The Focus

A Publication for ANSYS Users

Contents

Feature Articles
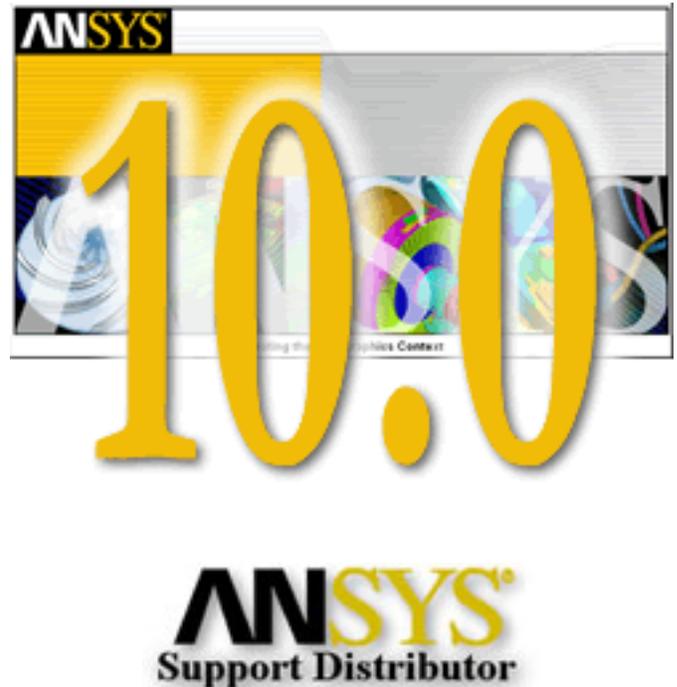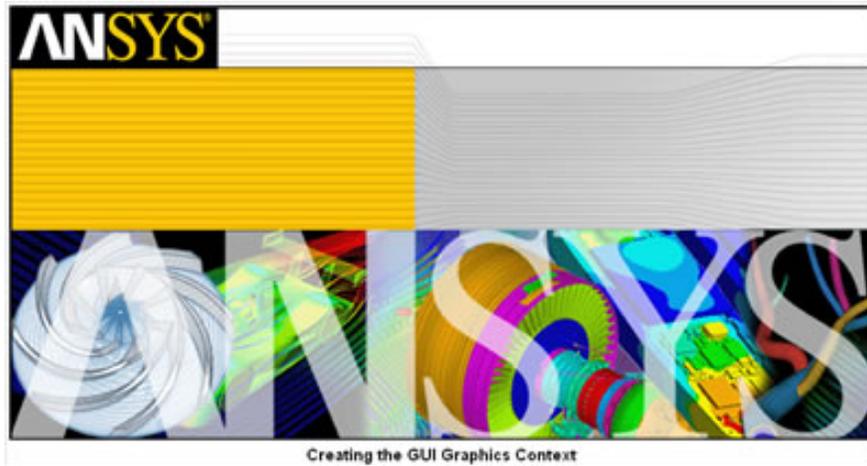
On the Web

Resources

A Publication for ANSYS Users

10.0 Features (for the Structural Thermal Analyst)
by Rod Scholl

The best way to learn about the new features in ANSYS 10.0 is to open the help section, and at the beginning is a chapter with Release Notes describing new features. Read it!

Creating the GUI Graphics Context

Too long? Well, you and me would have made good study partners in college. Since my analyses are predominately structural/thermal, I scan the notes for things that make my job easier. So this is my completely biased selection of what excites me most.

1. Licensing:
   a. During 10.0 installation, be sure and load the new FLEXlm version or things might not work right.
   b. The ANSYS Launcher has more features.
2. Contact
   a. 3-D Beam-to-Beam contact. Crossing beams can contact based on sections. Even think about a cable threaded through the interior of a hollow tube!
   b. A damping term can be included between surfaces. This should greatly reduce the "pass through" and other rigid body motion issues when establishing contact.
   c. Orthotropic friction model! Now MP,MU,mat#,value is out of favor compared to the more extensive TB,FRIC command.
   d. SOLID186 now has Layered Solid abilities like the SOLID46 element.
3. Manual Rezoning
   a. Now supports flexible-flexible contact and more material models.
4. MIDTOL gives greater control of Automatic Time Stepping in dynamic analyses. Test case shows 30% decrease in solve times.
5. NLGEOM should run a little faster and have smaller .esav files.
6. Rotor Dynamics:
   a. CORIOLOS command includes inertia effects!
   b. Campbell Diagram tools added.
   c. Gyroscopic Terms for stationary and rotating frames.

