



The Focus



A Publication for ANSYS Users



The developers at ANSYS Inc. have responded in ANSYS 8.0 with powerful load interpolation algorithms that accurately convey loads across dissimilar mesh boundaries. They continue their history of giving users control over their analysis by not only enabling significant control over mesh densities, but also by providing the user with two options for controlling the accuracy of the load transfer. In the first option, Conservative Transfer, the user can specify an interpolation where the program will ensure that the total load (force, displacement, heat flux, etc.) is the same on both sides of the boundary. In the second option, Non-Conservative Transfer, the distribution of the load is maintained but the total loads may not agree. This former is best for fluxes and forces, and the latter for displacements and temperatures.

The bottom line is that one of the more time-consuming and frustrating steps in accurate system simulation has been removed with this new feature, and in our initial use of 8.0, PADT has seen significant improvements in accuracy and productivity from applying this enhancement. There is a good discussion on how this works in Chapter 4 of the 8.0 Coupled-Field Analysis Guide documentation.

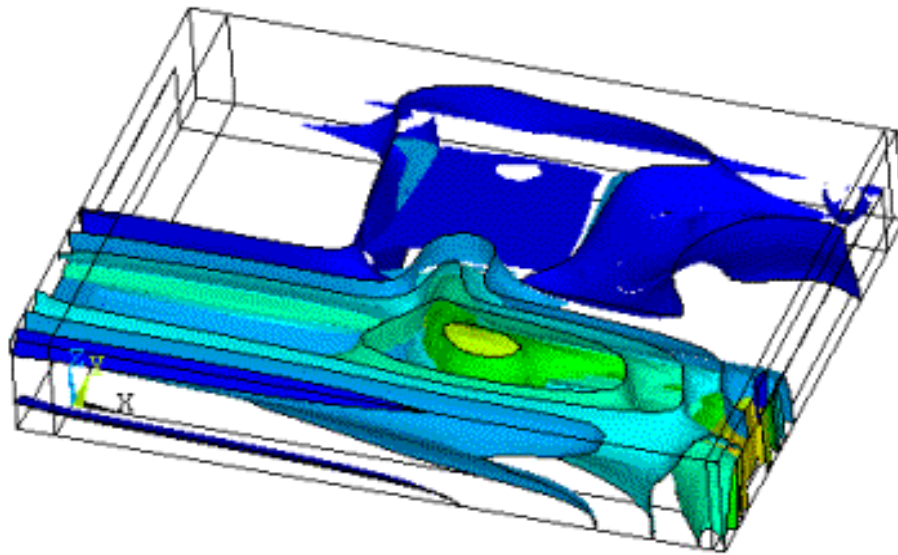
The Focus



A Publication for ANSYS Users

ANSYS 8.0/8.1: Something for Everyone

The recent release of ANSYS 8.0/8.1 is one of the most comprehensive releases of ANSYS in years. With new features in almost every area of the program, and significant additions to the Workbench product family, ANSYS Inc.'s development team has been very busy. This article will address some of the more important changes in 8.0/8.1, but please take the time to visit the links at the end for a more comprehensive review.



General Improvements

A considerable amount of effort has gone into improving existing capabilities to make them more robust, usable or powerful. The key enhancements in this are:

- Additional non-linear diagnostic tools to aid users in understanding and correcting difficult convergence issues. See NLDIAG, NLDPOST, NLHIST.
- Two new distributed memory solvers that allow users to solve using the PCG and JCG methods using multiple computers. PADT is finding that these solvers are much faster and more memory efficient than the DDS solver.



A Publication for ANSYS Users

- PADT's favorite general improvement is the ability to output ANSYS K, M, and C matrices into the industry standard Harwell-Boeing format. This allows users to create models in ANSYS, but use the matrices in programs like Matlab or NASTRAN.
- Surface calculations allow users to interpolate any valid result onto an arbitrary plane that passes through their model. Once the values are obtained, users can conduct detailed calculations, such as comparisons or integrations. PADT has found this to be a very simple way to get the total load across an arbitrary plane.
- Other enhancements:
 - Faster solvers across the board that use less memory
 - Better error messages that give useful feedback
 - Component manager
 - 4D and 5D arrays
 - New ANSYS launcher

Multiphysics

The powerful FSI solver that was introduced a couple of releases ago has been expanded to allow its use across all Multiphysics problems. This significantly simplifies how this type of analysis is done, and more importantly, allows the use of dissimilar meshes between different physics. This is a major improvement.

In addition, a new series of elements, 22X, has been introduced to provide modern coupled-matrix elements for solving Multiphysics problems without iterations. Currently they support piezoelectric and piezoresistive analysis. More are coming in future releases.

