

What's New in Workbench Meshing?

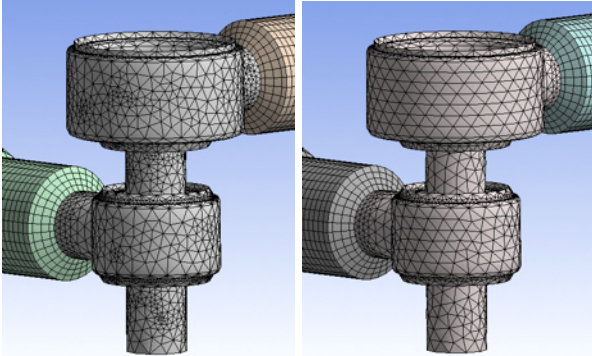


Figure 1: Improvements in Surface Mesh Quality from R11 (Left) to R12 (Right)

By Eric Miller

Most of the users out there have had R12 for some time now and hopefully you have had time to take a look at the changes in meshing. With all the other new paradigm changing features in this release, it is easy to pass over the changes in meshing, changes that we are finding have a very significant impact on the speed and quality of our models.

In our opinion, the meshing capabilities in Workbench are a good example of the new thrust in the world of Workbench to walk a balance between automation/ease-of-use versus power/user-control. This is done in meshing by keeping the pretty good default and high level control tools that have been there for a while and allowing the user to either set advanced global controls or attach detailed controls to specific objects. It is also a good example of how ANSYS, Inc. is merging different technologies and physics to offer a single look and feel, while keeping physics specific settings.

The first thing you will notice is going to be how the whole change in the project page makes meshing something that stands out more and a process step of its own. These changes also illustrate how meshing fits into the whole idea of combining physics and sharing things like meshes between systems.

Once you use the new meshing tools you will very quickly notice the improvement in mesh quality. Most visible is the "clustering" seen in previous versions of the mesher, which is almost gone now. Figure 1 shows a good example of this. Not only does this provide a more uniform result distribution and eliminate spurious gradients, it also (Cont. on pg. 2)

Implementing Fluid Pressure Penetration



GoogleMaps view of the Neighborhood Just West of PADT. There are a Lot of Pools in Arizona!

By Ted Harris

Backyard swimming pools are very popular here in the Valley of the Sun. There's nothing like hopping into a 92 degree pool to cool you off on a 117 degree day. Really. It makes outside bearable during our 4 months of, "Yeah but it's a dry heat," to which I always add, "Like inside your oven." With a pool comes maintenance and repair, though, which bring us closer to the point of this article.

A couple of years ago, I noticed a slow leak at the junction between my pool filter and the downstream piping. Being one of those engineers who loathes paying an expert to do something I feel I can do myself, I endeavored to fix it. A couple of pool supply stores later I returned home with a

\$0.99 O ring. Two things about O rings: they don't work if they are too loose and they don't work if they are too tight. Unfortunately I missed the sweet spot and the fitting leaked when I got it all back together. In fact, I had tightened the threads on the fitting so much that the fitting itself was now cracked. I had also learned at the pool store that my installation was non-standard and they couldn't match the fitting to my particular filter. Time to call in an expert pool guy. Over \$200.00 later I had a watertight connection again. In retrospect maybe the part was cracked to begin with, but if the O ring was indeed the problem in this case I gambled on a self repair and lost.

(Cont. on pg. 4)

In this Issue...

- 1.....What's New in Workbench Meshing?
- 1.....Implementing Fluid Pressure Penetration
- 5.....Announcing PADT's first Web Based Class: Workbench Mechanical
- 6.....PeDAL: The APDL Editor
- 7.....Faster Fourier! Kill! Kill! (Nehelam Benchmarking Update)
- 8.....The Top 10 Most Important New Features in Workbench 12.0

Details of "Mesh"	
[-] Defaults	
Physics Preference	Mechanical
Relevance	0
[-] Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	3.1416e-002 in
+ Inflation	
+ Advanced	
+ Pinch	
+ Statistics	

Figure 2: Mesh Details

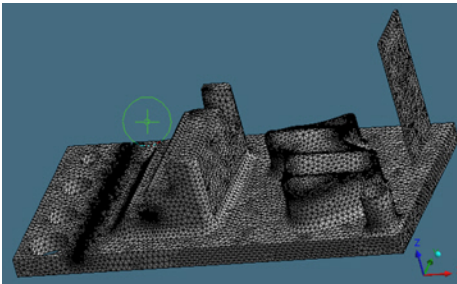


Figure 3: Curvature Advanced Size Function

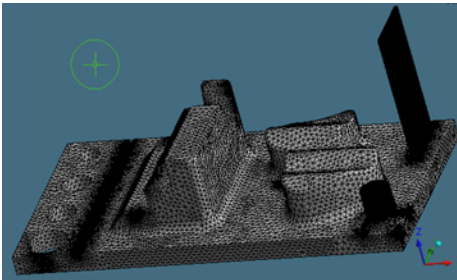


Figure 4: Proximity Advanced Size Function

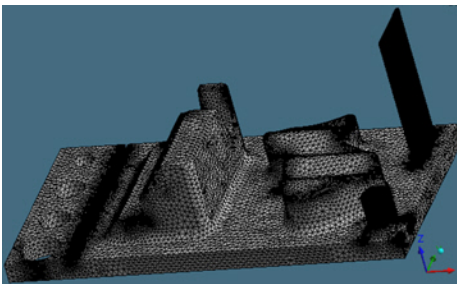


Figure 5: Curvature & Proximity

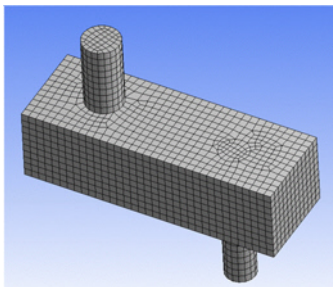


Figure 6: Multizone Mesh

(Meshing, Cont...)

generally results in fewer elements. This is the most obvious example of changes made throughout the internal algorithms that generate meshes that provide more uniform and “quality” meshes.

Most of the global controls are defined in the Details dialog for the Mesh branch of the tree, shown in Figure 2. The first option is the most important, Physics Preference. This determines what other options are available and sets the defaults for most commands for the type of problem you are trying to mesh. If you are doing CFD, you can also specify if you are using CFX or FLUENT and the program sets defaults appropriately. Most of Sizing options are the same as in earlier versions, with the exception of “Use Advanced Size Function.” Turning this to “On” exposes all the cool bells and whistles that make 12.0 a great release for meshing.

When you turn it on, you can tell the program to control the mesh based on proximity of features, the curvature of features or a combination of the two. You can also tell it to not use either, but allow you to set the advanced size options. Figures 3, 4 and 5 show the results of each option and both together. As you can see it goes in and looks at the size of features and automatically introduces refinement as needed. We have been very happy with the curvature option for mechanical meshes.

This release also introduces a few new meshing methods. The most intriguing is Multizone. This is the first step towards automatic HEX meshing. It basically goes in and attempts to slice up your geometry to make six-sided volumes that can be hex meshed. On some geometries it works pretty good, on others, not so good. We recommend you play with it and get a feel for how it works on your geometry. Figure 6 shows a nice example.