The Focus

7. Inertia Relief ability in substructure matrices.

8. MPC element has sliding/slotted joint ability.

9. Cohesive Delamination with new INTER20X elements.

10. Anisotropic Hyperelasticity model!

11. PCG more robust for ill-conditioned problems.

But really, don't take my word for it. Find the full list including enchancements to distributed solvers, changes to FSI, not to mention changes in Workbench and other ANSYS products.
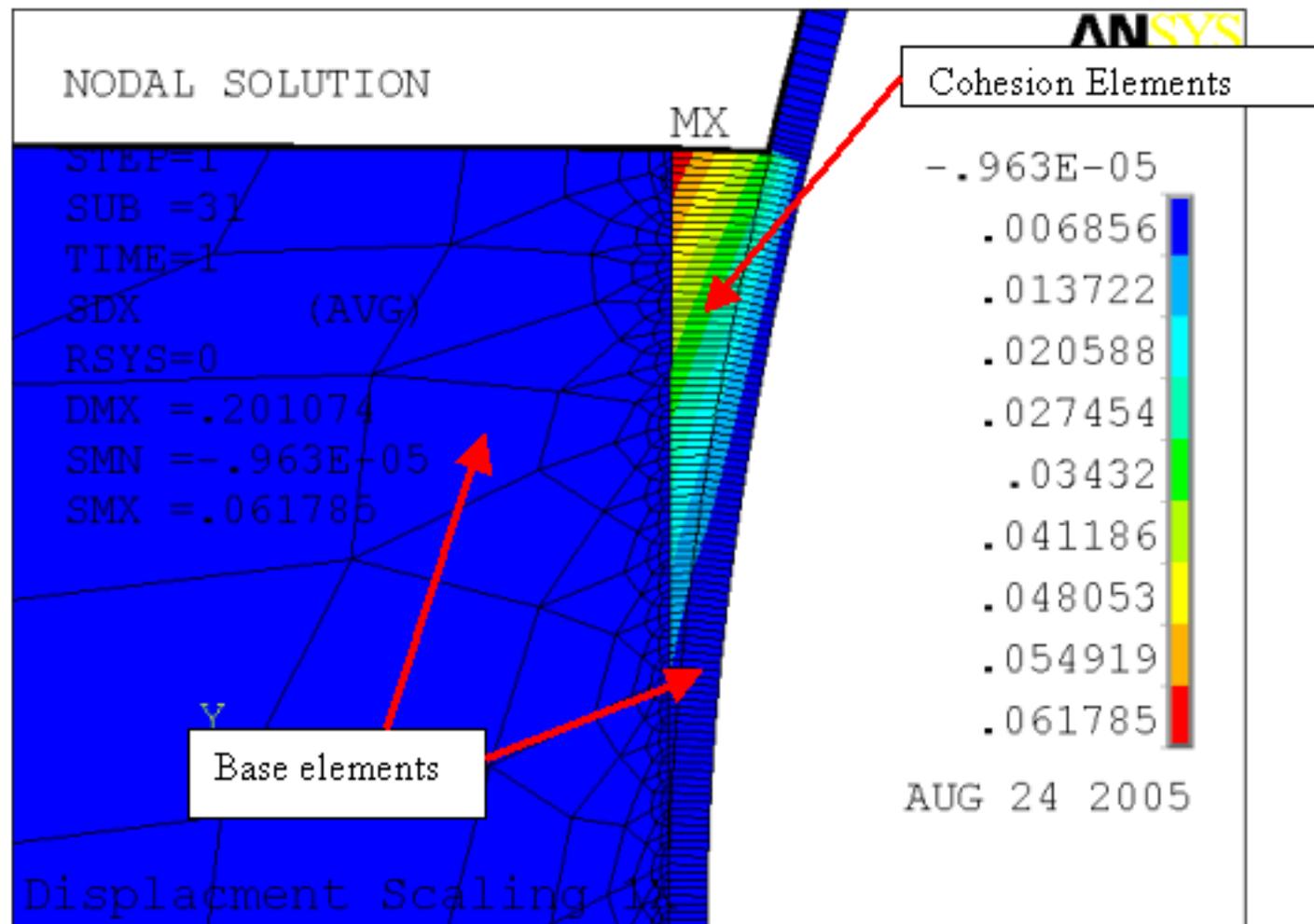
The Focus

A Publication for ANSYS Users

Cohesion in 10.0

by Rod Scholl

ANSYS 10.0 introduces the ability to model cohesion. Essentially this allows for the easy creation of initial zero-thickness elements between the boundaries of some "glued" components. These zero-thickness elements will then open up in peel and shear and even "let go" when a maximum stress is encountered.

What makes this superior to solid elements with failure criteria, or even contact elements with birth and death based on contact results and utilizing restarts, is that the formulation allow for modeling the exponential surface potential of an adhesive in deformation. Thus, through specified cohesion element material properties the amount of deformation, and subsequent load-sharing with adjacent elements is captured.