The Focus



A Publication for ANSYS Users

Structural

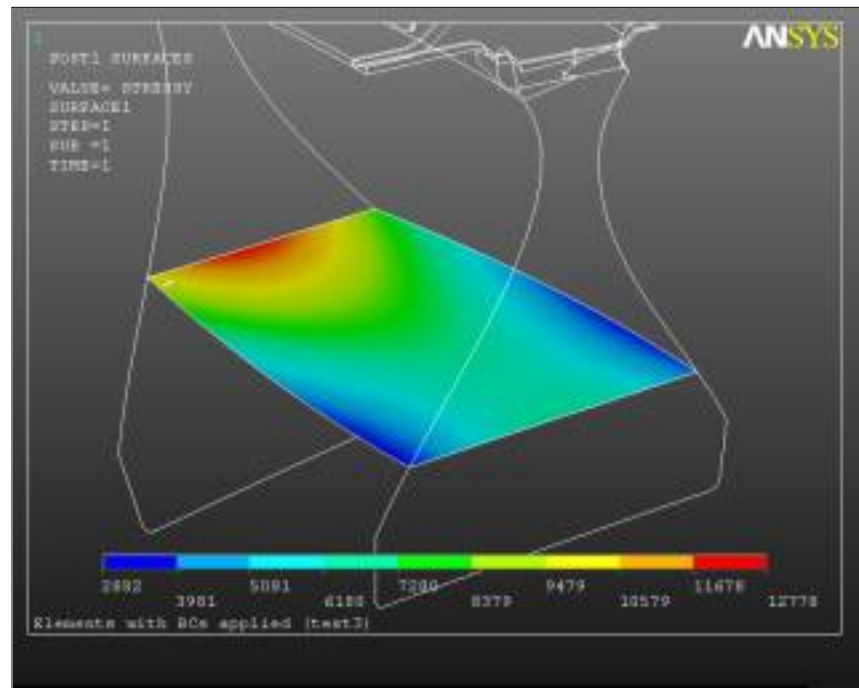
There are three major additions to structural analysis. The first is the addition of a full Lagrange Multiplier method to the generalized contact elements. This should provide very accurate contact analysis where penetration is a major concern.

The second is the addition of Component Mode Synthesis (CMS) to the dynamic analysis toolbox. This method, which dominates dynamic analysis on aerospace structures, allows users to solve very large problems accurately and quickly.

The final major enhancement, and the one that may have the largest long-term impact on users, is the addition of edge-to-edge and edge-to-surface contact. This will allow users to quickly and easily connect shell and solid meshes.

Additional structural enhancements include:

- Expansion of the material curve fitting capabilities beyond hyper-elasticity.
- Static cyclo-symmetry. This allows non-symmetric loading on a symmetric model.
- Support for shape memory alloys such as Nitinol.
- Complex eigenvalue solver. This is critical for non-symmetric damping and rotor-dynamics problems.
- Taper beams with BEAM188 and BEAM189.
- Kinematic joint options for the MPC184. This can be used to create simplified connections like universal, slider, or spherical joints.





A Publication for ANSYS Users

ANSYS LS-DYNA

Every year, more and more people find ANSYS LS-DYNA to be an indispensable tool for modeling highly non-linear time-dependent events. In this release, ANSYS now supports the 970 version of LS-DYNA. In addition, support has been added for the new 10-node Tet element that was recently introduced. This should eliminate the biggest headache with using this tool — creating a hex mesh.

ANSYS Workbench

The most noticeable changes to the product line are in the Workbench family of products. As promised, every release sees more and more functionality that allows the intermediate and advanced user to reap the benefits of the Workbench paradigm. There are far too many changes to list, but the important ones are:

- Major improvements to contact specification, control, and visualization
- A hex dominant mesher
- More detailed controls of mesh
- Display of pinball spheres
- Pretension bolts
- Thermal analysis on lines and surfaces
- Major improvements in handling of very large models (PADT has seen 10 million node models work just fine)
- Support for “enclosures” in support of field analysis
- VGLUE equivalent in DesignModeler

More Info

If you need more information or want to see what we left out, visit the [ANSYS web site](#). You can also contact PADT with your questions or review our [update seminar notes](#).



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
Feb '04	2/17	2/18	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	2/19	2/20	301	Heat Transfer	Irvine, CA
	2/26	2/27	102	Introduction to ANSYS, Part II	Irvine, CA
Mar '04	3/1	3/3	101	Introduction to ANSYS, Part I	Tempe, AZ
	3/8	3/9	203	Dynamics	Tempe, AZ
	3/11	3/12	501	ANSYS/LS-DYNA	Irvine, CA
	3/17	3/19	101	Introduction to ANSYS, Part I	Albuquerque, NM
Apr '04	3/29	3/30	301	Heat Transfer	Tempe, AZ
	4/5	4/7	101	Introduction to ANSYS, Part I	Irvine, CA
	4/19	4/21	101	Introduction to ANSYS, Part I	Albuquerque, NM
May '04	4/21	4/23	201	Basic Structural Nonlinearities	Tempe, AZ
	5/3	5/5	101	Introduction to ANSYS, Part I	Tempe, AZ
	5/6	5/7	102	Introduction to ANSYS, Part II	Tempe, AZ

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.