The other method to be aware of is Non-Conforming, what we prefer to call patch-independent. This basically runs the TGRID mesher in batch mode which ignores the edges of surfaces and fills the volume rather than meshing surfaces and growing a mesh inward as the patch dependent method does. Figure 7 is our silly test geometry meshed with the two methods. We have found that large, complex geometries work very well with the patch independent method. We expect to see this get better with each release, especially for CFD applications.

If you work with shell or 2D geometry, you will also like the new inflation controls for shells. Figure 8 shows an example of how it puts nice, normal elements on edges. You can of course specify which edges and what ratios for the normal layers. This is a much needed improvement.

There are many other modifications deserve their own articles, but we want to make sure everyone is aware of them:

Inflation: CFD users live and die by their boundary layer meshes and workbench now support better manual specification of inflation layers as well as some pretty

(Cont. on pg. 3)

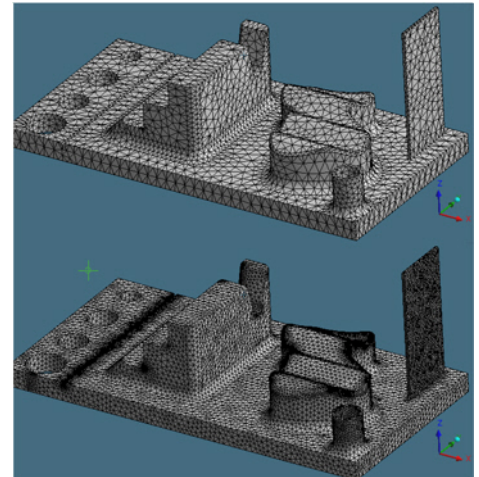


Figure 7: Conforming and Non-Conforming Meshes

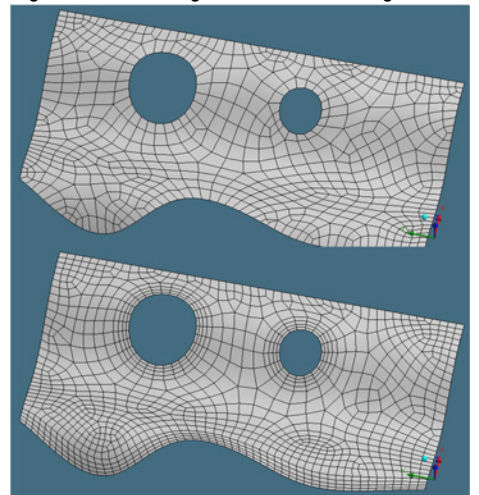


Figure 8: Inflation on Shells

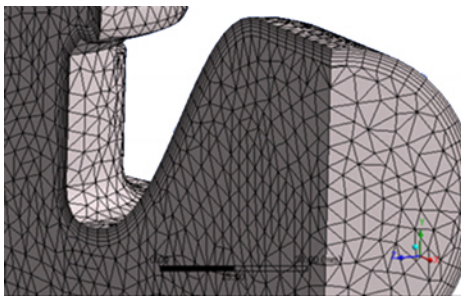


Figure 9: Inflation

(Meshing, Cont...) effective automatic methods. Figure 9 is a good example

Pinch: Sometimes you just want to ignore a small feature, and with pinch you can specify a feature size or select an entity and the mesher will mesh right over it as if it was not there. This is handy for getting rid of ledges, small fillets, and slivers. The model in Figure 10 shows a not-so-realistic application.

Advanced Map Meshing: This one really deserves its own article. It is basically a more sophisticated version of the map mesh by corner command in ANSYS MAPDL. But you can specify more information and use the map mesh as a seed for a swept mesh, making it possible to mesh objects that are not obviously six sided. Figure 11 is a simple example.

Body of Influence: Sometimes you want to specify a change in refinement that is not associated with a piece of geometry or a whole part. With Body of Influence you can create a “dummy” solid that sits coincident with your real geometry. You then define refinement for that dummy body and it applies the refinement to the intersection of the dummy geometry and your real geometry, as is shown in Figure 12.

As always, the only way to really get your head around this stuff is to get in there and play with it. Try all the different options on your geometry and see how it does. As Workbench 2 progresses and more meshing technology is merged under workbench, the number of tools and their effectiveness will improve at each release.

