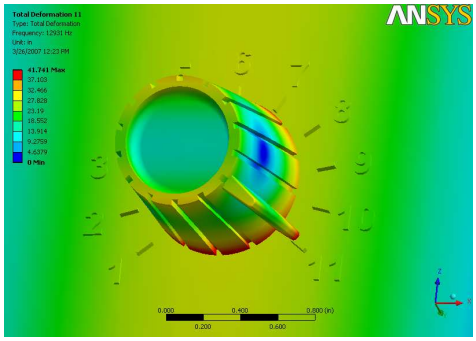
The Focus

PADT

April 20, 2007 | A Publication for ANSYS Users | Issue 56

This One Goes to 11!



By Eric Miller

PART 1: ANSYS Enhancements

We have been talking up Version 11.0 of the ANSYS, Inc. products for some time, and now that it is released we can finally start to share with "The Focus" readers what our engineers find in this major release. Consider this article an executive overview of new features and capabilities focused on pointing out what we feel is significant and letting you know how it can help you. We are skipping about 75% of the changes and only highlighting two or three in each part of the software. For the remainder of the year we will have detailed technical articles delving deeply into key areas, but if you want to know more, come to a PADT update seminar, attend an ANSYS, Inc. WebEx seminar, or read the release notes that come with the 11.0 documentation.

Before we start marching through features and functions, and to understand a little better why things are the way they are, we should step back and look at some key non-technical factors that had an impact on 11.0: 1)The entire development team wanted to take the time to make 11.0 more robust and to track down as many issues as possible before release, 2) ANSYS, Inc purchased FLUENT Inc, a company that actually had more people than ANSYS, Inc. did, 3) a strategic decision to provide users with a "system level" solution for simulation, and 4) a business decision to increase growth in large enterprise size customers.

All of these combined to force a change part way through the development process including being more cautious about rushing new features out the door, support FLUENT's organizational needs, add more time-transient and rigid-body capabilities, and address legacy model and interoperability with competitive codes. Not only was development direction changed, but the development schedule was stretched out and a much longer testing cycle was added to the end. In the end, PADT feels pretty strongly that these changes made sense from a business perspective, and resulted in a much better technical solution.

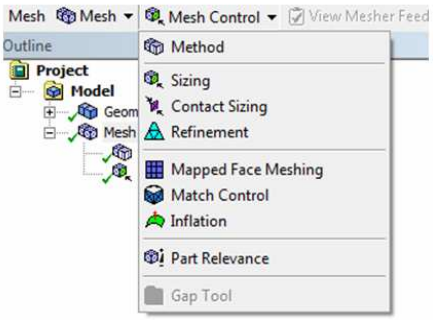
If we covered ANSYS, Workbench, TAS, ICEM CFD, AUTODYN, etc… we would fill up 20 pages. So in this issue we will only be looking at significant enhancements in ANSYS. Workbench will be in the next issue then we will cover all the other products. A key thing to notice while reviewing these changes is that fact that development on the core ANSYS product is still very robust and technically aggressive. Almost all of the features listed here were added to add capability that users have asked for directly.

(Cont. on Pg. 2.)

11.0 In Depth: Simulation Meshing

By: Doug Oatis

If you're like me, and by that I mean super-control freak when it comes to your mesh, the meshing in Workbench v10 sometimes fell short. This was especially true when it came to complex hex/sweep meshing. There was no way to explicitly define the source and target of your mesh. This, along with many other things, has changed in v11..

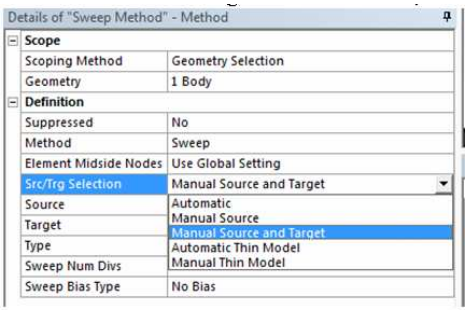


If you had played with the beta version, or have installed and are running the released v11, there are too many additions to cover in a single article. One of the first things you'll notice in Simulation is the meshing controls that are now available.

