

## Part 2: DesignModeler and Simulation

As promised in last month's article, this month we will focus on the new developments on the Workbench environment for ANSYS 11.0. Although the majority of develops at ANSYS, Inc. work on the "core solver" technology, what we see and deal with on a daily basis are the changes made to the pre and post pro-

# This One Goes to 11

cessing tools, and especially Workbench. This release has lots to pay attention to.

A significant number of changes fit into the "I'd use Workbench more if..." wishes that many of us have. As with the changes to the core ANSYS product, there are far too many new features and changes to talk about in detail, instead, we will look at DesignModeler and Simulation, and next month we will look at the other Workbench modules.

## DesignModeler

Already a mature tool, DM continues to grow in functionality, driven primarily by user input. Besides the various new capabilities and features, we have found it to be more robust and a tad faster. The

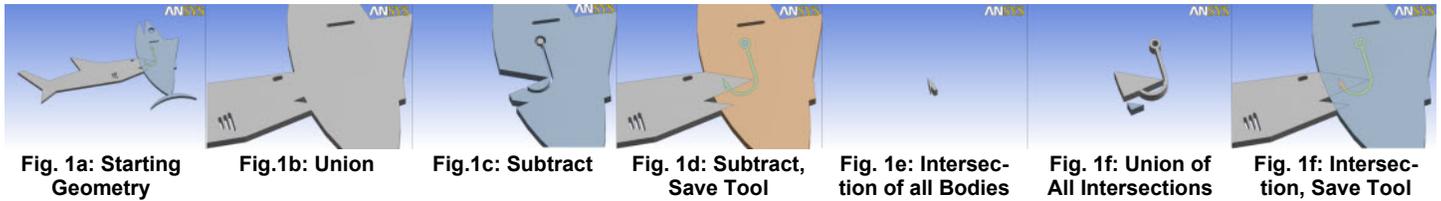
top three changes are listed below, but look for an article dedicated to DM enhancements for more breadth and depth.

### Booleans

One of the most requested additions to DM has been the ability to do Boolean operations. At V11 they are here, and added in a unique and powerful way. They work with solids or surfaces/2D areas and give you pretty much the same functionality that you should be used to from PREP7, but with some different commands.

The basic operations are: Unite, Subtract or Intersect. To give users more control, when you are subtracting or intersecting, you can say NO to "Preserve Tool Bodies." This keeps what remains of the cutting geometry around for future use. In addition, on the

*(Cont. on pg. 2)*



A PALMER/PLETSCH BOOK FOR CHILDREN



**My First Machine Sewing Book**  
STRAIGHT STITCHING  
by Winky Cherry

*By Doug Oatis*  
If you're like me, you've run into your fair share of "dirty" CAD

# Sewing in DesignModeler

tolerancing or an actual gap in the model. This was something easily fixed in ANSYS Classic by importing the areas, patching the hole, and creating a volume from the areas. In WB v11, you now have this capability in DesignModeler thanks to the Sewing Operation.

In DM V10, if you had a missing surface, the solid geometry import would fail and you would get an error message saying "No Valid Bodies Found".

Now, in V11, you can have DM import the areas if the solid generation fails. DM also has a new option where it will

*(Cont. on pg. 4)*

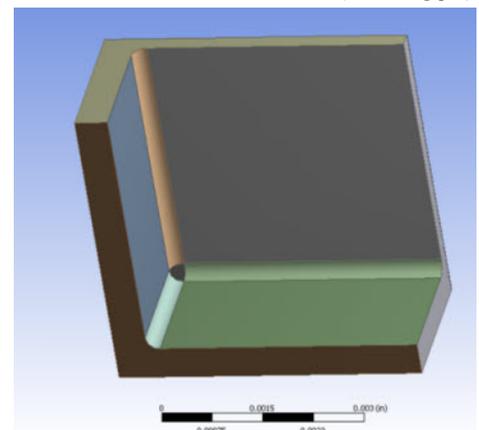


Fig. 1: Bad Geometry

Table of Contents	
This one Goes to 11: DM and Simulation	1
Sewing in DesignModeler	1
From /POST1 to Workbench	5
Advanced Search	4
Gravity in Workbench	4

