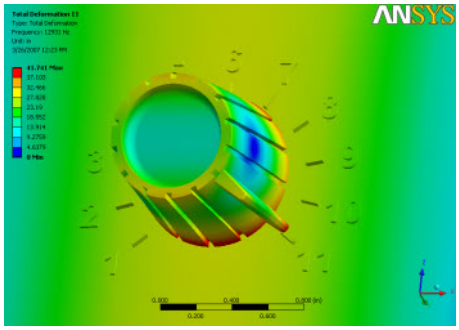
The Focus

PADT

June 22, 2007 | A Publication for ANSYS Users | Issue 58

This One Goes to 11

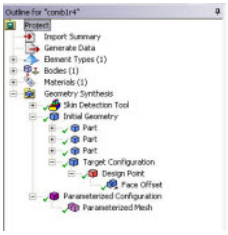


Part 3: The Rest of Workbench

As we work our way through the significant amount of enhancements in the 11.0 release of ANSYS, we now come to the parts of Workbench that are underutilized and underappreciated: FE Modeler, Meshing, CFX-Mesh and DesignXplorer. All of these are useful modules that work with outside programs or Simulation to provide greater functionality. In our support work, we have found that if users knew these modules better, they would find many of the capabilities that they want in Workbench already exposed.

FE Modeler

This module is where the developers put functionality that is required to deal with mesh-only models. Primarily used for legacy models or translating to/from other software, at Version 11 it really comes into its own and offers some very useful new functionality. We hope to have some tutorials on the key functions before the summer is over.



Geometry Synthesis

Every once in a while a feature gets added that changes everything, and this is one of those features. Once you have read a mesh into FE Modeler from ANSYS, NASTRAN or ABAQUS, you can convert the outside surface of the mesh into geometry. Don't get too excited, it is not NURBS geometry that you can then edit in a CAD tool or DesignModeler. It is

(Cont. on pg. 2)

Getting in Touch with ANSYS Contact

By Doug Oatis

One of the most popular questions I get asked as an ANSYS user, besides "Why are you so awesome, Doug?" is about contact in ANSYS. My answer to that question (about contact, because I can't explain the first), is "how much time do you have?"

Contact has become increasingly easier to implement in both ANSYS Classic and Workbench. In ANSYS Classic, you can use the contact wizard to easily define surface to surface contact. In Workbench, you can automatically define it for an imported assembly. After you've set it up (or had it setup by Workbench), you then face the issue of trying to decide which behavior, formulation, offset behavior, etc. to use. Table 1 is my quick explanation of what all those options mean.

As you can see, there are two basic motions that are described by various behaviors. The contact pair can either open, close, or slide relative to each other. Depending on which behavior you set, there are different ways that ANSYS enforces these relationships. These are shown in Table 2.

Table 1: Contact Names and Behavior

Name	Gap Open/Close?	Sliding Allowed?
Bonded	No	No
Rough	Yes	No, infinite μ
No Separation	No	Yes, μ = 0
Frictionless	Yes	Yes, μ = 0
Frictional	Yes	Yes, if $F_{sliding} > F_{friction}$

Table 2: Formulation Options

Name	Equations Solved	Default Contact Detection
MPC	Generates CE's	Nodes
Pure Penalty	F = Kx	Gauss
Augmented Lagrange	F=Kx + λ	Gauss
Normal Lagrange	Nodal Pressure DOF	Node

The MPC method simply writes constraint equations between the contacting bodies. Pure Penalty enforces a contact stiffness for a given penetration. The Augmented Lagrange is similar to Pure Penalty, except that it includes a factor for contact pressure to eliminate some chattering effect. The Normal Lagrange adds a Contact Pressure DOF to the model. In terms of run time, the formulations shown are listed in increasing run time.