Using the 'Method' control, you can specify whether you want to use the automatic mesher, Tetra (from ICEM), sweep, or CFX-Mesh (which used to be its own independent module). Using the method, you can also control mid-side node usage on a part-by-part basis.

If you set the 'Method' to be sweep, there are several options that are available. The one that I like the most is being able to manually define either just the source, or both the source and target. This control allows you to specify

(Cont. on Pg. 6.)



Contents

(This One Goes to 11, Cont.)

Debonding:

This capability started out as a separate element but users soon asked if it could be moved to the generalized contact elements, and development listened. So now you don't have to have a continuous mesh across your bonded joint and you simply add a new material property (`TB, CZM`) to your contact elements. You can specify different criteria for mode 1 (separation) and mode 2 (shear) failure and can use energy or gap distance/sliding models.

Non-Linear Stabilization:

Have you ever watched a model with local buckling or wrinkling diverge and day dream about hooking up little dampers to every node in the model? In version 11.0 you can now tell ANSYS to do that for you and it will add small damping to every free DOF using the new `STABILIZE` command. What is cool is that it does not use a damping matrix (that would really slow down the solution) but instead calculates the force from damping and applies it to the DOF's. There is no such thing as a free lunch, so turning this on can slow down an analysis. It is recommended that you run without stabilization and if you model diverges, use a restart with it on and step through the "trouble area" doing a another restart with it off once you have passed through.

Element and Solver Support for Multi-body Dynamics:

To better support system level models, a lot of changes were made in Workbench and ANSYS. On the ANSYS side the most significant enhancements "hinged around" updating and improving the `MPC184` family of joint elements. ANSYS now supports: spherical, revolute, universal, slot, point-in-plane, translational, cylindrical, planer, orient, rigid, and general joints. Another nice changes is that you use `SECTION` commands to define the joints, making it easier to create and store definitions. On the solver side, a lot of work went in to speeding up implicit transient dynamics to get better performance on rigid-flexible system models. In addition, a new solver was introduced to solve all-rigid models. This evolving technology combined with users' continues to shape the direction of simulation tools..

PCG Lanczos:

Every once in a while ANSYS, Inc. introduces a new solver that changes everything. This time they replaced the sparse solver inside the Block Lanczos solver with the PCG solver and created a new solver for Eigenvalue solution of large (greater than 1 Million) DOF that is very fast. The basic difference is that the sparse solver does a ton of I/O whereas the PCG solver uses memory. So, if you have a ton of RAM, you can crank through big model runs very quickly. To prove that point, we solved a 20 MDOF problem with 30 GB of RAM in about 12 hours. It simply filled up the disk and crashed when we used `LANB`.

Rotordynamics:

Since about half of PADT's staff comes from the turbomachinery world, we are crazy about Rotordynamics (RD). At 11.0 a lot of critical details have been worked out based on feedback from users around the world who have really taken to the enhancements made at 10.0. To summarize a few key changes at this release: Most of your common structural elements now support `CORIOLIS` effects, a new bearing element (`COMBI214`) was introduced to make it easier to specify non-linear bearing behavior, orbit printing, plotting, and animating was added, and major changes were made to the solution process and Campbell plotting to allow for the inclusion of prestress effects during post processing. Based upon our usage and discussion with a couple of heavy users, the consensus is that most people should no longer feel the need to use in-house RD tools. If you find something you need is missing, please let us know what so we can get feedback to development. For more info on this feature, check out the article in the last "The Focus:" Spinning up Rotordynamics.

CMS Enhanced

Another major area of focus to support system modeling is support for the Component Mode Synthesis method (See issue 43 for a description of the method and ANSYS capabilities at V 10.0). At 11.0 CMS now supports static, transient, harmonic and spectrum analysis. In a key enhancement for modeling large mechanisms, CMS now support large deflection and substructure prestress. And if you are a CMS junky, you will also be happy to hear that Residual-Flexible Free interface conditions are now supported.