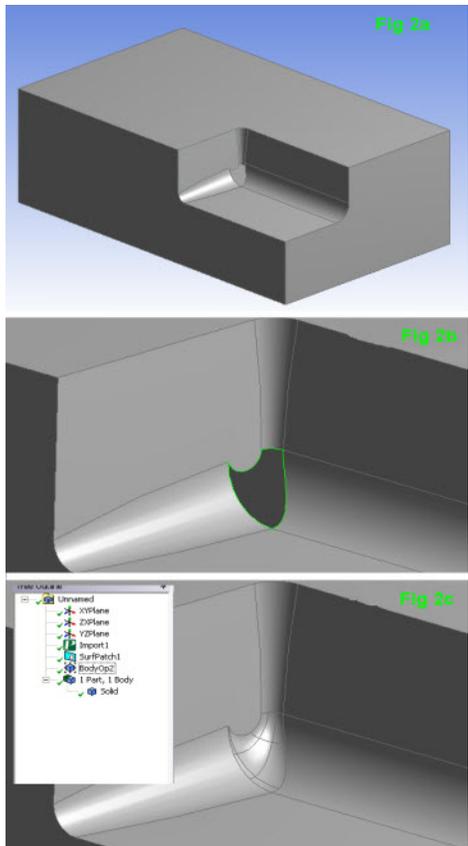
(ANSYS 11.0, cont...)

Intersect command, you can tell it to calculate the "intersection of all bodies" or the "union of all intersections." Figures 1a-1g show what happens if a shark, a hook and a tuna swim into DM and start playing with Boolean options.

**Geometry Repair Tools**

In an ideal world, all CAD files are perfect and read into Workbench without any problems. However, we don't live in that world so the DM team has started adding tools and features that let you import and repair real world geometry. The first important step that they took is that you can now do a "partial import" of geometry. What that means is that if your import fails, it reads in what it can and creates a surface body. Now you can go in and repair things.

The most important repair tool is the "patch" command that lets you click on a loop of edges, and the program sticks in a surface. Combine this with various delete and slicing operations and you can repair a lot of problems. Once you have all your services fixed up, you can then use the new "sew" command (a body operation option) to build a solid. Again, not all that different from what you do in PREP7, just simpler,

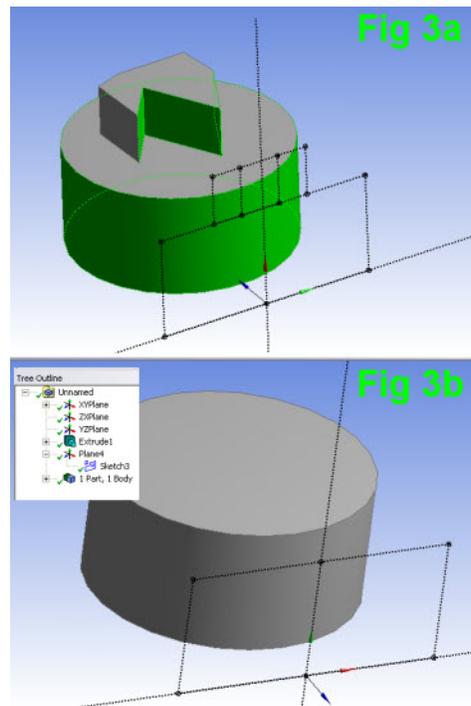


more robust, and fully associative and repeatable. Figures 2a-2c show a simple application.

**Sketch Projection**

One capability that exists in most CAD tools that was previously missing in DM was the ability to associatively project 3D entities onto the sketch plane. You use it by creating a plane that you want to project onto, right click on the plane, then choose Insert->Sketch Projection from the menu. Next, click the points, edges or faces you want and regenerate to get your new 2D sketch that projects the selected entities onto the plane you chose.

Figure 3 shows a good example- where both surfaces and edges are projected onto a new plane, ready to have new geometry associatively linked to it.

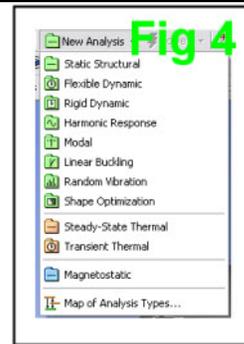


**Simulation**

The road to full high-end mechanical simulation pre- and post-processing in Workbench has been long and steep. Every release takes us a lot closer to the top, and V11 may be one of the most ambitious in this area so far. There is a lot of material, too much to detail, so the following is a summary of the changes that we feel will have the biggest effect on user productivity:

**Analysis Types**

A big fundamental change was the introduction of "Analysis Types". This tells



Simulation what type of analysis you are doing so that the user interface can filter and only show you stuff you need. When you pick the analysis type, it also inserts a setting and result branch into your model.