The Focus

A Publication for ANSYS Users
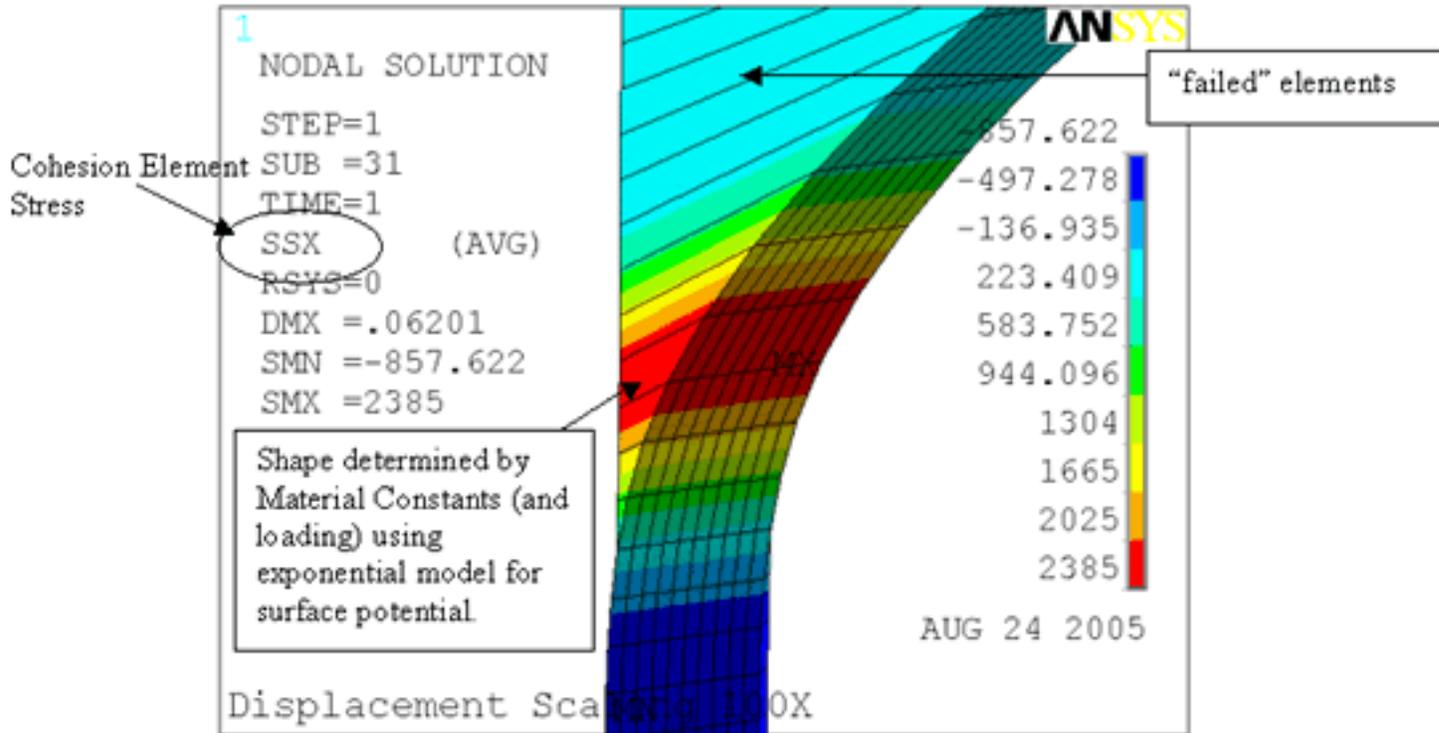
The help manual describes how to input material properties for the cohesive region — and cohesive elements are generated pretty easily:

1. Define element type INTER20X to make the cohesive elements available.
2. Define INTER20X material properties in terms of maximum stress and separation length at failure stress (see materials manual).
3. Build the joint region line-on-line and share nodes along a defined separation surface. Surfaces can be identified by an area, or by element components on either side.
4. Ensure that there are no MESH200, Shell, Contact, or other elements sharing nodes with the cohesive joint. They can be applied later, but cannot be present during the creation of the cohesive elements without an error message.
5. Issue the CZMESH command referring to the element components on either side, or an area number if two volumes share it. ANSYS then separates the nodes and paints the surfaces with the INTER20X elements.

That's it. From there, you can solve the model normally. There are a couple post processing terms for epoxy separation and epoxy stress.

The help manual is pretty clear on their usage, so combined with the attached example script, tchohesion.mac, you should find implementation simple.

The Focus

A Publication for ANSYS Users

Modeling Stiff Epoxy

By stiff epoxy, I refer to something about 0.01" thick, with a stiffness of 105 psi and maximum stress of 1,000 psi. Such an epoxy only expands 0.0001" before giving way. This scenario is difficult to achieve convergence without resorting to a very fine mesh. For 2-D models, the mesh density requirements may not be significant, but for 3-D models prepare for long solve times.