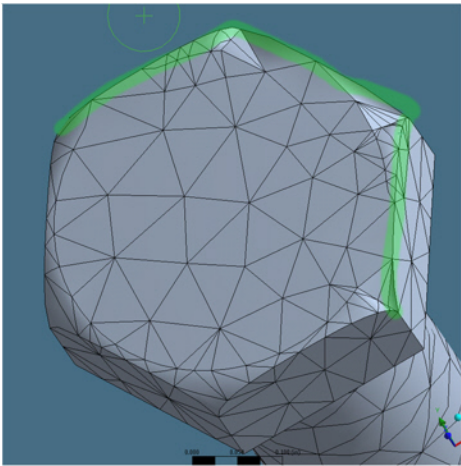


Figure 10: Pinch

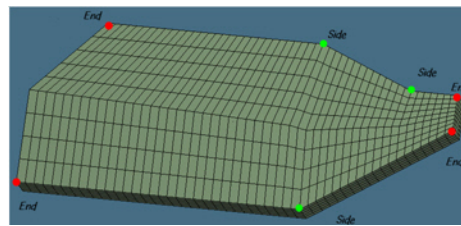


Figure 11: Advanced Map Meshing

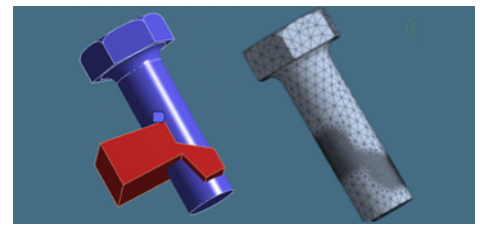


Figure 12: Body of Influence

PADT's Training Schedule

Month	Start	End	#	Title	Location
Nov '09	11/6	11/6	702	ANSYS DesignXplorer	Tempe, AZ
	11/9	11/10	102	Introduction to ANSYS (Mechanical APDL), Part II	Tempe, AZ
	11/12	11/13	204	ANSYS Mechanical APDL Advanced Contact and Fasteners	Tempe, AZ
	11/19	11/19	111	ANSYS Workbench DesignModeler for CFX	Tempe, AZ
Dec '09	12/3	12/4	103	Introduction to ANSYS Workbench Mechanical	Tempe, AZ
	12/7	12/8	302	ANSYS Workbench Simulation 11.0 Heat Transfer	Tempe, AZ
	12/8	12/17	113	Introduction to ANSYS Workbench Mechanical (Web Class)	Web Based
	12/9	12/10	207	ANSYS Workbench Mechanical – Structural Nonlinearities	Tempe, AZ
	12/14	12/15	801	ANSYS Customization with APDL	Tempe, AZ
Jan '10	1/8	1/8	107	ANSYS Workbench DesignModeler	Tempe, AZ
	1/11	1/13	101	Introduction to ANSYS (Mechanical APDL), Part I	Tempe, AZ
	1/14	1/15	100	Engineering with Finite Element Analysis	Tempe, AZ
	1/21	1/22	121	ANSYS Mechanical 12.0 Advanced (Using Command Objects)	Tempe, AZ
	1/25	1/26	103	Introduction to ANSYS Workbench Mechanical	Las Vegas, NV
Feb '10	1/27	1/28	207	ANSYS Workbench Mechanical – Structural Nonlinearities	Las Vegas, NV
	2/1	2/2	301	ANSYS Mechanical APDL Heat Transfer	Tempe, AZ
	2/3	2/4	801	ANSYS Customization with APDL	Tempe, AZ
	2/5	2/5	411	ANSYS Workbench Simulation 11.0 Electromagnetics	Tempe, AZ
	2/11	2/12	652	CFX Multiphase Flows	Tempe, AZ
	2/16	2/25	113	Introduction to ANSYS Workbench Mechanical (Web Class)	Web Based
	2/19	2/19	206	ANSYS Workbench Mechanical Rigid & Flexible Dynamics	Tempe, AZ
Mar '10	2/22	2/23	202	ANSYS Mechanical APDL Basic Structural Nonlinearities	Tempe, AZ
	9/28	9/29	104	ANSYS WB Simulation – Intro	Las Veg., NV
	3/1	3/2	103	Introduction to ANSYS Workbench Mechanical	Tempe, AZ
	3/3	3/4	207	ANSYS Workbench Mechanical – Structural Nonlinearities	Tempe, AZ
	3/8	3/9	203	ANSYS Mechanical APDL Dynamics	Tempe, AZ

(Pressure Penetration, Cont...)

After hearing about the new fluid pressure penetration capability in ANSYS 12.0, I decided to investigate the O ring problem again, but this time as a simulation using ANSYS Workbench Mechanical. The idea of fluid pressure penetration is that we simulate the effect of fluid pressure in a contact region that is opening up by allowing an applied pressure to act on the portion of the contact region that has separated. The fluid pressure penetration capability is not directly supported in the ANSYS Workbench menus so some ANSYS Mechanical APDL commands need to be added to the analysis type branch.

Loading Conditions

I decided to use the ANSYS Bolt Load capability to tighten all 6 bolts and then selectively loosen some of them to allow the joint to separate a bit. Bolts were numbered #1 through #6 circumferentially around the joint. All six bolts were preloaded with a compressive force in the first load step. At load step 2, bolts #1 through #3 had the load reduced to zero, #4 had the load reduced to a small compressive force, and bolts #5 and #6 were locked at the applied preload. In the third and final load step, bolts #1 through #4 were opened, meaning all preload was removed, while bolts #5 and #6 remained locked.

This bolt loading/unloading scheme was followed to allow the joint to open up a bit due to the applied internal pressure, which was zero at load step 1, ramped up to the full value at load step 2, and held at full value for load step 3. The goal was to have the fluid pressure penetrate the O ring joint due to the selective bolt loosening around the joint.

Contact Conditions

Frictional contact was used to model the O ring axial surfaces to both joint parts. Bonded contact was used to connect the bolts to the blind holes in the opposing joint section as well as to connect the bolt heads to the through-hole side of the joint. Frictionless contact was used to connect the O ring inner diameter radial surface to the adjacent radial joint surface, as well as the radial surfaces between the two connector pieces that touched.

Mesh

The new at 12.0 Multizone meshing method was used with the Hexa Dominant free mesh type option to mesh the two connector bodies as well as the bolts. The O ring was sweep meshed. All elements were quadratic. The mesh was purposely coarse to benefit solution time since this was a demonstration exercise for this article.

Commands Defining Fluid Pressure Penetration

The following commands were inserted under the Static Structural branch. The parameter 'cont1' was defined using another command snippet under the contact branch which defined contact between the gasket and the through-hole connector axial face. This was set to cid in that command snippet so that that set of contact elements could easily be selected via parameter cont1.

A pressure of 60 units was applied to the contact elements on the gasket to through-hole connector axial face. Note the pressure is applied directly to the contact elements, not the underlying solid elements. For details on utilizing fluid pressure penetration, see section 3.11 - Applying Fluid Pressure-Penetration Loads in the Contact Technology Guide in the ANSYS Mechanical APDL Help. The remaining commands define the fluid pressure 'leak' starting location using a cylindrical coordinate system and radial coordinate value.

Note that Appendix A of the Contact Technology Guide mentioned above has an example input file which uses fluid pressure penetration:

(Cont. on pg. 5)

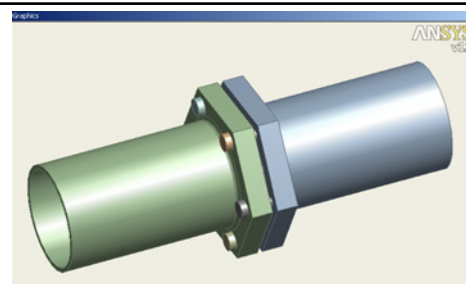


Figure 1: Geometry Created in Design Modeler

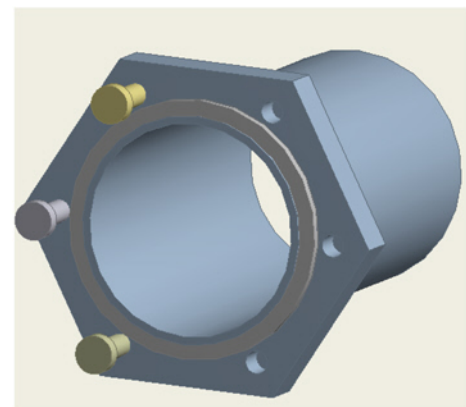


Figure 2: Certain Parts Hidden to Show Square Cut O-Ring and Bolt holes

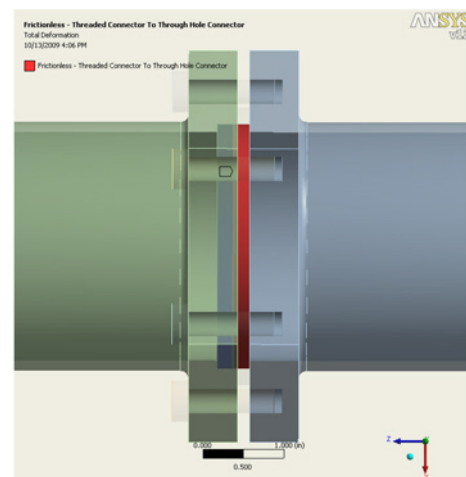


Figure 3: Contact Between Radial Surfaces

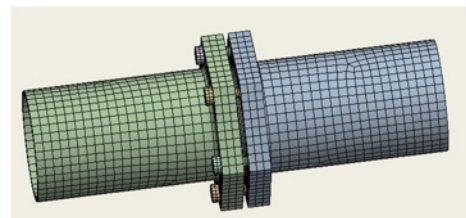


Figure 4: Mesh

(Pressure Penetration, Cont...)

```

esel,r,type,,cont1
sfe,all,1,pres,,60
sfe,all,2,pres,,-1
nsle
csys,13 !cylindrical system
nsl,r,loc,1.7
esln,
esel,r,type,,cont1
sfe,all,2,pres,,1 ! pressure leak start location
csys
allsel

```

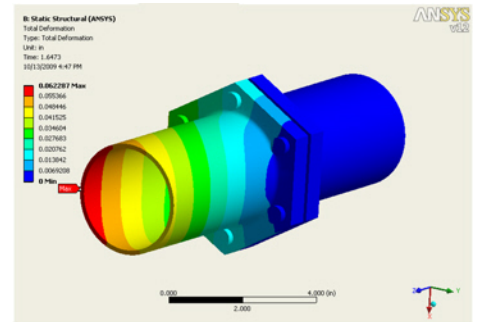


Figure 5: Total Deformation, Joint Separating

Solution

Since this was a nonlinear analysis with large deflection effects and nonlinear contact, multiple substeps were used per load step. Note that in the final load step, the model converges for only a few substeps once the fluid has broken through. This is due to the fact that the O ring distorts quickly once contact is lost between the ring and the mating face of the connector. The goal of this simulation was to model the fluid penetrating the contact region, not the post failure deflections of the O ring.