We have found it more intuitive than the old method and easier when you are doing multiple types of analysis on the same model. Figure 4 shows the options.

**ANSYS RST File**

Many long time Simulation users will rejoice in the fact that V11 now reads the ANSYS result file direction (insert sound clip of trumpet blaring?). This has several positive implications: 1) it allows access to more complex result types, such as shell element results, 2) it is much more compact when dealing with transient results, 3) you can more easily use the results from one run as a boundary condition on another. 4) It allows users to shuffle RST files around a little.

All in all this is not a flashy change, but will have major positive implications for complex simulation in the future.

**Multibody Dynamics**

The most significant effort from sheer development effort at V11 was the introduction of Multibody Dynamics. This is a major addition in capability that required changes all across simulation and in ANSYS as well. Most of the changes are subtle or behind the scenes, but some stand out and are deserving of special notice. Look for one or more future editions with detailed articles on this subject.

The most obvious is that all bodies now have a new option in simulation: "Stiffness Behavior." The default is flexible, but if you choose rigid the program will create a rigid mass, with properties based on your geometry and material properties, connected to other parts through MPC's. This greatly simplifies your model.

Another important change was a complete redo of what was the Contacts branch into a new branch called Connections used to define joints, springs and contact areas. The contacts are still automatic, but the joints

(Cont. on pg. 3)

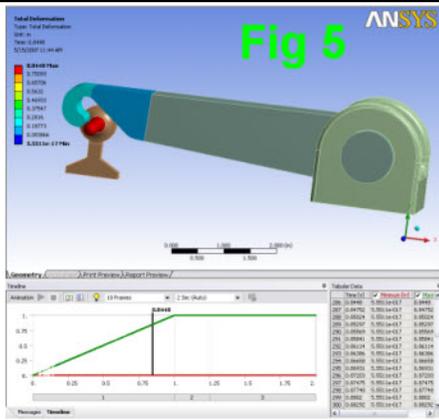


Fig 5

(ANSYS 11.0, cont...)

and springs are defined by hand. Of course, the solution options and post processing have also changed to support this important new capability. Figure 5 shows a simple latch model, and you can download an animation and the model itself at [ftp.padtinc.com/public/downloads/latch2\\_wbmdl.zip](http://ftp.padtinc.com/public/downloads/latch2_wbmdl.zip)

**Meshing Improvements**

Meshing and meshing control are two areas experienced users were always beating on development to improve. V10 saw some important steps in this area, and even more are now available in V11. The most important is a new option called "Physics Preference." With this, you tell the mesher the type of problem you are trying to solve and the program changes the mesher defaults and checking based upon what type of mesh each physics requires.

With the introduction of more non-linear capabilities in Workbench, it was time to tighten down the shape checking. So, under the advanced meshing controls you can turn on aggressive checking, which is a lot like the default checking in ANSYS PREP7.

Moving down the list, the next important change consisted of a lot of enhancements to Sweeping, allowing you to hex mesh a much larger number of geometries than in V10.

Newly exposed in V11, Inflation layer meshing is something that the CFD world has used for years to get a very refined mesh on a surface of a tet mesh. You basically mesh the surface with triangles then grow wedges inward in thin layers to get refinement normal to the surface. Then you fill in the rest of the volume with tets. In V11 you can use this advanced feature for CFD, EMAG and stress meshes. Some preliminary testing has shown it to be a powerful

way to get very good refinement in key areas.

The last meshing change to really highlight may be the simplest, but the most significant. You can now specify edge biasing on mesh controls on edges. For CFD and many stress models, this has been a long-needed functionality.