New Element: CONTA177

Rounding out the generalized contact family, 177 is a line-to-surface element that lets you model contact between beams and surfaces or the edge of shell surface to a general surface. It works like the rest of the family and replaces the need to model such situations with node-to-surface contact. We expect to see heavy use with MPC bonded contact to connect the ends of shells to shell surfaces at T-Joints.

New Element: SHELL281

This is basically an 8-noded version of the Shell 181 and brings the new element technology to the 8-noded shell topology. It was added based upon (Cont. on Pg. 3.)

Resources

Want More information about Version 11.0? Besides the thorough documentation in the help manual under Release Notes, visit the ANSYS inc. website and download the webex seminars you may have missed in March.

Under the Customer Portal, Left Menu, select WebEx Presentations under the Product Information menu.

http://www1.ansys.com/customer

Resources

Review ANSYS, Inc.'s Quick Start Guides which are multi-media training sessions, similar to a slide show, but with user controlled pace, downloadable files, and index, etc.

So far they are all Release 11.0 related, focusing primarily on the Workbench Interface.

See menu on the left side of the Customer Portal:
http://www1.ansys.com/customer

(This one goes p to 11, Cont.)

in-house testing and user input to provide more accurate and stable modeling of buckling and wrinkling problems when using shell elements.

New Element: USER300

A long time request from users has been to make the creation and control of user elements easier. After a lot of research and discussion, development has added the `USER300` element. You still have to create some FORTRAN code, but it is much more straightforward and you control the elements behavior with APDL and the new `USERELEM` and `USRDOF` commands. Basically, they have created an element API that greatly simplifies the definition and control of the element so you don't have to get down-and-dirty anymore. Check out the Programmer's manual, Section 6.1.1 for details.

Drucker-Prager Placticity

Every once in a while a new material comes along that everyone gets excited about. We started getting requests for Drucker-Prager about 2 years ago and it became quite the trend. So development did their research, found some verification problems, and plugged away to deliver this rather interesting model for materials that have pressure dependent material behavior.

J-Integral Calculations

In the past, if users wanted to carry out fracture mechanics assessment on their models they had to use 2D geometry or a 3rd party tool for 3D. At 11.0 this area has been redone to support 2D and 3D with the 18x family of elements. Results can be viewed with the new `PR/PLCINT` commands and detailed info can be extracted with `*GET`.

Coupled-Field for Electrostatic-Structural

In the past, electrostatic MEMS devices required an iterative solve to capture displacement and electrostatic behavior. At 11.0 the `PLANE223`, `SOLID226` and `SOLID227` elements have been upgraded to support matrix coupling of UX, UY, UZ and VOLT DOF's. This should speed up and simplify a large number of MEMS simulations

New Solver: PCG Lanczos

Modal extractions can now be done using the Block Lanczos method with PCG as the matrix solver. This is much faster and more robust than the standard Block Lanczos which uses the Sparse solver because PCG does much less disk I/O. It is preferred for models with more than 1 MDOF but be warned, if you don't have a lot of RAM available, it will swap to disk and you won't see much of an advantage. If you do have RAM, it is very fast and robust.

Reset Picking Button

If you use the ANSYS GUI you may notice that sometimes picking "goes away" A new button in the GUI should allow you to fix this problem. Not a major technical enhancement, but something that will help a lot of people.

Automatic *DIM and no More Need for Compress

You no longer have to create arrays with `*DIM` if you are filling them with vector and matrix commands, it makes arrays on the fly and dimensions them as needed. Another APDL change was a new `*VGET, ParR,ENT,,NLIST` which creates a compressed array of selected entity numbers. So no more `*GET`, `*VMASK`, `*VFUN,,COMP`! Both of these changes show that APDL is alive and well and development is listening to users and making things better for us.

We hope you have found this information useful. As was stated earlier, this is only a sampling of the changes and we have not even touched Workbench yet. Look for more in future issues of "The Focus."