Results

Deflection and contact status results were plotted in ANSYS Workbench. The fluid pressure penetration plots were created in Mechanical APDL, although they could have been created in Workbench Mechanical by inserting the appropriate commands.

Conclusions

The new fluid pressure penetration capability can be used to simulation fluid leaks in joints that separate. That separation requires that contact elements be used at the joint. As with most nonlinear analyses, care must be taken to avoid convergence difficulties. Further, although the technique is not yet directly supported in Workbench, the appropriate ANSYS commands can be added to enable the capability.

Now, about that hot water heater...

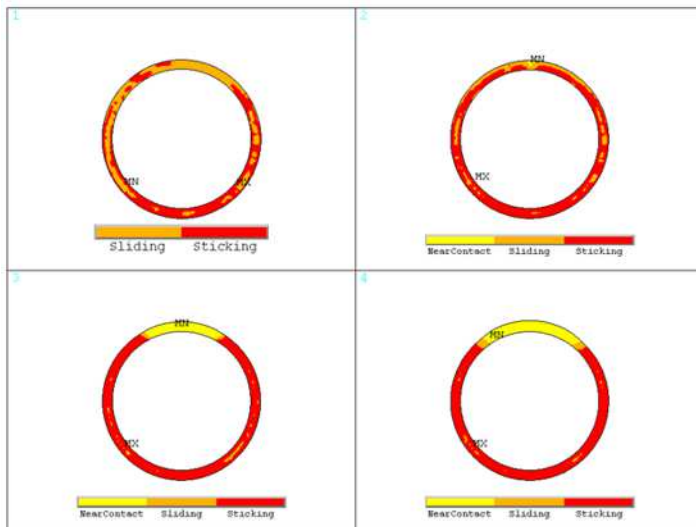


Figure 6: Contact Status for Progressive Pressure Penetration and Joint Separation

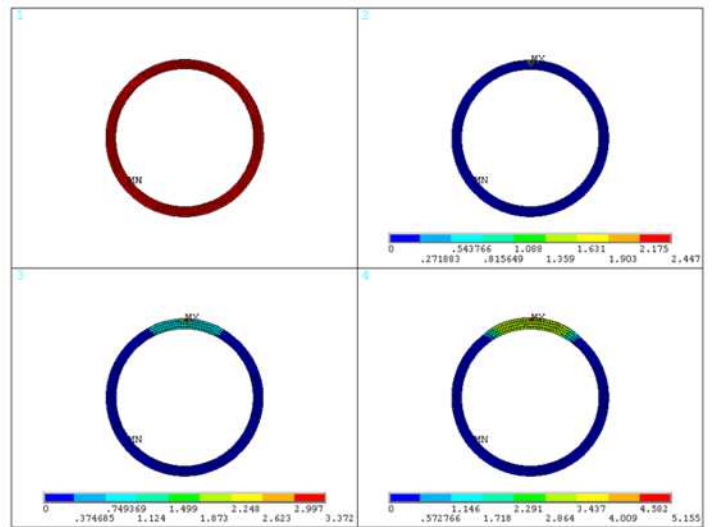


Figure 7: Contours of Fluid Pressure Penetration for Progressive Joint Separation

PADT Adds a new Workbench Mechanical Class and offers it On-Line

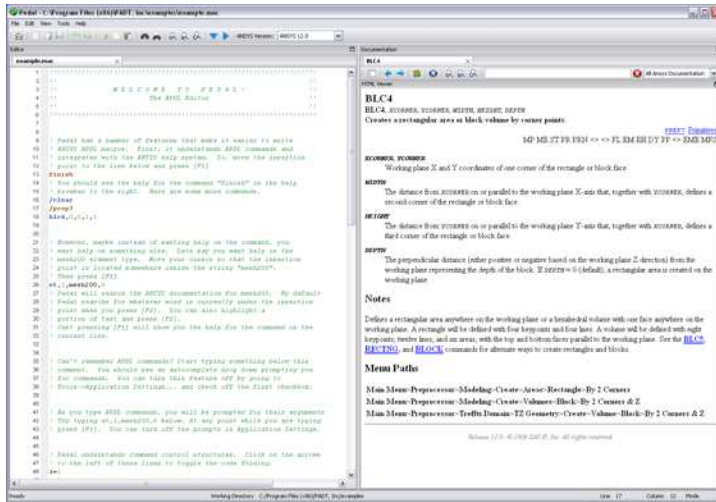
For some time now customers have asked for versions of our most popular classes to be given over the web. We are pleased to announce Course 113 - Introduction to ANSYS Workbench Mechanical (Web Class). The class is presented in six 2.5 hour sessions over a two week period and was developed from scratch by PADT's technical staff based upon 15 years of experience teaching ANSYS. Students will view the class over the Internet and listen on a conference call. The class does include workshops and students will be able to share their work over the web with the instructor or other students. This "virtual classroom" has been tested with a few test classes already and our team is ready to welcome people from around the world to any of the 5 scheduled dates or, if needed we can set up a date that works for you. Learn more at: www.padtinc.com/support/training/course.asp?c=113



Introducing PeDAL: The APDL Editor

By Matt Sutton

After years of writing ANSYS macros I began to feel as though my brain had simply reached full capacity when it came to remembering APDL commands. Full capacity turned out to be about 50 commands that I could recite in my sleep. The other 1000 or so commands existed in my mind along a continuum ranging between "who knew that existed" and "something, , something, something; what's that fourth something again?" Macro writing consisted of virtually equal parts text editing and ANSYS help searching. And, when it came to searching the help, I seemed to spend 80% of my time looking up help on various commands. So, when version 12 of ANSYS was released and I saw their new help system, I realized I had an opportunity to really enhance my macro writing experience by building an editor in which the ANSYS help would be displayed in a pane directly beside my text editing window. If I could link up the documentation on a given command to what I was typing then