**Tables and Functions for Loads**

There are certain capabilities in ANSYS that are very powerful, but require a certain amount of skill and patience to use, and mastering them gives you a strong sense of superiority. Unfortunately, those pesky developers continue to take advanced capabilities and making them easier to use - and adding tables and function loads to Workbench is a good example. You can now specify loads that vary with temperature or time using functions or tables. And the tables can be stored in engineering data. To top it off, the values are shown right in the main window as a graph to make sure you did everything correctly. Look for spatial variations in the next release. Figure 6 shows a function and the table/graph it produces

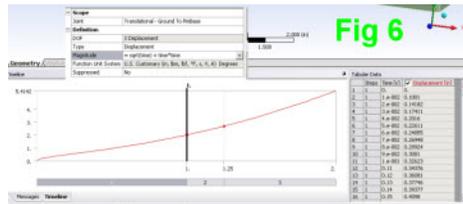


Fig 6

**FSI**

As PADT's sales team goes out with their carpet bags beating on doors, we run across customer after customer looking for powerful yet easy to set up Fluid-Solid Interaction (FSI) modeling. ANSYS has had this capability for many years, but only with the introduction of CFX and Workbench has it become accessible beyond the specialist user. In V11 more functionality has been added in several ways. The one that will be used most is one-way load transfer from a CFX solve to an ANSYS solve. At V11 Temperature and Convection join Pressure as the different types of surface loads that you can transfer.

For two-way FSI there has been a rewrite of the interface on both the Simulation and CFX side to make it even easier. Look for specific articles on this in the focus as well as tutorials on the ANSYS web site.

**PSD**

Another core ANSYS capability moves to workbench with Power Spectral Density (PSD) simulation. In the electronics and aerospace industries, as well as many civil engineering applications, you use PSD to get the statistical response of a structure to a random vibration input. The implementation is pretty complete and includes support for pre-stressed modal, different units, scaling to get 2, 3, N sigma, and the all-important Segalman-Reese method for calculating accurate 1-sigma Equivalent Stresses.

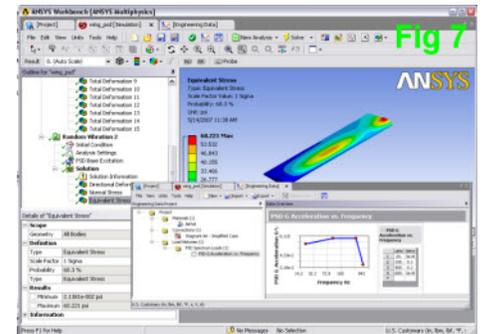


Fig 7

**Shell Element Result Support**

Yet another core functionality showing up at V11 is the ability to plot shell results on the top, bottom or middle of shells. Nothing really that fancy but if you need it, you have to have it and now you don't have to post-process in POST1 to get your bending stresses.

**Chart Tool**

The last enhancement we are going to talk about is the first tool in Simulation for what can be called advanced post-processing. The Chart Tool makes nice X-Y graphs of results information from any multi-step simulation. This may be another relatively simple addition to the program, but one that is needed to allow users to stay in Simulation for a complete analysis.

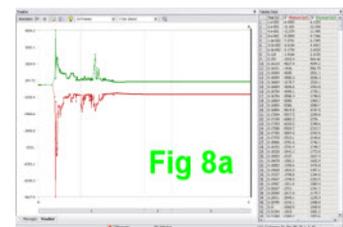


Fig 8a

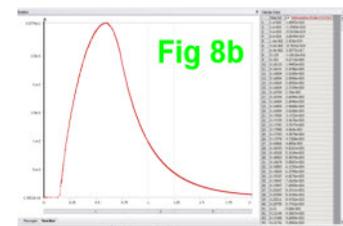


Fig 8b

(Sewing, cont...)

try and stitch the surfaces together automatically. This feature (on by default) will cause the model with the missing corner fillet to import as shown in Figure 1.

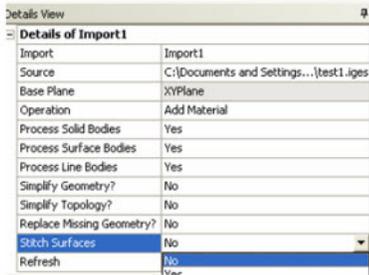


Fig. 2: Turn off Stitching on Import

As you can see, it created a solid and surface body. This isn't quite what we wanted, so you can turn off the automated stitching in the details window (Fig. 2) of the import object. You will then get the quiltwork of surface areas imported.