11.0 In Depth: Modeling Bond Failure

By: Rod Scholl

Prior to ANSYS version 10.0, if we wanted to model delamination of two solids, our best option was to use standard contact, and a script which would check the tension between elements and then kill those that exceeded a given criteria.

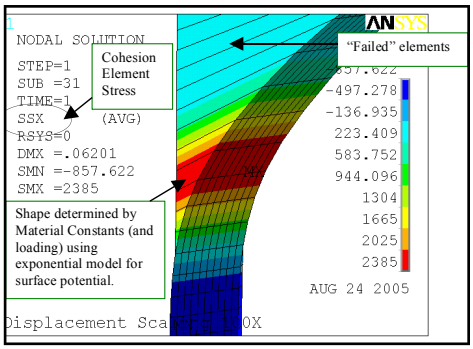
This type of script was highly sensitive to mesh size, and was a little inefficient because one had to build a `*do` loop to go into `/post1` to evaluate the contact elements, kill the "failed" ones, and then do a restart in between each substep. Also, this method is highly path dependent, meaning one might need to model the whole "unzipping" process to get to a final state.

Interface Technology of version 10.0

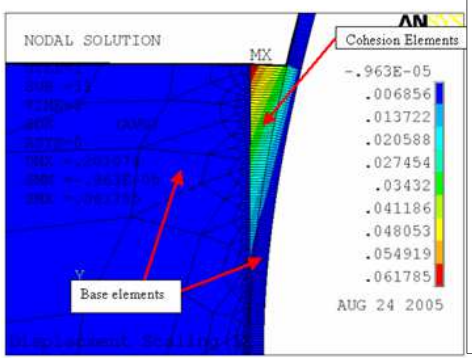
At 10.0, the interface elements (`INT20X`) were introduced which not only automated this interface failure based on criteria, it also implements a function to describe the separation before failure of the interface substance, which we will call "epoxy". This function allowed for capturing the load transmission deep into an unzipping seam using a very accurate exponential function.

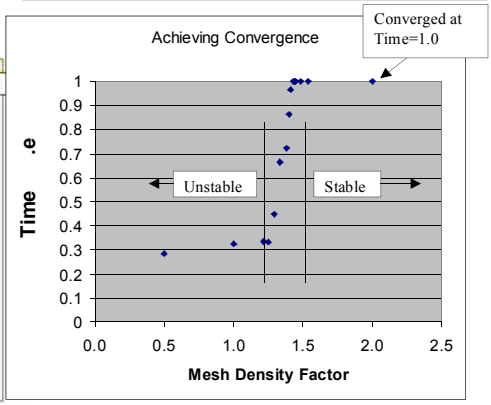
As we explored in the past article, this methodology was easy to implement, although convergence required some critical mesh density as shown to the right:

Thus for the epoxies I typically encounter, the required mesh density made this approach too expensive for 3-D analysis.



(Cont. on Pg. 4.)





(Modeling Bond Failure, Cont.)

Version 11.0 Enchancement

Now in 11.0, we have another option for modeling separation. This new approach conveniently uses the existing contact element technology. One big advantage is unlike the INT20X method, one doesn't need mesh continuity at the interface. In fact the implementation is a snap, requiring only a couple commands beyond what normal contact requires. As usual, the help documentation is fantastic and can step you through the implementation… you will likely need to go to the theory manual 4.11 to understand the material property implementation – but here's my basic instruction overview:

1) Build contact normally
2) For the elements suspect to separation, create a new material property using TB,CZM
   a) TB,CZM,*Matl.#,,,CBDD or CBDE*
   b) TBDATA,*Matl.#,,,epoxy info.,damping coefficient*

That's it! Just by the presence of the CZM property those elements will now expand and fail according to the epoxy information entered on the TBDATA command. Also, the data input is fairly intuitive, such as maximum tensile stress… and max separation at failure.