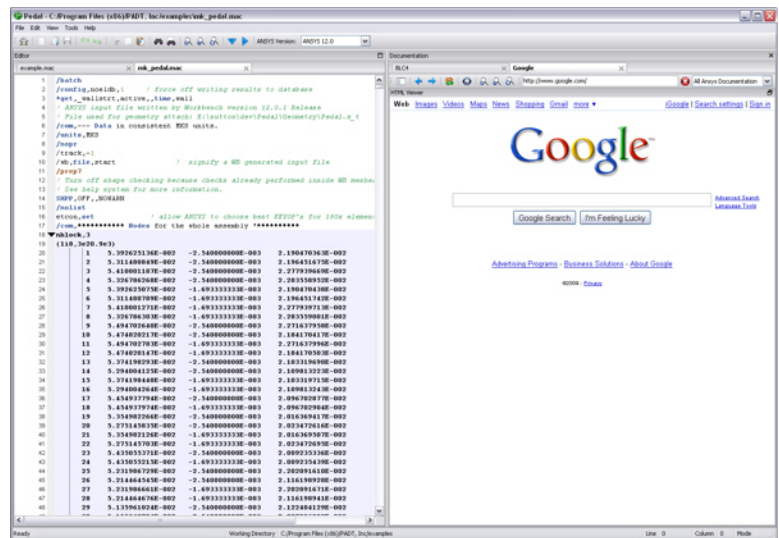


I could eliminate the intermediate step of searching the help and my limited cranial capacity would no longer be a detriment. Thus, PeDAL was born. I initially wrote it for myself and it continues to somewhat selfishly serve my needs; however, other folks at PADT and elsewhere begin to use it and over time it has evolved into a more full featured editor. PeDAL includes all of the things you would expect in an editor, like syntax highlighting, multiple file editing, etc... but also includes other small features that make writing ANSYS macros easier and more enjoyable. Furthermore, there are a litany of features that I will be adding in the coming months, but even now it is my text editor of choice when it comes to writing/editing ANSYS macros. You can try it out for free for 30 days and if after that time you wish to keep using it, you can purchase a copy for \$49. A purchased license will enable you to receive free upgrades and bug fixes between major versions of the software. PeDAL does require version 12.0 to be installed on your machine so that it can integrate with the ANSYS help system.

Joseph Metrisin from Florida Turbine Technologies, Inc was a beta tester and his comments on PeDAL were:

"PeDAL is going to be a "must have" tool for any ANSYS macro writer. The auto-complete, syntax highlighting, and integration of ANSYS help system are hard to do without once you've tried them."

I encourage you to download it from www.padtinc.com/pedal and try it out.



News - Links - Info

- ANSYS once again makes more Small-Cap best-of lists, this time the Forbes 200 Best Small Companies [<link>](#) and Deloitte's Technology Fast 500 [<link>](#)
- ANSYS is used to simulate the Wimbledon Retractable Roof [<link>](#)
- Newest version of Ansoft's HFSS is released with tons of enhancements [<link>](#)
- A couple of great issues of the ANSYS magazine have been published since the last The Focus [<link>](#)



What is it about PADT engineers and bad B movies? Visit [wikipedia](#) to learn about this obscure title reference: *Faster, Pussycat! Kill! Kill!*

Faster, Fourier! Kill! Kill!

By Jason Krantz

Now is a good time to buy a new computer. Why? Intel's latest set of Xeon chips has proven to be much, much faster than the last ones. We've run a long series of benchmarks on the new chips, and the results are tantalizing.

In June of 2008, *The Focus* ran an overview of the upcoming Nehalem processors. A few preliminary benchmarks run by [anandtech.com](#) predicted improvements of around 28 percent, but it was unclear whether ANSYS users would see such a large improvement.

We do. PADT was lucky enough to pick up a dual-socket, eight-core Nehalem workstation (an HP z800) soon after they became available. The record solve time for our internal large-model benchmark was previously 2.40 hours, set by another HP box—this one with two quad-core Xeon X5482 chips. The new machine, with two 2.8 GHz quad-core Xeon X5560 CPUs, finished the run in just 1.78 hours.

You'll notice that the previous record holder ran at a higher frequency: 500 MHz faster than the z800. This makes the z800 machine look especially impressive, but it also makes us wonder: what's the clock-for-clock improvement? That is, how much faster is the new machine if we control for clock speed?

If we assume the results scale linearly with clock speed, we can adjust the 3.2 GHz Harpertown time from 2.40 hours at 3.2 GHz to 2.74 hours at 2.8 GHz. On a clock-for-clock basis, the Nehalem machine is 35% faster than the Harpertown machine.

For a technical comparison of why the 5500 series is so much faster than the 5400 series, please see the June 2008 *Focus* article.

Another advantage of the newer chips is their power efficiency. We have several older, dual-core AMD Opteron boxes in our cluster. We used a consumer power consumption gauge (a Kill-a-Watt) to see how much power we'd save by replacing four of these boxes (8 cores total) with a single 8-core z800 machine.



Each of the four 2-core machines gulped down 400 watts while solving an FEA model; that's 1600 watt-hours per hour of solving time. In contrast, the z800 consumed about 435 watts while solving. So the newer computer consumes about 1.2 kilowatt-hours *less* power per hour of solving time than the older machines. If you're solving constantly, you save about 850 KWh per month when you replace older computers.

What's more, the z800 solves a given model faster than the collection of four older machines, so it's operating at peak power consumption for a shorter time, saving still more power. Finally, every watt dissipated as heat in a server room must then be air-conditioned out of the room; saving watts on individual machines then saves additional watts in air conditioning.

Test notes: All benchmarks run on CentOS 5.3 Linux with ANSYS 12.0 (full release, not beta)

How to Make a Fast Computer Even Faster

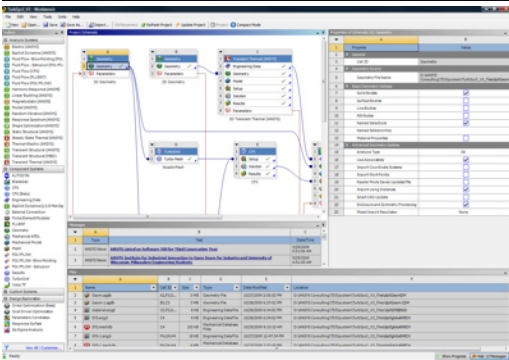
Though the z800 machine referred to here is very, very fast, it doesn't hold the PADT record. What does? The same machine with a recompiled kernel. Recompiling the Linux kernel allows you to optimize the core of the operating system for the hardware you have. This speeds things up because the pre-compiled kernel that comes with your Linux distribution is set up to run adequately on many different kinds of hardware rather than to run quickly on one specific kind of hardware.

Recompiling a kernel requires that you be very comfortable with Linux. You need to be quite sure of what you're doing, but if you are, there are substantial benefits to be had.

We used a recent version of the Gnu C Compiler (gcc 4.3), a recent kernel version (2.6.30) and generally chose to optimize for execution speed rather than file size. The results were promising: solve times fell from 1.78 hours to 1.63 hours—an 8.4-percent improvement. This is analogous to bumping your CPU from 2.8 GHz to 3.0 GHz.

This small-but-significant improvement would be worth doing if, for example, you had a homogenous cluster of several identical machines. You'd only have to compile the kernel once; it would run on all the machines, since the hardware is the same. Each machine would benefit, solution times would drop and the time involved is reasonable. If you're compiling a separate kernel for each of seven different machines, it's probably not a great idea.