Next, use the 'Surface Patch' underneath "Tools", and select the three edges

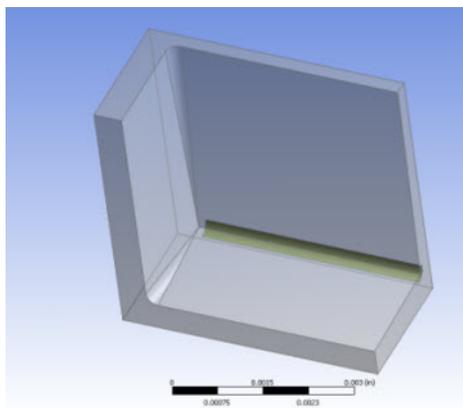


Fig. 3: Surface Bodies Ready for Repair

Details View	
Details of BodyOp1	
Body Operation	BodyOp1
Type	Sew
Bodies	12
Create Solids?	Yes
Tolerance	Normal

Fig. 4: Bad Geometry

es that define the gap. Finally, insert a body operation, and change the 'Type' to "Sew". Select all the surfaces, make sure to set 'Create Solids' to "Yes". Click Generate and you'll see you now have a solid body ready for meshing in Simulation.

In case you're curious about what happens if 'Create Solids' is set to "No", it simply joins all the surface bodies together. In the parts list you will just see a single surface body (think of it as unfreezing multiple surface bodies together).

Not my finest mesh ever...but it's better than fighting a surface model to repair

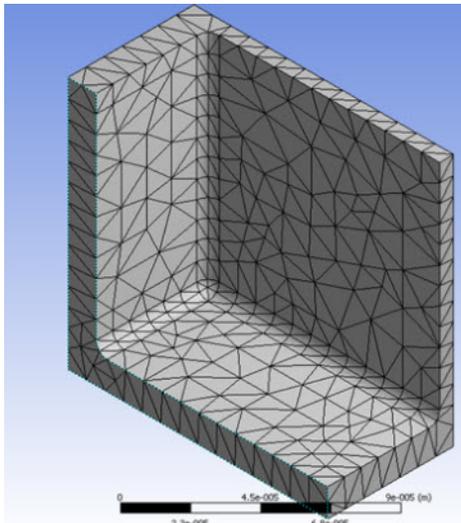


Fig. 5: The Mesh

### Einstein is Spinning in His Grave: Gravity in Workbench

Everyone should be aware of a simple little change in Simulation for V11.0. In ANSYS and Simulation prior to V11 you specified gravity as an acceleration acting on your geometry. The whole point of general relativity was that there is no way for an observer to tell the difference between being accelerated at 32.2 ft/sec<sup>2</sup> or standing on the earth. In fact, gravity is the acceleration caused on an object as it travels through time, duo to the warping of the space-time fabric by the mass of a large object. Therefore gravity was defined as an "up" acceleration. But this confused a lot of people so bowing to public pressure, and ignoring the equations that govern the universe, Simulation V11 specifies gravity in the direction of the force, or down. We shall see if the International Physics Congress lodges an official protest, but until then remember that it is different in Simulation than ANSYS. (Is it obvious that we don't agree with this change?)

### Your Search for Advanced Searching is Over

Many of us spend much of the day roaming around the ANSYS help files. Even after learning how to use Booleans in the search

(<http://www.padtinc.com/epubs/focus/common/focus.asp?I=40&P=article3.htm>), we still wanted more advanced searching capabilities. The often unnoticed folks in documentation have come to our rescue with an modern web-based search engine on the ANSYS, Inc. customer portal:

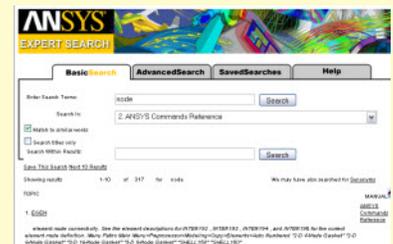
<http://expert.ansys.com/ExpertHelp/>

Some of the great features are:

- Choose which manual to search
- Search within results
- Search titles or content
- Match on similar words
- Advanced search building tools with controls like "must/must not contain," "case sensitive," "Match Synonyms"
- You can also save common searches to come back to later. There is even a decent "help" manual for the Expert Search Capability.