At the load or substep that cohesive separation begins to occur, you may find non-convergence. Increasing mesh density may not have any effect for a while, until a minimum mesh density is achieved. At this point, the debonding analysis will struggle along. A slightly increased density beyond this, and the analysis is quite stable. See the graph below created by varying MSIZE in the script tcohesionhard.mac. Thus, a 10% increase in Mesh Density (MSIZE) resulted in about a 10% decrease in element edge length.
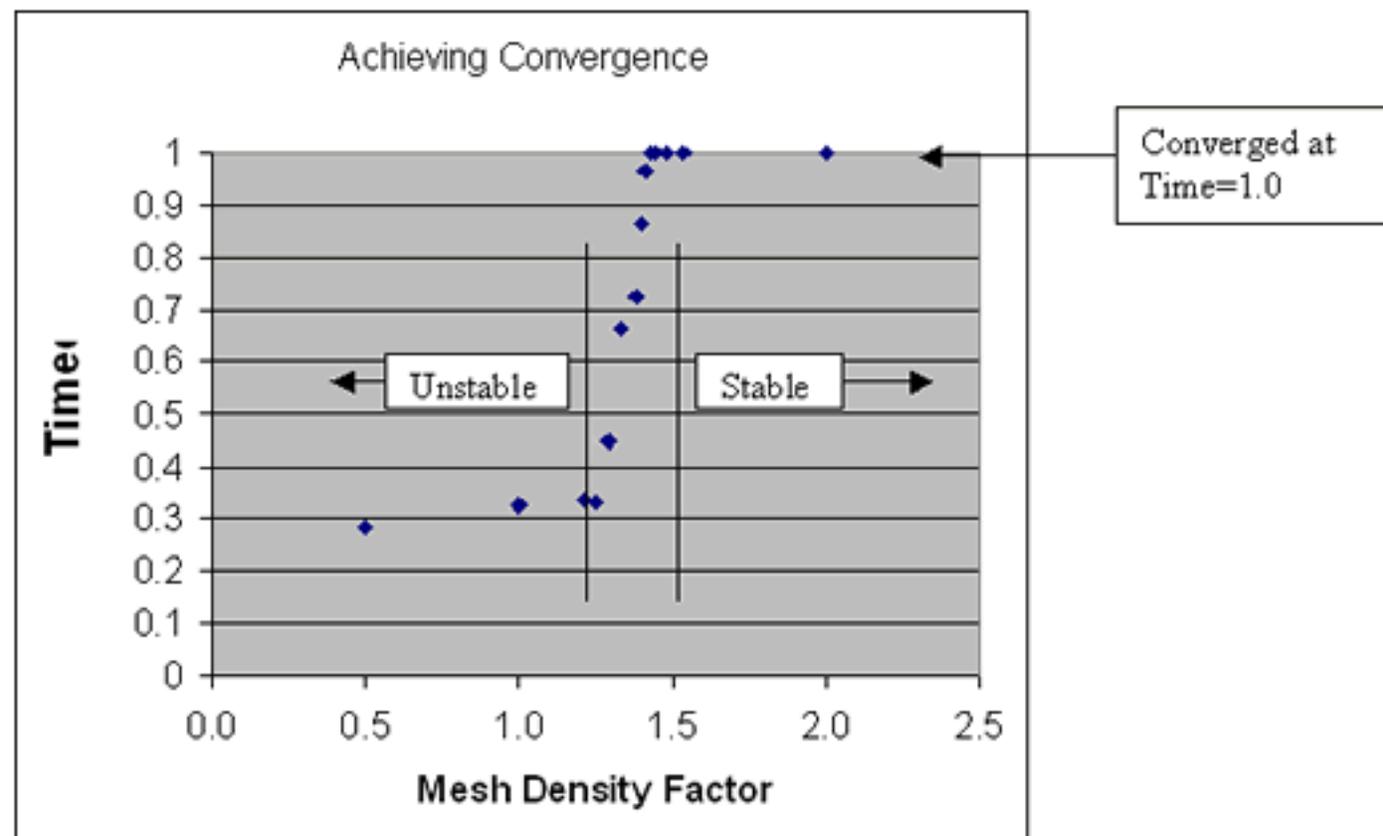

Also note that for some problems, increasing mesh density slightly beyond the minimum necessary for convergence greatly reduced the number of equilibrium iterations required (by a factor of 5). However, for other problems, the behavior was reversed, where increasing mesh density *increased* (5X) the number of equilibrium iterations required! If you have any unexpected behavior, try moving the mesh density a little.

The Focus

A Publication for ANSYS Users

Also note that density in the peel direction is important, but edge effects may also slow convergence greatly. As indicated below, beware of your mesh density in the transverse direction as well.
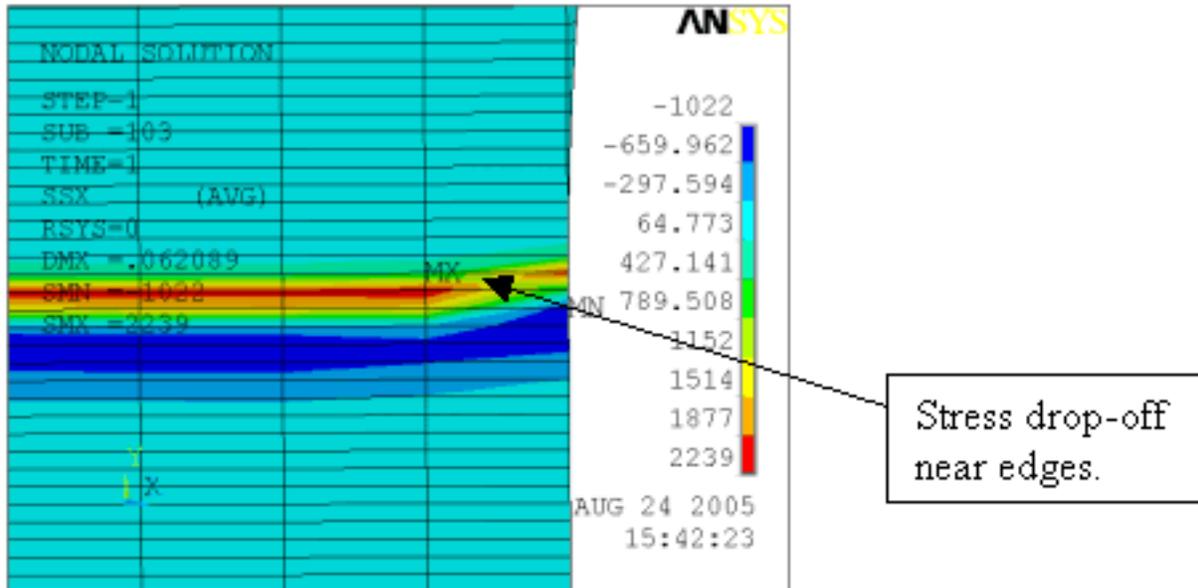