(Cont. on pg. 4)

Table of Contents

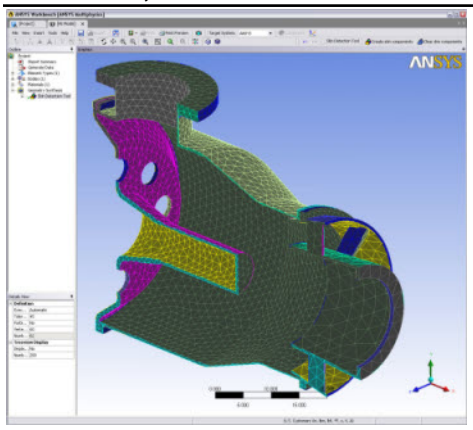


Fig. 1: Extracted Geometry

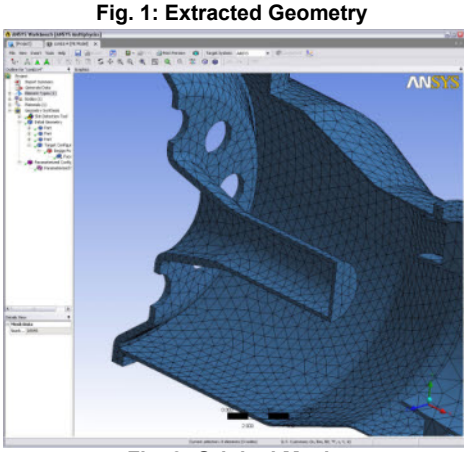


Fig. 2: Original Mesh

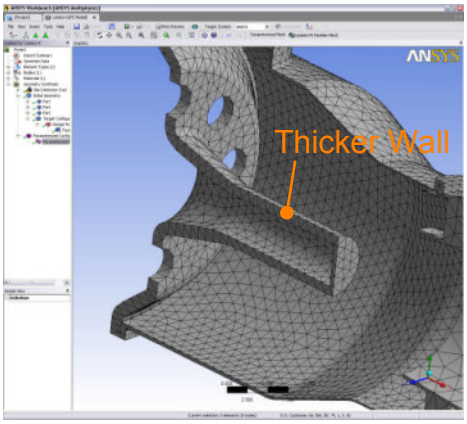


Fig. 3: Modified Mesh

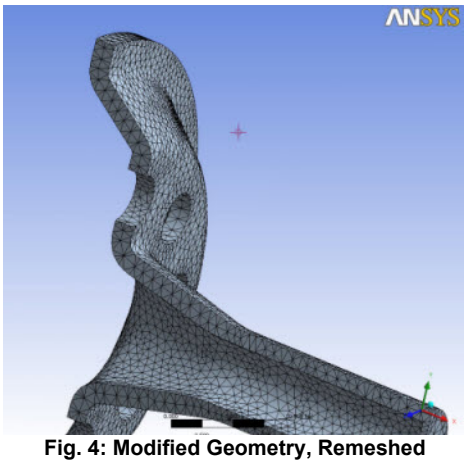


Fig. 4: Modified Geometry, Remeshed

(ANSYS 11.0, cont...)

a faceted surface representation that you can either send directly to Simulation or Advanced Meshing for remeshing, or that you can modify in FE Modeler using mesh morphing.

To try this out I found an old turbine-engine aft bearing mount from ANSYS 5.3 on a 10 year old archive CD. I read that into 11.0, wrote out a CDB, then read that into FE Modeler. Figure 1 shows how the Skin Detection tool builds a surface model from the mesh, Figure 2 shows the original mesh in FE Modeler and Figure 3 shows what happens if the Mesh Morpher (what used to be paramesh and is now in FE Modeler) is used to offset one of the surfaces. We have found this tool to be useful on simple to moderately complex geometry, and especially good when the original mesh is relatively refined. Figure 4 shows the key part, remeshed in Simulation.

Mesh Metrics

The mesh metrics area has been enhanced to include a variety of checks and great graphics feedback. The checks include: Element Quality, Jacobian Ratio, Aspect Ratio, Warping Factor, Parallel Deviation, Maximum Corner Area, Volume/Area.