Converting Epoxy Material Data to TB,CZM and FKN

The arguments for the CZM model are show below for the CBDD case. (The CBDE arguments are similar but use an energy criterion rather than gap at debonding.)

Constant	Symbol	Meaning
C1	$\acute{o}_{max}$	maximum normal contact stress
C2	U^C_n	contact gap at the completion of debonding
C3	τ_{max}	maximum equivalent tangential contact stress
C4	U^C_t	tangential slip at the completion of debonding
C5	η	artificial damping coefficient
C6	β	flag for tangential slip under compressive normal contact stress

Also, for this article I think we can get a good understanding by considering only C1, C2, and C5 as though our epoxy only fails in tensile stress.

So let's look at the implementation for a test case:

Failure Stress	**1000 psi**
Epoxy Modulus	**2E6 psi**
Epoxy Layer Thickness	**0.003"**
Max Elongation	**50%**

With this data and a failure of the epoxy at 50% elongation we have a C1 of 1000 psi and a C2 as follows:

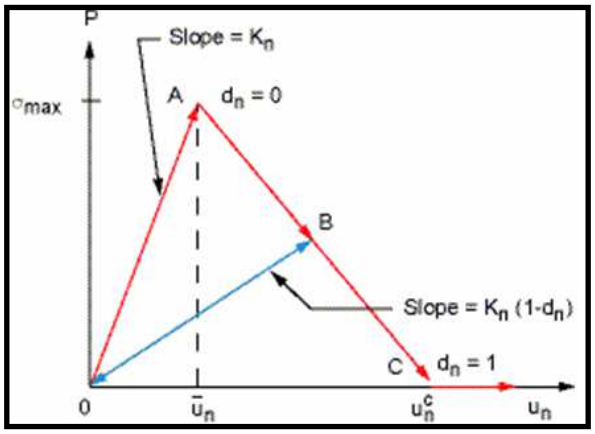
```
C2 = Max.Elongation*EpoxyLayer thickness
```

And thus we have:

```
C2= 0.0015"
TB,CZM,1000,0.0015,,,C6(η)
```

ANSYS implements this as follows:

Following the red line is the implemented load path. So we can see we have some flexibility to change $Ubar_n$ to have this bilinear curve match our material data. Thus the more quickly the epoxy unloads after reaching $\acute{o}_{max}$ – the closer $Ubar_n$ should be to C2 (U^C_n) …



To control $Ubar_n$ and hence the curve shape of our epoxy we need to control the slope Kn, which is simply the FKN real constant for the contact pair.

So for the case where $Ubar_n$ is 75% the way to (U^C_n) we calculate FKN as follows:

```
FKN = -C1/(0.75*C2)
```

Note that the negative value tells ansys to interpret FKN as an absolute value and not based on the underlying element stiffness/dimensions.

Presto! You now have a bilinear implementation of your epoxy's load/displacement using existing element technology and a few arguments on a TB,CZM command

Test your Properties

While investigating this new feature, I built a script which pulls apart two blocks and measure the force required to do so. Use this script to enter your material properties and then verify you are getting the force/displacement behavior your expect. It seems to be stable over typical values.

Watch your Pinball Size

Keep an eye on your contact pinball radius… if the epoxy deflection allows the elements to move outside the pinball radius… the contact will let go, no matter HOW strong of an epoxy you are modeling.

Damping Coefficient

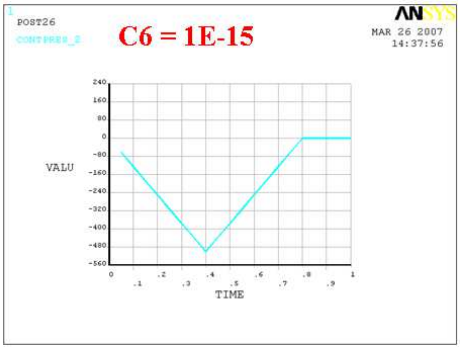
So stability is going to play a big role in the actual implementation of this methodology. Although it was not necessary for the simple blocks in pure tension of the script above, you will likely needed it for peel loading scenarios. The good news is that, no matter how stiff your epoxy, given a large enough stability number, convergence will be a snap. The bad news is that accuracy of failure stress of the elements degrades rapidly with changing this stability factor. This C6, damping coefficient implements a force balance based on time-step size and change in deflection. Thus changing C6, or the time between substeps will have an impact on results. (See the new STABILIZE command for what I suspect is a very similar implementation.) Although the manual recommends stability numbers like 0.1, and 0.01 – I found that I was much happier with the accuracy afforded by 0.001 or 0.0001 and below. (I also note that the VM test case uses 0.0001).