I expect convergence to improve with later releases of ANSYS, and of course this may not be an issue at all for you if your epoxy is soft and thick enough.

As a side note, I tried an alternative approach using a very fine layer of elements at the bonded region. This layer was joined to the substrate via bonded contact. For a given face element size, the node counts were actually similar between a solid mesh with transition to fine, and a coarse mesh with fine substrate layer. I had hoped that the node count would be smaller for the substrate layer approach, but they were in fact quite similar. Some geometries might be more or less conducive to node count reduction by inclusion of a substrate bonding layer. However, the model with bonded contact had less delamination length (20%) and a notably large region of load distribution through the epoxy in the unzipping direction when compared to the results from the straightforward approach. So it seems the substrate bonding layer, although an interesting approach, is not necessarily helpful.
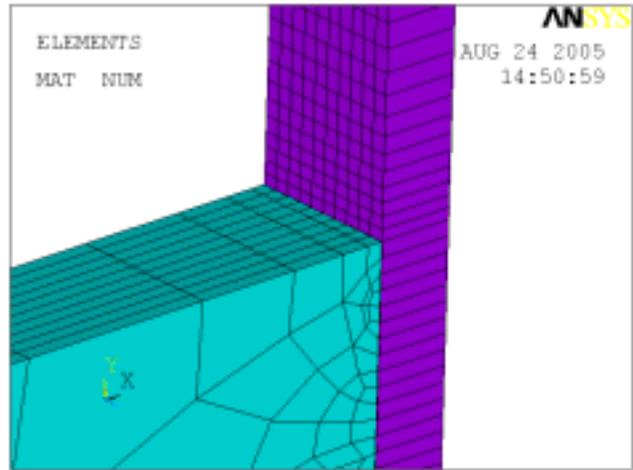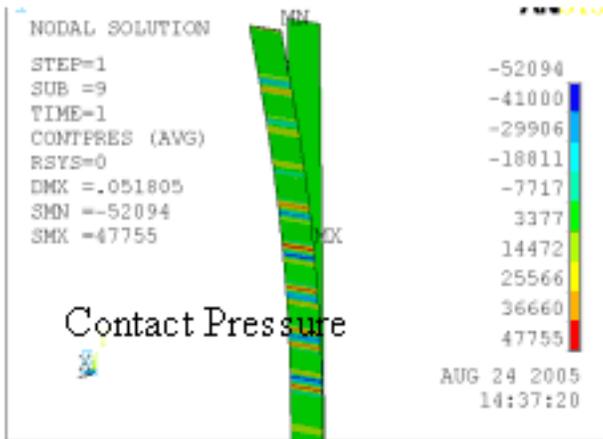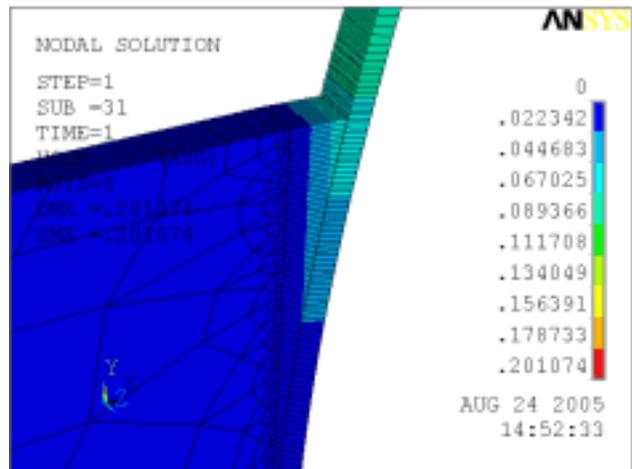
Substrate Layer Approach
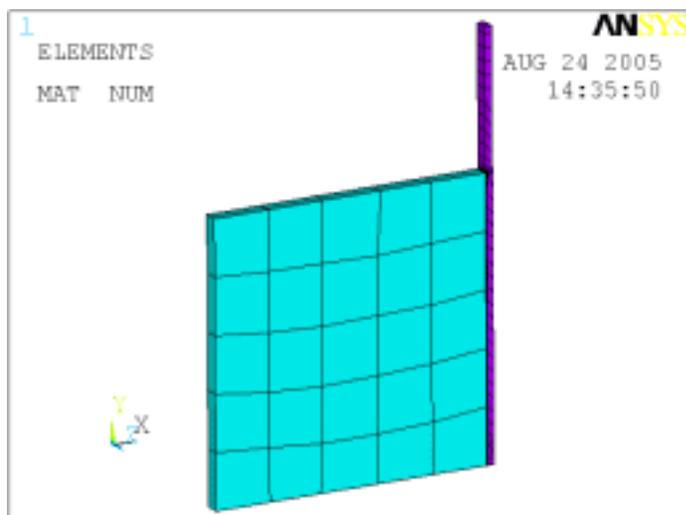
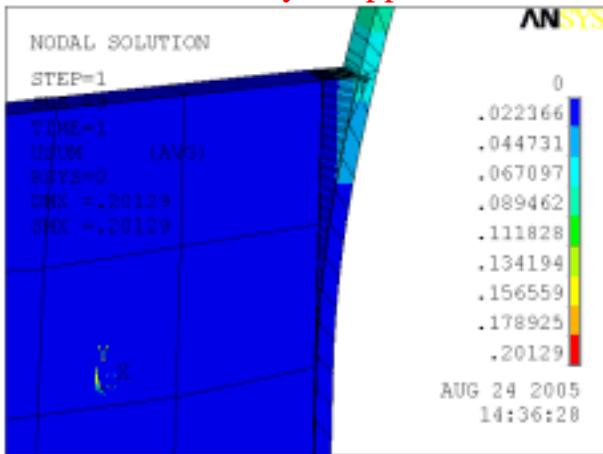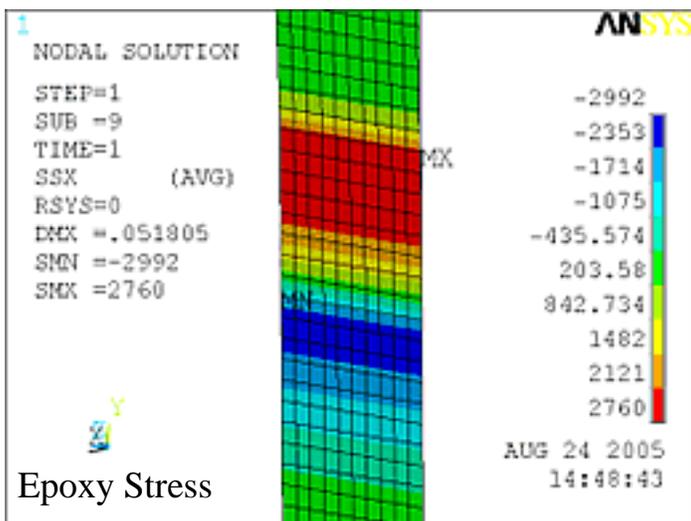Straightforward Approach
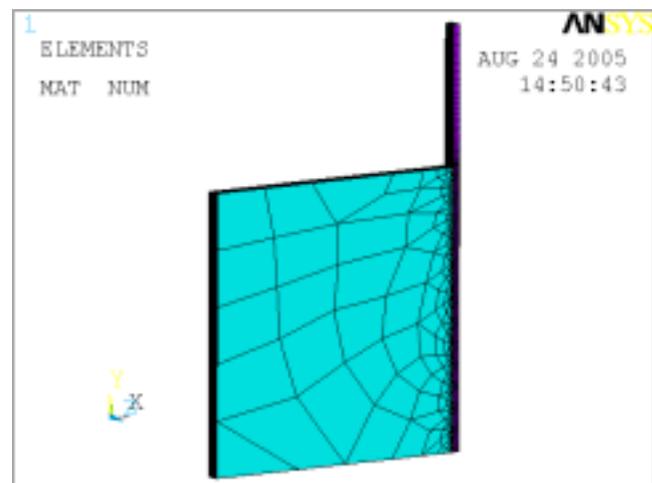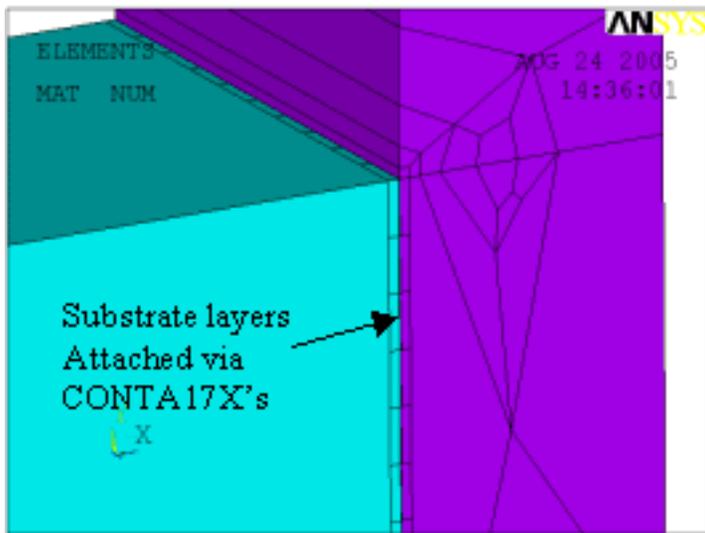
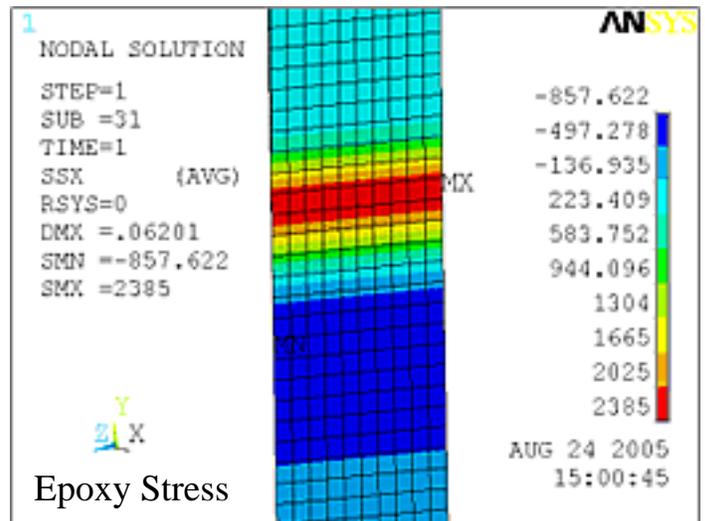Substrate Layer Approach