As an example, we got a question on how to define the normal direction of a pre-tensioned entity. If I simply search all for "pretension normal" I get 576 references. If I do "pretension" I get 51. Doing a search within results on "normal" I get 16. Doing one more "search within" on direction gets me down to 14. If I cut that down to just the ANSYS Basic Analysis Guide I get only 1 listing. I could of done the same thing on the AdvancedSearch page with even fewer steps

So next time you find yourself looking for something in the help, don't bring up the old interface. Go to the customer portal (you will need a login) and use the new advanced help.

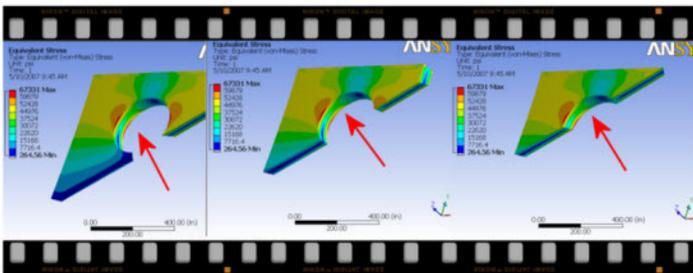


# From /POST1 to Workbench

By Rod Scholl

With release 11.0, you can now take your ANSYS results and post-process them in Simulation!

Every time I rotate an image in Workbench's Simulation I am freshly amazed. After years of waiting for redraws in ANSYS, it just seems too good to be true. And when I turn on something slick like Simulation's sliced section abilities (a dynamically updating capped-hidden display) and can drag the slice plane back and forth through result contours in real time... it really makes for better understanding of a state of stress...



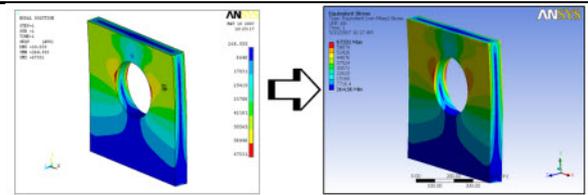
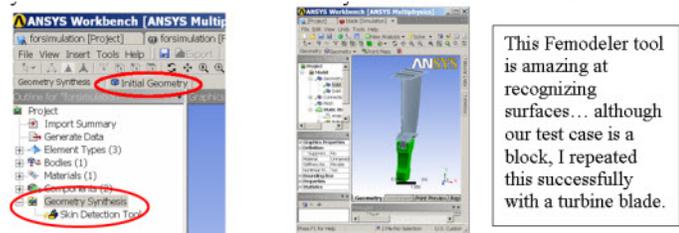
But hopefully I don't have to sell you on some of the great and fast post-processing features of Simulation - rather let's detail the process of getting your ANSYS created and solved model in to Workbench -- its not just a push of a button:

1. In ANSYS write out your mesh as a .cdb file. Use CDWRITE,DB,, -- the (CDWRITE,GEOM option will not work properly.)
2. In Workbench, launch FEModeler specifying the .cdb file from step 1.



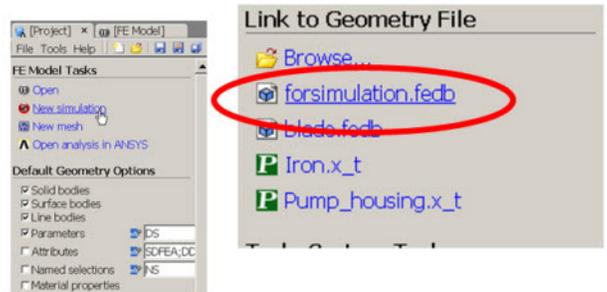
3. Create the geometry. Because Simulation expects a solid model on which to apply BC's there must be some present. Do this by highlighting the Geometry Synthesis branch and select Initial Geometry. Save the file.

4. If you are still in the same project database, you can simply click on "New Simulation". If you are reopening WB and specifying an

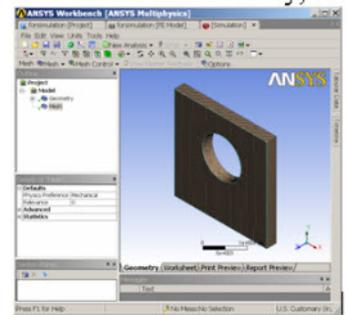


existing FE Model session, then on the project page select (or browse to) the desired .fdbb file which you preciously created. Then on the Workbench project page, you can click on New Simulation.