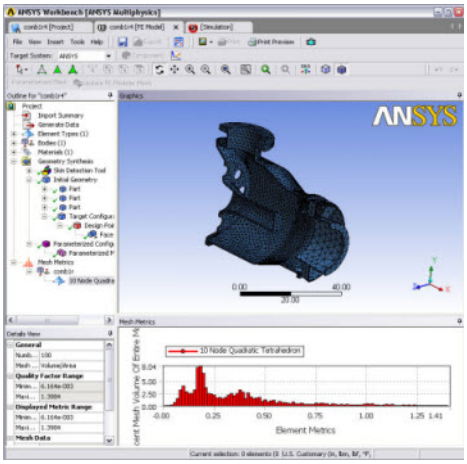


Fig. 5a: Mesh Metrics

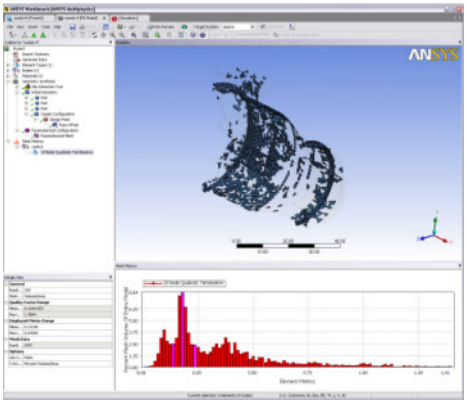


Fig. 5b: Histogram Selection of Quality

If you click on the bar in the histogram then the elements that fall in that range plot within a transparent view of your model, a nice way to see where you have problems. Figures 5a and 5b show typical plots.

Better Import/Export Support

A general area of improvement for V11 is improved support for more entities in the NASTRAN and ABAQUS translators. The output also gives better feedback on entities that are not read in. You can now also read ANSYS CDB files directly in, a beta feature at 10.0. To learn what is and what is not supported, read the online help.

Meshing

The Unified Meshing Initiative at ANSYS, Inc. has been working hard at pooling various meshing technologies inside workbench, and you will see the biggest changes in the Meshing module. Most of the changes are minor, but they add up to a more powerful and useful tool.

The first thing you will notice is that the look-and-feel is much more like simulation. In fact, it is the same as Simulation except there is no solving or boundary condition insertion capability. Now when you want to start meshing, you choose meshing algorithms by inserting a meshing method under the mesh branch. Here you can pick from 5 methods: Tetrahedron (Patch Dependent), Tetrahedron (Patch Independent), Hex Dominant, Sweep and CFX Mesh. You can

(Cont. on pg. 3)

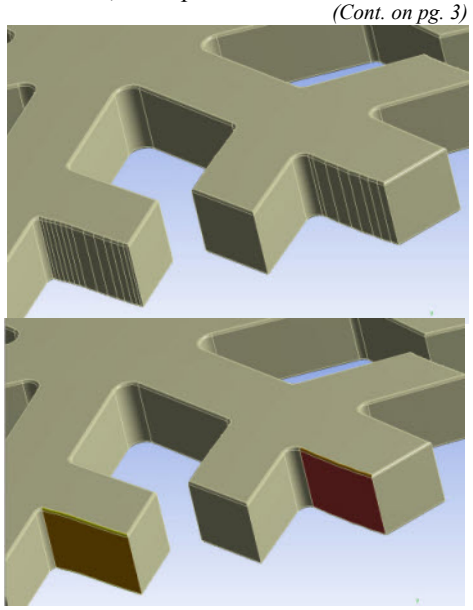


Fig. 6a,6b: Virtual Topology

(ANSYS 11.0, cont...)

then set nice options like global sizing, transition speeds, smoothing and much more. Your choices vary depending on which algorithm you choose. Look for the area to evolve in future releases into a more general high-end meshing tool for going beyond what Simulation has to offer, with a new product name, or maybe not depending on how usage drives development.

CFX-Mesh

This CFD centric meshing tool has also seen some benefits from the unified meshing initiative.

Full Virtual Topology Support

A major advantage in Simulation has made its way into this module in the form of Virtual Topology Support. Now you can do virtual merges of surfaces and curves inside the meshing module before you mesh. Figures 6a and 6b show a model (the snowflake used in our Christmas greetings) and the tiny little surfaces that make meshers go nuts. Using the Virtual Topology these can be merged into nice fat surfaces.

Edge Spacing

The edge spacing object now allows you to specify all of the options you have for faces, plus the ability to specify a bias when using patch dependent meshing. Not very fancy, but again, a much needed feature. Figure 7 shows an example.