Straightforward Approach

Contact Pressure

Epoxy Stress

Epoxy Stress

The Focus

A Publication for ANSYS Users

10.0 Input Window Repair

by Rod Scholl

There is a minor bug on the input window of version 10.0. If it is detached from the GUI (by using the icon as shown below) it seems that when one cuts and pastes a series of commands into the window, they don't delete themselves after execution. Thus, after pasting and hitting enter, the commands execute normally, but the text still remains.

Click to make Input Window Float
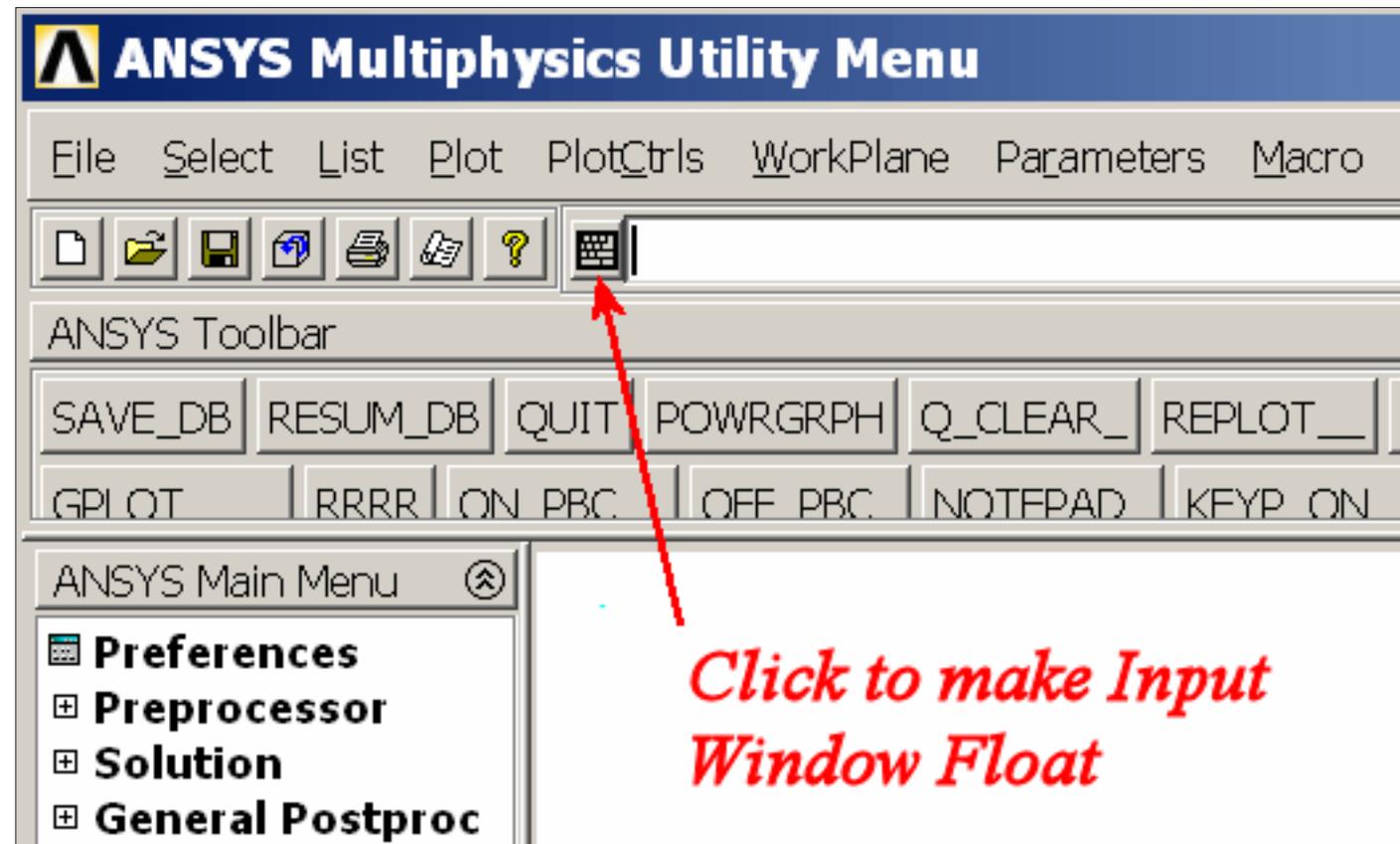
You can do a right-click, select all, then delete. Or, if like me, you enjoy monkeying around in files that are well beyond your knowledge, you can fix the script as follows.