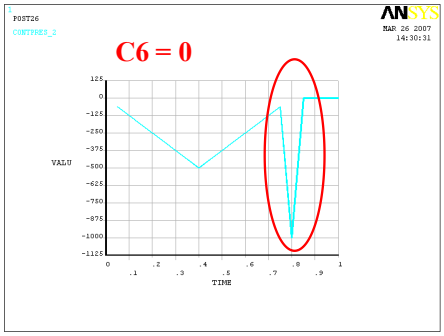
(Cont. on Pg. 5.)

(Modeling Bond Failure, Cont.)

Do not use a default or 0 stability factor (C6)!

It seems there is a requirement for some entry on the stability factor. Even something inconsequentially small like C6=1e-15 will do. Just not a zero or you may get an odd increase in load to debond as shown below:



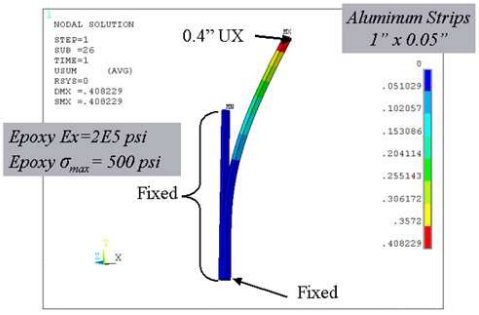


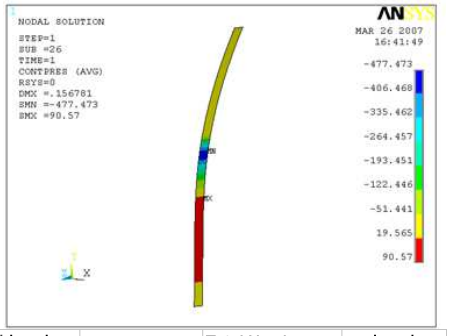
To give us a feel for what stability factors may be necessary and the domain of epoxies that require small factors (and hence give high accuracy predictions) I explored a test case for various factors using this script.:

Note that some scenarios require hundreds of equilibrium iterations for convergence. You might alter the test script for your particular epoxy properties, application thickness, and then determine what damping coefficient (C6) meets your required accuracy. Note how in cases A-D that the actual failed bond length is quite dependent on C6 but the last little bit of accuracy requires many iterations to achieve.

As Case F indicates you also might lean towards a bilinear curve of your epoxy's load/displacement that leans towards a small ratio $(Ubar_n/U_n^c)$

Good luck, and happy debonding!





Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
2.00E+06	2500	50	0.003	0.5	0.00001	2344	1550	11
2.00E+06	2500	50	0.003	0.5	0.0001	2205	1185	11
2.00E+06	2500	50	0.003	0.5	0.001	2375	441	10
2.00E+06	2500	50	0.003	0.5	0.01	2321	134	8
2.00E+06	2500	50	0.003	0.5	0.1	2625	32	2
Case A	This is a quite rigid and thin epoxy, with 0.1 tip deflection							
	Note the high number of equilibrium iterations required for this scenario							

Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
2.00E+05	500	50	0.003	0.5	0.0001	477	929	33
2.00E+05	500	50	0.003	0.5	0.001	489	644	33
2.00E+05	500	50	0.003	0.5	0.01	486	295	28
2.00E+05	500	50	0.003	0.5	0.1	691	46	9
Case B	This is a more typical epoxy application with thin layer and 0.1 tip deflection							
	C6 = .001 is sufficient							
	Although the iteration number is high... the model did have a debonding length 3-4X that of Case A							

Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
2.00E+05	500	50	0.01	0.5	0.0001	496	247	11
2.00E+05	500	50	0.01	0.5	0.0001	494	254	11
2.00E+05	500	50	0.01	0.5	0.001	481	200	11
2.00E+05	500	50	0.01	0.5	0.01	494	93	9
2.00E+05	500	50	0.01	0.5	0.1	533	37	3
Case C	This is a more typical epoxy application with thicker layer and 0.1 tip deflection							
	C6 = .001 is sufficient... iteration count is kinda high... but likely tolerable							

Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
1.00E+04	150	50	0.05	0.5	1.00E-04	148	68	16
1.00E+04	150	50	0.05	0.5	0.001	150	79	16
1.00E+04	150	50	0.05	0.5	0.01	147	66	14
1.00E+04	150	50	0.05	0.5	0.1	150	31	6
Case D	This is a soft, medium strength adhesive, applied generously with 0.2 tip deflection							
	Note the low iteration count for this softer epoxy							

Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
2.00E+05	500	10	0.05	0.5	0.001	470	677	56
2.00E+05	500	25	0.05	0.5	0.001	500	347	42
2.00E+05	500	50	0.05	0.5	0.001	495	175	26
2.00E+05	500	75	0.05	0.5	0.001	496	144	18
2.00E+05	500	100	0.05	0.5	0.001	493	81	11
2.00E+05	500	200	0.05	0.5	0.001	493	35	2
Case E	This is a typical epoxy applied generously, 0.4 tip deflection							
	Note how the low energy epoxies have longer convergence times, because of the unzipping effect							

Epoxy EX	Epoxy Max	Max Elong	Epoxy thick	$Ubar_n/U_n^c$	(damping coef.)	Failure stress	Total Number iterations	unzipped Elements
2.00E+05	500	50	0.05	0.1	0.001	496	73	15
2.00E+05	500	50	0.05	0.25	0.001	484	90	14
2.00E+05	500	50	0.05	0.5	0.001	499	136	15
2.00E+05	500	50	0.05	0.75	0.001	480	233	16
2.00E+05	500	50	0.05	0.9	0.001	495	500	17
2.00E+05	500	50	0.05	0.95	0.001	495	850	17
Case F	This is a typical epoxy applied generously, 0.3 tip deflection							
	Note how (Ubarn / UnC) affects convergence time... with small impact on unzipping length							

(Meshing, cont.)

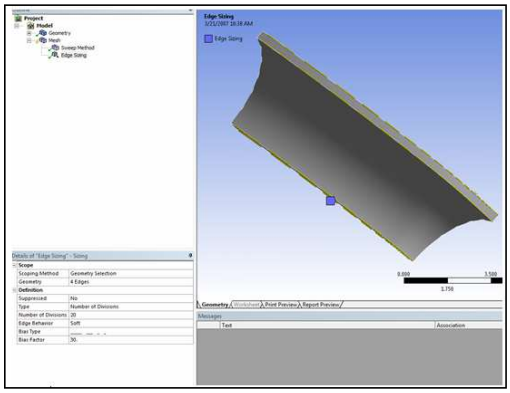
whether you want to mesh "through" a part or "along" a part.