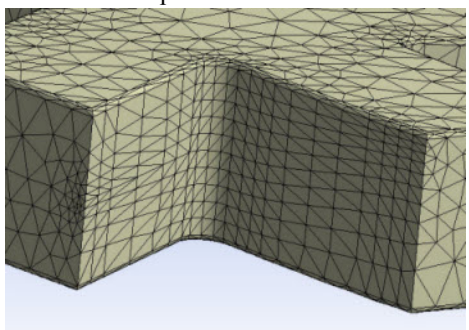


Fig. 7: Edge Spacing

Other

Some small changes include the addition of a Mesh Generation Interrupt button to stop lengthy meshing sessions and support for named selections that come in from Design Modeler. Lastly, parallel meshing has been added for CFD style meshes as a beta feature at 11.0. See the help for how to set it up, it is currently really only for power users who can control their compute environment. But look for this capability to improve over releases as users demand larger meshes and shorter meshing time.

DesignXplorer

The last module we will cover this month is DesignXplorer. This area of Workbench is where all the parameter variation capabilities pay off because it automates the process of DOE studies, Optimization, and various other probabilistic studies. A review of all of the changes points to another case where users are starting to really give development good feedback and we are seeing lots of small but important changes. If you are not familiar with the module, take a look at the help to learn more and see all of the changes, since the following are just the most significant.

Startup Wizard

For the occasional user, all the options in DX can be a little overwhelming. So at 11.0 there is now a wizard that helps users jump into a study by giving guidance as to what study types and options are most appropriate. Once in the environment, the user interface has been updated a little to make it more intuitive and informative.

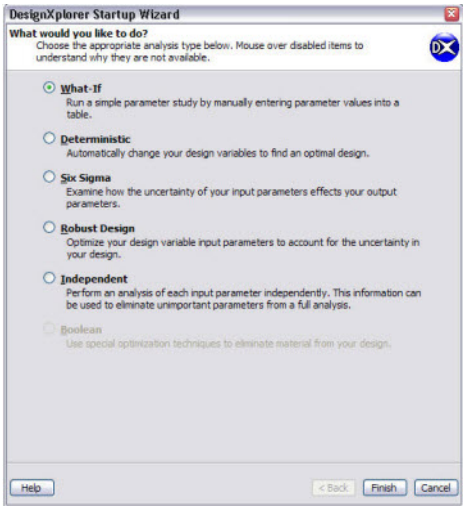


Fig. 8: DX Wizard

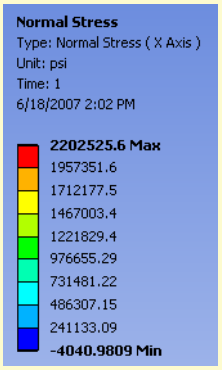
Excel Support

The beginnings of a link to Excel are exposed in DX in the form of an import and export capability from and to Excel. Simply click the Import or Export buttons to use this feature. Although a little basic in this first outing, look for more in this area in the future.

New Sampling Options

User input is best represented in the new and better ways to do statistical studies. These include: 1) Fault tolerant sample generation for optimization, DFSS and Robust design, and 2) Weighted Latin Hypercube sampling for random number generation.

5 Cool Controls for the Result Contour



A lot has changed in the look and control of the legend in Simulation. If you click on it and fiddle around, you'll soon discover most of the changes. Here are the five that we find most useful:

1. Save Layout

Making all the modifications doesn't do you much good unless it is easy to save and recall your settings. In the RMB menu you can choose Named Legends then New, Manage or pick a pre-defined legend. Very, quick, very easy.

2. Click-and-Drag Relocate

If you find your legend needs to move out of the way to better show your results, simply click on the title text at the top and drag it to wherever you want to go. One cool feature of this is that if you drag to the right edge of your plot, everything flips around so it looks correct.

3. On the Fly Add-and-Remove Contours

Ever want to quickly change the number of contours from 9 to 10 or 9 to 4. Click on the contour area and +/- icon shows up. Click on + or - to add or subtract contours.

4. Specify Number of Digits

Another thing that we always seem to want to change is the number of digits used to show result value. The RMB menu under "Digits" lets you specify from 2 to 8 digits. The program will switch from fixed to scientific notation as needed when you change this value.

5. Reset All

The down-side of being able to make all these changes so quickly is that you can screw your menu up really fast. The developers have been thoughtful and put a "Reset All" pick on the bottom of the RMB menu. Choosing it puts everything back to its default.

(Contact, cont...)

The final setting that is most commonly used is the offset behavior. This option is used for many reasons, ranging from fixing a dirty assembly to modeling a press fit. For the sake of time, and Focus length, I'll only cover offset behavior listed within Workbench.