ANSYS supplies its own pre-compiled FORTRAN libraries (e.g., libgfortran), and it might be possible to speed things up even more by re-compiling those for your specific hardware. However, one would quickly reach a point of diminishing returns; recompiling these things is labor-intensive, and the speed gains would likely be small.



The Top 10 Most Important New Features in Workbench 12.0

By Eric Miller

Over the years the tools that we collectively call ANSYS Workbench have changed considerably, and at release 12 we are seeing some of the biggest changes to date. In this article we will cover the ten new features that we feel are the most important. Some

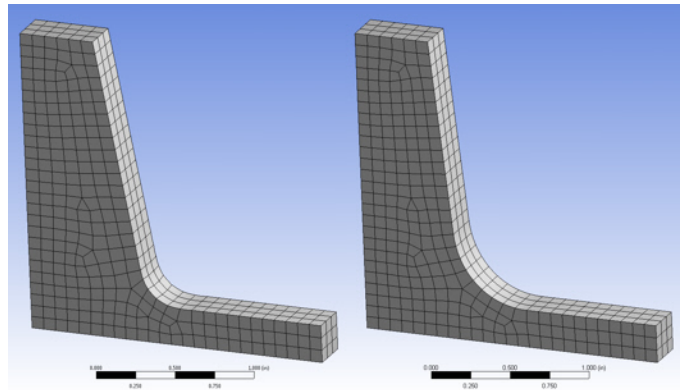
of them will have a big effect on user productivity while others simply show a commitment by ANSYS, inc. development to meeting the needs of their users.

So, in order from least to most significant, here are the feature we think every user should be aware of and understand:

10: Mesh projection improvements in FE Modeler

If you have been a long time reader of The Focus you should know that PADT has been a big fan of the mesh morphing technology that ANSYS, Inc. acquired when they purchased CADOE. Over many releases that technology has been moving deeper and deeper into the Workbench to where it is now firmly planted in FE modeler and working well.

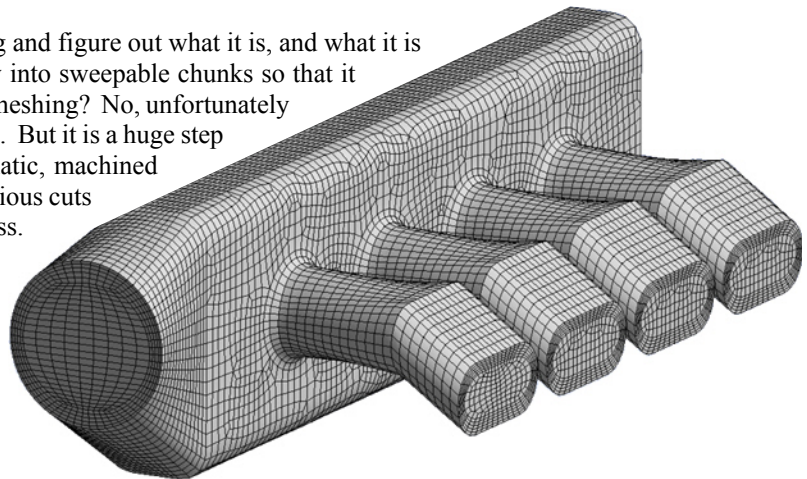
The basic concept is fairly simple: 1) read in a mesh, 2) find the outside element faces of that mesh and break those up into surfaces (faceted) with edges, 3) import some CAD geometry, 4) project the nodes on various faces onto the new geometry, and 5) smooth the mesh. Easy to describe, but hard to program. At R12 we have found that the tools are working well and will allow you to fairly quickly morph an existing mesh onto new geometry, solving the age-old problem of having to compare results from two different geometries without changing the mesh too much. It also has the long-standing capability to translate, rotate, or offset faces or edges. And because it is part of FE Modeler, you can do this with NASTRAN, ABAQUS or ANSYS meshes. We hope to add a detailed review of this feature in a future issue of The Focus, but if you want to try it out, use the online help to guide you through the process. There are some new concepts that you need to digest before you get it down.



9: Multizone Meshing

It took us a while to get our head around Multizone meshing and figure out what it is, and what it is not. What it does is automatically break up your geometry into sweepable chunks so that it can hex mesh things. Is this the holy grail of automatic hex meshing? No, unfortunately it only works on a subset of geometry that it can block easily. But it is a huge step in the right direction and for a ton of uses that have prismatic, machined parts it may work more often than not. And with a few judicious cuts in DesignModeler, you can increase the likelihood of success.

One of the things we really like about it is that the developers did not take an all-or-nothing approach. If the default zoning fails, the user can go in and specify some source and target surfaces and you can also tell it to Tet mesh zones it cannot fit hexes into. Pretty cool stuff and we hope to see it grow and apply to more geometries in future releases.



8: Stronger Beam & Shell in ANSYS Mechanical

This item covers a whole collection of small features that add up to some nice improvements for users of beams and shells. The most significant change is support of offsets of sections for beams and shells. This allows you to specify a line where it makes sense from a modeling standpoint and offset the beam. For beams, you can now also now define your own cross section by sketching it, something that a lot of users were asking for.

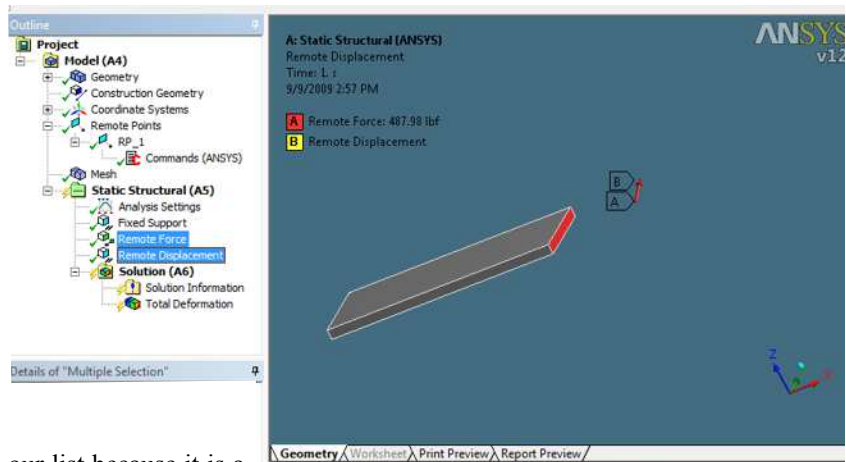
Another huge improvement just for beams is improvements to how meshes are created for beams. Now, if you have different orientations for beams, you get different element types. In addition, the program creates orientation nodes for you. This may overcome some of the more common objections that users have for creating beams in Workbench.

(Cont. on pg. 9)

(WB Top 10, Cont...) Post processing is now easier for beams with the new Beam Probe. This will give you detailed forces and moments on the beam, especially useful if you are using a beam to connect geometry. The last significant change is the ability to fix rotation on beam and shell nodes based upon a user assigned coordinate system.