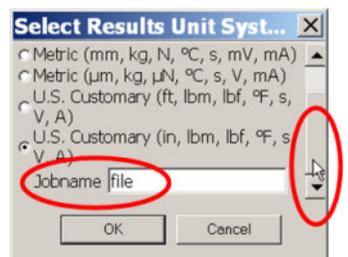
5. Okay, now we have the geometry in Simulation. Catch your breath. (I didn't say this would be easy, hehe.)
6. To read in the results, we have to prep Simulation for the incoming information. So we will create a dummy branch (of the same



physics). Also, you have to specify a type of result (similar to Simulation behavior before release 11.0) such as a single Equivalent Stress plot. Finally, there must be a force of some sort applied under the environment branch. For our structural model, I applied a dummy force of 5 lbs to an arbitrary surface.



7. Now, along the top menu under Tools, the "Read ANSYS .rst" file is no longer grayed out. Note that you will be only specifying the folder, and a jobname in the unit system dialog box (may have to scroll down to find the jobname specifier). Also, this folder is going to get duplicated on your haddrive, so you might want to stick the .rst in a empty folder to avoid delays and wasted space. Simulation now begins converting the .rst to the format which Simulation uses.



8. Once complete... you can save the Simulation database, and your results can be operated on normally. Well, relatively normally. You will notice that the lightening bolt color scheme doesn't match - but add postprocessing items and retrieve results for them and the plotting works fine.

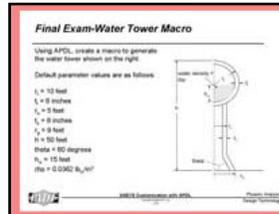
As easy as 1,2,...8! (Expect it to get more streamlined in future releases.)

## News - Links - Info

- ANSYS, Inc. does a 2-1 Stock Split [\[story\]](#)
- ANSYS, Inc. Recognized on list of Top R&D Sponsors [\[story\]](#)
- FLUENT for CATIA V5 Released [\[story\]](#)
- Not using the customer portal? You should. There is a ton of useful information and a growing on-line community: [www1.ansys.com/customer/](http://www1.ansys.com/customer/)
- ANSYS, Inc. has combined their ANSYS and FLUENT magazines into a new publication: **ADVANTAGE**  
This first issue has lots of technical content and a special supplement with examples on how people use simulation in the sports and leisure business. Subscribe to the print version or view the on-line version at: [www.ansys.com/magazine](http://www.ansys.com/magazine)



Upcoming Training Classes					
Month	Start	End	#	Title	Location
May '07	5/21	5/22	105	ANSYS WB Sim Struct. Nonlin.	Tempe, AZ
June '07	6/4	6/6	101	Intro. To ANSYS, Part I	Tempe, AZ
	6/11	6/12	201	Basic Structural Nonlinearities	Tempe, AZ
	6/13	6/14	204	Adv. Contact & Fasteners	Tempe, AZ
	6/21	6/22	301	Heat Transfer	Tempe, AZ
	6/25	6/26	107	ANSYS WB DesignModeler	Tempe, AZ
July '07	7/9	7/10	104	ANSYS WB Simulation - Intro.	Albq, NM
	7/12	7/13	105	ANSYS WB Sim Struct. Nonlin	Albq, NM
	7/19	7/20	801	ANSYS Customization w/ APDL	Tempe, AZ
	7/23	7/24	203	Dynamics	Tempe, AZ
Aug. '07	7/26	7/27	102	Intro. To ANSYS, Part II	Tempe, AZ
	8/1	8/2	104	ANSYS WB Simulation - Intro	Tempe, AZ
	8/6	8/7	152	ICEM CFD/AI*Environment	Tempe, AZ
	8/9	8/10	202	Adv. Structural Nonlinearities	Tempe, AZ
	8/13	8/14	501	ANSYS/LS-DYNA	Tempe, AZ
	8/16	8/17	604	Intro to CFX	Tempe, AZ



Learn APDL the "Home Schooling" Way  
Purchase PADT's  
**Guide to ANSYS Customization with APDL**

The Focus is a periodic publication of Phoenix Analysis & Design Technologies (PADT). Its goal is to educate and entertain the ANSYS user community. More information on this publication can be found at: <http://www.padtinc.com/epubs/>



# The Shameless Advertising Page