In your installation directory is a folder with a path something like:

C:\Program Files\Ansys Inc\v100\ANSYS\lib\Euidl1.0\scripts

Inside this folder is a file named anscmdprompt.itk, and inside this file is a bit of text that needs to be added at line 433:

catch {$itk_component(f_cmdprompt) configure -state normal}

The Focus

So, at line 433 (near the second occurrence of the text "ans_greygui" if you don't have a line counter), the corrected file should look like:

```
if { $::AnsysGUI::ansGUIArray(WB) && $exiteofCmd } {
  ::AnsysWBGUI::exitANSYS
} else {
  catch {uplevel #0 [list ans_sendcommand $cmd]} err
}
}

if {$numCmd > 1} {
  after 250 [list ans_greygui ALL 1]
  catch {$itk_component(cmdprompt) delete entry 0 end}
  catch {$itk_component(f_cmdprompt) configure -state normal}
  catch {$itk_component(f_cmdprompt) delete 0 end}
}

if {[info exists ::AnsysGUI::ansStatusArray(help)]} {
  set _helpTxt $::AnsysGUI::ansStatusArray(help)
} else {
  set _helpTxt ""
}
```

Voilà! You are back in business!

— *Rod "mouse-click optimizer extraordinaire" Scholl*

The Focus

A Publication for ANSYS Users

From Fryer to Freezer

by Rod Scholl

I grew up in Arizona, lived briefly in Rhode Island, did a quick stint in Holland, and have now settled (fingers crossed) in the Twin Cities, Minnesota.

So I'd lived almost fifteen years as a desert rat. I've even lived in Arizona as a high school and college student. Life, of course, is hotter on someone that age. "Really," you ask? Yes. For example, in my Arizona youth I have:

- Pushed my out-of-gas car along the sizzling blacktop — *barefoot* — 'cause at that age who needs shoes?

- Sat in gridlock traffic in the peak of summer with no air conditioning *and* the heater on to help cool the engine. The shiny black paintjob from Earl Scheib was just an added touch.

- Slept in my car … a *lot*. Many times I would drive to the college, sleep a sweaty fitful sleep in the summer heat, then drive home having missed class.

- Been lost in the desert. They say cactuses may have water in them, but ever try to cut one open with a shard from a broken bottle? That part wasn't shown in the cartoons.

- Ran on foot from scorpion sightings. Mean looking buggers.

- Been to a tanning booth, because who in their right mind would spend time outside? My friends kept asking if I was sick because I was so pale. Okay, okay, I'll get some color.

- Re-roofed a house in summer. One guy joked every day saying it was hotter because we were closer to the sun. Not even funny the first day.

The Focus

And now I will learn to be a snow hermit. My real ambition is to build an igloo. If I can work some FEA into its design somehow, I'll write a Focus article on it. Or, I know! I will take my laptop and do some ANSYS work from inside the igloo! Of course, in order to stay warm, it will be a thermal analysis … hehe. Alright, almost as bad as the "closer to the sun" joke. Maybe I did bake some brain cells.

Anyway, I'm new to the Minnesota area and am looking forward to becoming active with the user community here. Hopefully us locals can all meet to share tips and methods in ANSYS. Maybe even a 10.0 feature exploration?

Additionally, I would like for my outsource analysis work (75% of my daily job) to be native to Minnesota for mutual benefit of proximity. Are you an ANSYS user in the region? Know someone who is? At this point I don't know anyone, so any contact you can help with is appreciated. Just e-mail me!

Funloving ANSYS user seeks same for technical relationship in MN. New to area and looking to meet other people who share my interests. I like long non-linear runs on the beach that include contact and plasticity. I enjoy a good APDL script and talking about new element technology in front of a warm, cozy, CPU. Must like Classic ANSYS and Workbench. No Hard Point users, please.

Going back to the theme of hot and cold, to butcher a Robert Frost poem:

Some say the component will end in Fire, some say in Ice.
With low K and HTC's so dire, I hold with those who favor fire.
But if it had to perish twice, I think I know enough of cracks
To say destruction ice has its attacks
And would suffice.

A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact Rod Scholl at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online library, which can be searched in a variety of different ways.

Contributor Information

Please don t hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to Rod Scholl.

Subscribe / Unsubscribe

To subscribe to   or unsubscribe from   *The Focus*, please visit the PADT e-Publication subscriptions management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.