7: Remote Point

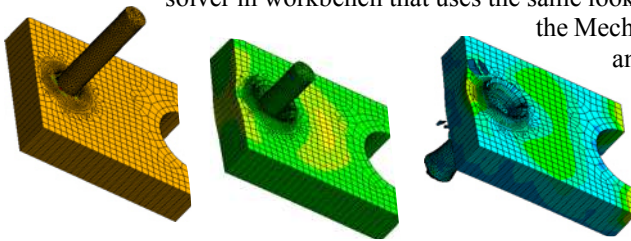
If you do a model in Mechanical APDL after working in Workbench for a while, one of the features you miss is the ability to define a remote load. This is a load applied to a point in space that is “tied” to a surface, edge or point on your geometry. As you can imagine, the APDL to do this can be pretty complicated. At R12 this capability has been expanded to allow you to create remote points in space and then apply multiple forces or constraints to that point. More importantly, you can attach a command snippet to the point and use the node in an APDL macro. This gives the user considerable power and flexibility to create a node in space and then add elements, loads, constraints or constraint equations beyond what Workbench currently supports.



Although this is not a very glamorous feature, it made our list because it is a very quick and simple way to include complex loads in your model without scripting or “fake” geometry. We recommend that the first couple of times you use it, you bring your model up in Mechanical APDL to study the way that the load or constraint is actually applied.

6: ANSYS Explicit STR

This is more than a new features, it is an entirely new solution option within Workbench. Explicit STR is a structural explicit dynamics solver in workbench that uses the same look and feel as ANSYS Mechanical, but uses the AUTODYN solver instead of the Mechanical APDL solver. It is great for solving impact, drop shock, containment, and large deformation problems that you would normally need LS-DYNA or AUTODYN to solve. But it has the advantage of workbench, including bi-directional Associativity to CAD. It doesn't support the FSI and explosions that full AUTODYN does, but for most explicit dynamics users, it does what they need. So far we are very impressed with how simple and powerful this interface is, taking this area of simulation from the realm of wizards to the preview of most ANSYS users.

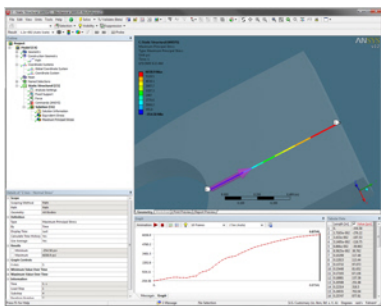


5: User Defined Results/Full RST File Contents Available

A big complaint that Workbench users have had for many releases is the fact that much of the results that the ANSYS solver calculates were not available in Workbench. At R12 users now have access to everything in the RST file through something called User Defined Results. This is a much simpler solution than the Etables and SMISC/NMISC long time APDL users are used to. The way it works is that you click on the Solution branch on your model tree and then click on the Worksheet Tab. This brings up a table of all the valid results in your result file. To plot those results, simply right click on them and choose “Create User Defined Result.” You can then go back to your geometry tab and you'll see the new result added to the tree.

Type	Data Type	Data Style	Component	Expression	Output Unit
U	Model	Scalar	X	UX	Displacement
U	Model	Scalar	Y	UY	Displacement
U	Model	Scalar	Z	UZ	Displacement
U	Model	Scalar	SUM	USUM	Displacement
U	Model	Vector	VECTORS	UNVECTORS	Displacement
S	Element Nodal	Scalar	X	SX	Stress
S	Element Nodal	Scalar	Y	SY	Stress
S	Element Nodal	Scalar	Z	SZ	Stress
S	Element Nodal	Scalar	XY	SXY	Stress
S	Element Nodal	Scalar	YZ	SYZ	Stress
S	Element Nodal	Scalar	XZ	SXZ	Stress
S	Element Nodal	Scalar	1	S1	Stress
S	Element Nodal	Scalar	2	S2	Stress
S	Element Nodal	Scalar	3	S3	Stress
S	Element Nodal	Scalar	INT	SINT	Stress
S	Element Nodal	Scalar	EQV	SEQV	Stress
S	Element Nodal	Tensor	VECTORS	SVECTORS	Stress
S	Element Nodal	Scalar	MAXSHEAR	MAXSHEAR	Stress
EP10	Element Nodal	Scalar	X	EP10X	Strain
EP10	Element Nodal	Scalar	Y	EP10Y	Strain
EP10	Element Nodal	Scalar	Z	EP10Z	Strain
EP10	Element Nodal	Scalar	XY	EP10XY	Strain
EP10	Element Nodal	Scalar	YZ	EP10YZ	Strain
EP10	Element Nodal	Scalar	XZ	EP10XZ	Strain
EP10	Element Nodal	Scalar	1	EP101	Strain
EP10	Element Nodal	Scalar	2	EP102	Strain
EP10	Element Nodal	Scalar	3	EP103	Strain
EP10	Element Nodal	Scalar	INT	EP10INT	Strain
EP10	Element Nodal	Scalar	EQV	EP10EQV	Strain
EP10	Element Nodal	Tensor	VECTORS	EP10VECTORS	Strain
EP1L	Element Nodal	Scalar	X	EP1LX	Strain
EP1L	Element Nodal	Scalar	Y	EP1LY	Strain
EP1L	Element Nodal	Scalar	Z	EP1LZ	Strain

An added feature is that you can also now do math on results values. If you look at the table of results, there is a column labeled “Expression.” You can insert an empty User Defined Result and use the values in the Expression column as variables in equations.



4: Paths in Post Processing

Path plots are another one of those obscure ANSYS Mechanical APDL features that you did not know how much you used until you did not have it in Workbench. The developers at ANSYS listened and added path plots at R12. The implementation is different in that it takes two steps. In the first you define your path and in the second, you insert a result object and scope that object to your path.

You can create the path three different ways: 1) pick or specify two points in space, 2) use the x-Axis of a coordinate system, and 3) use an edge on your geometry. Once you have calculated the

(Cont. on pg. 10)

(WB Top 10, Cont...) results on the path you get a graph and a table, and the table can be exported to a text file.

3: Repair Tools in DesignModeler

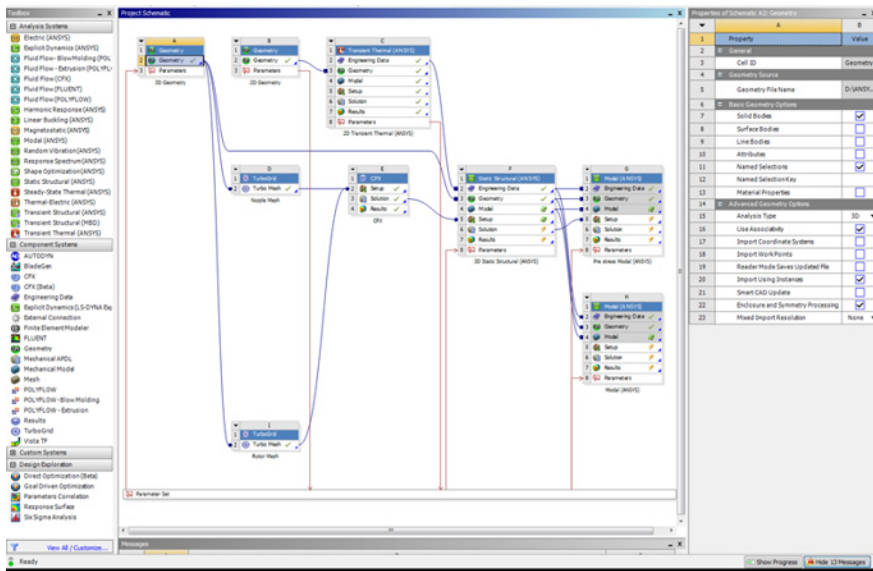
One of the biggest time savers in R12 is looking to be the set of repair tools found in DesignModeler. DM has always had some tools that could be used to simplify and sometimes repair bad geometry, but never the full suite of tools that users wanted. To begin with there are two “automatic” tools, the first being connect which closes small gaps and lines up vertices and edges, and the second which merges surfaces or edges that touch into a new single entity. The first is great at repairing sloppy geometry and the second is useful in getting rid of small slivers or those annoying edges made up of tons of segments.