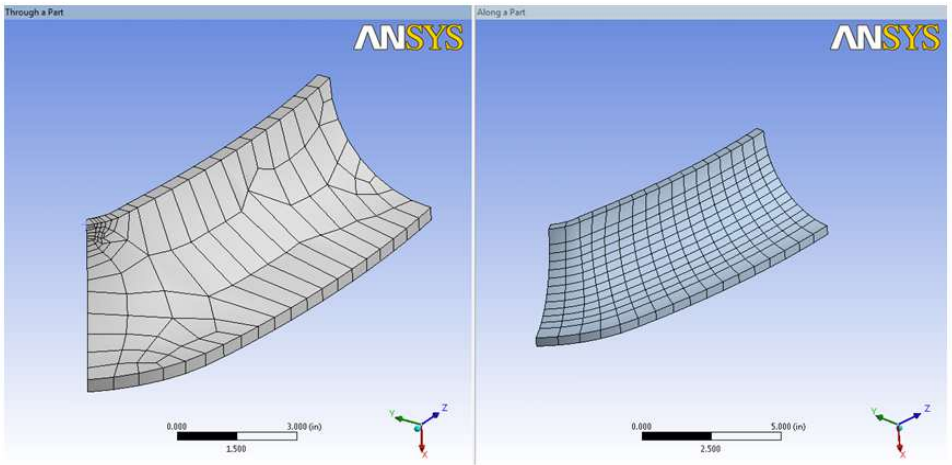
The 'Thin Model' options are a very nice feature, because that's how you specify the usage of the `SOLSH190` element. You will also see the ability to specify the number of sweep divisions and define a sweep bias. In my experience, I've found it easier to specify the biasing on the lines in the sweep direction.

My preferred method is to insert a sizing control on a line, specify the number of divisions, and then enter my bias type/spacing. The benefit of doing the biasing on the line level is that it actually shows you the line divisions in the graphics window.

This is only one enhancement made to the meshing branch of Simulation. You can also use patch independent meshing to tackle sloppy geometry (just like you could in ICEM). There are also new automated settings depending on the type of analysis you're doing. If you select explicit, Simulation will automatically drop all midside nodes and use a slow transition value to go from small to large element sizes.

Look for future articles going over other module enhancements in Workbench v11.





Links

There's a new ANSYS Publication! Subscribe to the "ANSYS Advantage" A CFD and FSI related publication.

http://www.ansys.com/advantage/index.htm

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

Resources

The best resource for Workbench users out there is the ANSYS Workbench Community. Files, discussion groups and related resources help WB users share with each other and the developers' at ANSYS, Inc. It just passed 1400 members:

http://www.Resources ansys.com/customer

Upcoming Training Classes

Month	Start	End	#	Title	Location
Apr '07	25-Apr	27-Apr	152	ICEM CFD/AI*Environment	Tempe, AZ
May '07	2-May	4-May	104	ANSYS WB Simulation - Intro	May '07
	7-May	8-May	100	Engineering with FEA	Tempe, AZ
	10-May	11-May	203	Dynamics	Tempe, AZ
	18-May	19-May	106	ANSYS WB Design Xplorer	Tempe, AZ
	21-May	22-May	105	ANSYS WB Sim Struct. Nonlin.	Tempe, AZ
June '07	4-June	6-June	101	Intro. To ANSYS, Part I	Tempe, AZ
	11-Jun	12-Jun	201	Basic Structural Nonlinearities	Tempe, AZ
	13-Jun	14-Jun	204	Adv. Contact & Fasteners	Tempe, AZ
	21-Jun	22-Jun	301	Heat Transfer	Tempe, AZ
	25-Jun	26-Jun	107	ANSYS WB DesignModeler	Tempe, AZ
July '07	9-July	10-July	104	ANSYS WB Simulation - Intro.	Albuquerque, NM

Links

To keep up to date with global and European events and news, read ANSYS, Inc.'s newsletter, E-News:

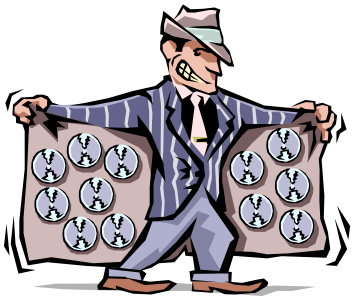
http://www-harwell.ansys.com/newsletters/index.htm

News

ANSYS partners with PCA Engineers LTD. To integrate VISTA (TM) (VISual Turbomachinery Analysis) with ANSYS® solutinos.

Online news release.

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



The Shameless Advertising Page