If you have an assembly that should be initially in contact, but isn't, you can use the "adjust to touch" to close whatever gaps exist. If you want to analyze a press fit that is modeled (parts are interfering), you can specify an offset value of 0 and have the solver slowly ramp up the interference. On the other hand, if you model your parts as being initially in-contact, you can specify a manual offset to simulate an actual press fit. This way you can simulate several different press fits without having to create multiple CAD models. If you don't want to ramp the effects during the solution, you can specify that with the last option. No matter which behavior you choose, the Contact surface is offset from the body mesh, and a rigid region is defined between to tie the meshes together.

Hopefully, this has helped clarify some of the settings available within ANSYS. For typical contact problems, I try to stick with Augmented Lagrange with ramped effects turned on. If I'm concerned about contact penetration, I may go as far as using Normal Lagrange. There are still many other settings, such as contact stiffness, stiffness update, and pinball radius. If you're looking at becoming an expert, I would highly recommend the Advanced Contact Course, which spends 2+ days covering all of these topics. (link)

Table 3: Offset Options

Name	Option	Translation
Adjust to Touch	None	Moves contact surface to touch target surface
Add Offset, Ramped Effects	Offset Value	Move contact surface by offset value, ramps offset during solution
Add Offset, No Ramping	Offset Value	Move contact surface by offset value, no ramping

Awesome APDL: Finding Common Block Info

Once again the Focus was almost done and we needed a macro for the Awesome APDL section. Searching on ANSYS.NET, we found a macro that looked really useful, and surprise, someone at PADT had written it a year ago and totally forgot about it. So here it is, resurrected for your coding pleasure.

When you are trying to extract information from ANSYS and there is no good *get command, you sometimes need to delve into the common blocks. Common Blocks are the way you create global variables in FORTRAN. Since ANSYS is FORTRAN, the program stores info that it needs to share with itself in common blocks.

To make things easier for us, and for user written routines, the include files for common blocks are always stored in the ansys/custom/include directory. The first step on using a common block is to search the include files for a variable you are looking for. However, the name is not what you need. The *get,,common commands require the name of the common block and the position in the block

So your next step is to find that offset. You can count in the include file, but if you have arrays and such it gets complicated. So we wrote this macro to figure out the position in the block by: 1) storing the settings in the block you are looking in, 2) changing the value you are looking for, 3) comparing the contents of the block to find any differences, and 4) reporting back what the number is on values that changed.

Want to know more about common blocks, buy PADT's "Guide to ANSYS Customization with APDL" (link)

An example would be if I wanted to know what the global minimum and max number of divisions per line was (DESIZE):

```
desize,10,11
cmndif,0,'cfprp7',0
desize,20,21
cmndif,1,'cfprp7',0
```

Yields:

```
Found: 99 10 20.
Found: 100 11 21.
```

So to get the value use:

```
*get,_mnlnsz,common,,cfprp7,,int,99
*get,_mxlnsz,common,,cfprp7,,int,100
```

Here is the macro: CMNDIF

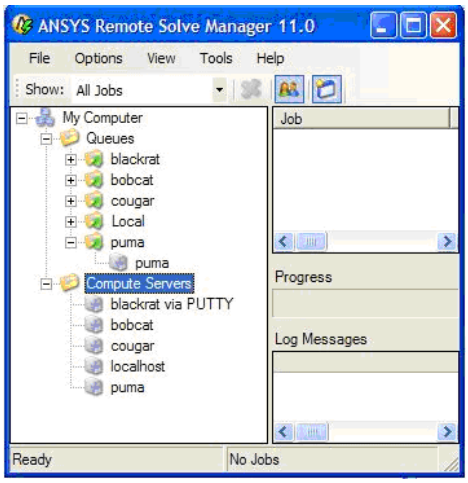
```
/nopr
!cmndif,IPASS, ABLK, IRTYP
! IPASS: Pass number,
!       0 or blank for first pass
!       1 for sedond pass
! ABLK: Name of common block, in
''s
!IRTYP:   1 = REAL
!         0 or Blank = INT
!
imax = 5000
ipass = arg1
ablk = arg2
*if,arg3,eq,1,then
   irtyp = 'REAL'
*else
   irtyp = 'INT'
*endif
*if,ipass,eq,0,then
  v1=
  v2=
  *dim,v1,,imax
  *dim,v2,,imax
  *do,i,1,imax

*GET,v1(i),COMMON,,%ablk%,,%irtyp%
,i
  *enddo
*else
  *do,i,1,imax

*GET,v2(i),COMMON,,%ablk%,,%irtyp%
,i
  *enddo
  *msg
  +++++++ Looking for Changes:
  *do,i,1,imax
    *if,v1(i),ne,v2(i),then
       *msg,,i,v1(i),v2(i)
  Found: %g %g %g
  *endif
  *enddo
  /gopr
*endif
```