Beyond these global tools, there is a complete set of semi-automatic tools that allow the user to identify entities that require repair based on the type of problem. They include repair for: edges, seams, holes, sharp angles, slivers, spikes and faces. Within each repair type, there are usually several methods that can also be chosen to fit the needs for your geometry. The help in this area is very good and we recommend that users read up on it so that they know what these powerful tools can do and when to use them.

2: WB2 Infrastructure

Much has been said and written about the Workbench 2 infrastructure. The sad part is that the huge amount of work done to create it is mostly not visible to the user. We can see the new project page or the interface on EngineeringData, but we can't see how it quickly allows CFX to calculate pressures for an ANSYS Mechanical run or how it extracts and manages parameter from 4 different pre- and post-processors. And if you use your ANSYS, Inc. product the way you always have, you will see no advantage to WB2.

But if you find that you can do your simulation from within the WB2 environment, you will probably see some significant productivity improvements. This is especially true if you build 3D parametric solid models and often repeat the same type of analysis over and over or if you are transferring loads between models. Although we grumbled a bit when we started, PADT's engineers are finding value in this infrastructure every day.



And this is just the beginning. WB2 will start to deliver scripting and journaling at 12.1 and grow with each release. We should also start to see more native applications that are 100% WB2 that deliver even larger productivity gains.

1: Introduction of Project Schematic Concept

By far the most important new feature in R12 of Workbench is the new concept of the Project Schematic. We covered this in detail in the last issue of The Focus [link](#) so there is not much to add here other than to say that we really have grown to like it and it does make our simulations easier to understand, simpler to reuse and faster to iterate on. The graphical representation of sometimes complex analysis processes really resonate with our engineering brains. It makes also makes it easy to go back to old models and see what you did. We expect this way of doing things to be copied by ANSYS, Inc.'s competitors over the next couple of years.

Beyond the Top 10

This article focuses on the 10 things introduced at R12 that PADT thinks are significant. But in reality they are just a few of many important changes to the software that will change the way most analysts do their job. The gap between when this article was written and published has allowed PADT to use R12 even more, and as we do so we find more and more that it allows us to do things much more efficiently than we thought. And we are still learning.

The most difficult part of taking advantage of these new capabilities, and we say this in every issue of The Focus, is stepping back and letting go of the old way of doing things. As a real world example, I was trying to demonstrate to someone how to do a typical turbine blade geometry optimization (hot-to-cold) in Workbench. I kept using complicated APDL commands imbedded in my tree and following the same convoluted process I've used for over 20 years. Then I tried it again assuming I knew nothing - the result was that I could set up the problem with a few input and output parameters and DesignXplorer. Nothing complicated, no imbedded commands. Just a few clicks.

So please, take a look at R12 and see what your top 10 features are. And if you can not find something you need, contact your support provider or talk to your co-workers. If you look at the problem differently, you may find what you want.



Phoenix Analysis &
Design Technologies

ABOUT PADT...

In the past we have finished up The Focus with a page we called "Shameless Advertising..." The truth was that the page was really only advertising PADT, and PADT related things. So, instead of doing advertising we thought we would just dedicate the final page to explaining who PADT is, what we do and how we can hopefully help you. And, to make sure you read it, we will try and stick something funny in. Want to know more? Call Stephen Hendry at 207-333-8780 or e-mail steve.hendry@padtinc.com.

What PADT Does

One of the reasons why PADT's full name, Phoenix Analysis & Design Technologies, is so long is because our initial vision was to provide a complete mechanical engineering solution. In fact, we almost called ourselves Phoenix Analysis, Design & Manufacturing Technologies. But PADMT is even more annoying than PADT! We manage this diversity through three business units: CAE Sales & Services, Product Development, and Rapid Manufacturing.

The CAE Sales & Services group is the group involved in all things ANSYS. Besides publishing The Focus, the CAE team does sales in Desert Southwest of ANSYS, Inc. and Stratasys products, Technical Support for our ANSYS customers, Training and a large amount of consulting where we use ANSYS tools to solve simulation problems for our customers. This part of the company makes up half of our technical staff.

The next group is Product Development. They do complete research, development, design and production of mechanical and electro-mechanical systems. Most of our work in this are is focused on Medical Devices, Alternative Energy and Fuel Cells, and Semiconductor Manufacturing Equipment. With design engineers, laboratories and a stable of high quality vendors, our PD team is capable of taking almost any idea from concept to product.

The Rapid Manufacturing part of the company is the smallest, but the one with the most customers. We have SLA, SLS and FDM systems in house as well as a full model shop with an NC lathe and mill. In addition, PADT can do prototype molding with soft tooling or rapid tooling. And when customers are ready for production we can serve as their project coordinators to get injection molded parts made and assembled off shore.

So, no matter what your needs, PADT can probably help with software, hardware, services or a reference to one of our partners. Learn more at www.PADTINC.com



PADT on the Web

www.PADTINC.com	PADT's main website
www.PADTMedical.com	Medical device development
www.DimensionSCA.com	A machine that PADT makes
www.uPrintStore.com	A place to buy 3D Printers & Supplies
www.XANSYS.org	ANSYS User forum

Need ANSYS Help?

PADT can help in many different ways, here are a few:

- We hold training here or at your facility <[link](#)>
- Leverage our APDL knowledge with the APDL Guide <[link](#)>
- Consider one-on-one support through mentoring, a great way to get a quick start on something new <[link](#)>
- Attend a PADT Webinar <[link](#)>

Join us on Facebook!

Search for PADT, Inc. And become a fan!

Humor:

How can you tell if your child is going to be an engineer? Watch for these tell-tale warning signs:

- You buy your child an educational software program, and she asks which authoring tool it was written in.
- He has removed the voice box from his Talking Elmo doll and reprogrammed it to recite the periodic table.
- He is picked last on every sports team.
- He throws a temper tantrum every time you refuse to take him into Fry's Electronics
- She has accepted a scholarship to MIT. And she's five.
- He gets in fights in school because he owns a MAC and the other kids use a PC.
- He/She can't get a date.
- Forget Dr. Seuss and Beatrix Potter. She wants you to read her Hawkings.
- When he is asked to play the Star of Bethlehem in the Christmas pageant, he asks, "Am I a white dwarf or red giant?"