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

SSH: Secure Remote Solving



By Jason Krantz

"No way. Never. Are you crazy?"

This is the typical response of a Unix system administrator when told that users want ANSYS installed so they can use Workbench to send jobs to the Unix machine. The request sounds simple, but it alarms admins because it requires the Unix server to run something called Remote Shell, a notoriously insecure protocol.

Remote Shell (RSH) sends usernames and passwords as cleartext — unencrypted text — which means that anyone "listening" to traffic between the Workbench machine and the Unix machine acquires handy username/password pairs — and therefore access to the Unix computer.

Security for FEA is important for a couple of reasons:

a) ANSYS is often used for classified work for government agnecies;

b) Many models, while not actually classified, are new products in development and therefore trade secrets; and

c) I.T. departments with otherwise secure networks are faced with a great big hole created by ANSYS users submitting solutions to remote solvers.

It is now possible to close that security hole. Although Workbench users can still submit jobs over RSH, Secure Shell (SSH) is now a viable option, and an infinitely more secure option at that. SSH uses a variety of encryption schemes, but all of them are much better than RSH. If RSH were Peter Sellers' Inspector Clouseau, SSH would be Daniel Craig's James Bond: same rough idea, much better execution.

So how do we switch over to this magical SSH? Currently, it's not entirely straightforward, but detailed instructions are available here: [how-to link] The configuration process promises to become easier with the next Service Pack for ANSYS.

The key to Workbench 11.0's newfound SSH goodness is its Remote Solve Manager (RSM). The RSM calls a Windows-based SSH client called Putty, which can be downloaded here:

www.chiark.greenend.org.uk/~sgtatham/putty/

The RSM uses Putty to create a secure tunnel between the Windows machine and the Unix machine. SSH is usually used with passwords, but the RSM uses a particularly secure (passwordless) login procedure called hostkey-based authentication.

The inherent security of hostkey-authenticated SSH raises interesting possibilities such as submitting jobs from the internet over SSH (as opposed to submitting them over a local network). As intriguing as this option is, it's not practical because 11.0 pulls entire results files from the remote machine to the Workbench machine. For large models, these files can be multiple gigabytes — so bandwidth is a limiting factor.

For organizations motivated to send jobs to servers in a secure manner, configuring the RSM to use SSH is worthwhile. The process should get easier with progressive releases, and there's no reason that SSH couldn't be incorporated directly into the RSM — the OpenSSH libraries are available to developers for free. Until then, the installation process link above should do the trick.

News - Links - Info

- ANSYS, Inc. Was recently recognized as the biggest R&D spender in the CAE industry. (link)
- ANSYS does a 2-1 split on stock price based on continued industry beating growth. (link)
- Want to learn more about FLUENT products? Visit their really well done web pages at: www.fluent.com/software
- The ANSYS, Inc. Virtual Demo Room is growing every month with useful videos and PowerPoints of products in action. Demos, tutorials, presentations, 100's of files ready for you to download and few. A very good way to get to know other products better: www-harwell.ansys.com/demoroom

Upcoming Training Classes

Month	Start	End	#	Title	Location
June, '07	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
	7/26	7/27	102	Intro. To ANSYS, Part II	Tempe, AZ
Aug '07	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
	8/22	8/24	402	ANSYS High Frequency Emag	Tempe, AZ
	8/30	8/31	701	Design Optimization and Prob.	Tempe, AZ
Sep '07	9/6	9/7	301	Heat Transfer	Tempe, AZ
	9/10	9/12	101	Intro to ANSYS, Part I	Albq, NM
	9/13	9/14	102	Intro to ANSYS, Part II	Albq, NM

The Shameless Advertising Page