

Shell to Solid Interfaces

By Rod Scholl

This is probably an ambitious subject... one that, properly treated, would include many equations and derivations of error terms exploring the various methods of interfacing a solid with a thin region appropriate for shell meshing. We would trot out shape function equations, error analysis, and so on.

But of course, we will proceed much more haphazardly... this reminds me of a joke I saw as a teenager, long before I even cared about something like a career choice. So with that caveat about engineers in place, let's kick off a series of

All Odd Integers are Prime

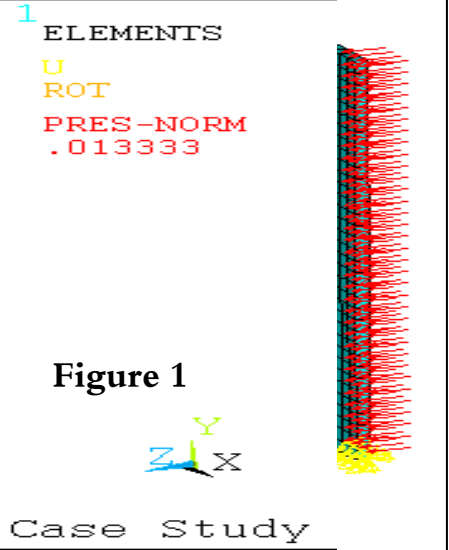
Mathematician: 3 is a prime, 5 is a prime, 7 is a prime... the result follows by induction.

Physicist: 3 is a prime, 5 is a prime, 7 is a prime, 9 experimental error, 11 is a prime, 13 is a prime...

Engineer: 3 is a prime, 5 is a prime, 7 is a prime, 9 is a prime, 11 is a prime...

articles addressing the subject. This first article will establish our test case and explore the results of implementing the shell-solid connection through manually created overlaid shell elements.

First to establish our test case, we will take a case shown in Figure 1 of a uniformly loaded cantilever.



Solid Elements

The theoretical stress maximum is 100. Using finely meshed solids, we see the resultant stress for high-order elements (mid-side nodes) is within a gnat's hair (Figure 2), and is 5% for the coarse (Cont.. on pg. 3)

Function Loading - F{x}

By Doug Oatis,

A relatively new addition to ANSYS Classic was the ability to define loads by functions, rather than tables. Gone were the days of carpal tunnel causing table creations in Excel, or if you were extremely brave the ANSYS Classic GUI. Just like tables, primary variables (x, y, z, time, temp, etc) could be used to create very complex loading conditions. Unlike tables, you could de-

fine multiple primary variable regimes governing the equations (e.g. for $0 < x < 1$; $y = f_1(x)$, for $1 < x < 10$; $y = f_2(x)$). Unfortunately, as far as my ANSYS exposure is concerned, function loading has been ignored. Well, I'm here to say that nobody puts baby, err...functions, in the corner.

Yeah, that's right, I just used a quote from *Dirty Dancing* in an article about ANSYS. I'm not proud of it either, but it's the only thing that seemed to fit. Also, I haven't written an article for some time, so my



ANSYS Movie reference ability is a little rusty.

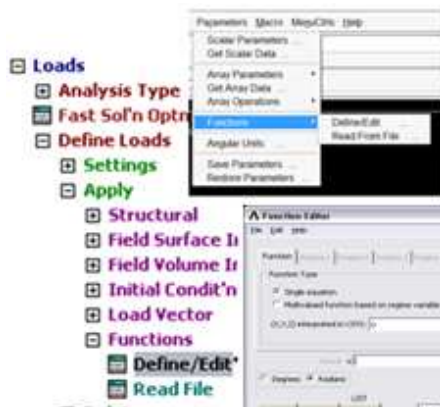
First of all, let's refresh our collective memory on function usage. You can define a function either through the /prep7 tree or 'Parameter' utility menu dropdown. This will (Cont. on pg. 2)

Table of Contents

Shell to Solid Interfaces-----	1
Function Loading - F{x} -----	1

(Function Loading, cont...)

bring up the 'Function Editor', which has a drop-down list of all the primary variables, calculator-type function creator (good for eliminat-



ing syntax mistakes), and up to 6 regime tabs.

After you build up your function you save it to a file (default extension is .func). At this point, you're 1/3rd of the way through. Next, you need to read that function in. When the function is read in, you need to specify a table parameter name and a coordinate system for x/y/z interpretation. The reason you need to specify this is because the function is read in as a "special" table. I say "special" because if you look at the table, it makes no sense to anyone but the solver. Finally, to use this function, simply apply a load where the magnitude is governed by a table, and specify the table you just created.

Well, it wouldn't be a Doug Oatis article unless a bad movie reference was used to discuss some Workbench Tip/Trick. I've obviously covered the movie reference, so here comes the WB tie-in. Let's say I want to apply a function load in Simulation. In v11, you can now do this, but only with time as your pri-

mary variable. If you want to use any of the other variables...you guessed it, command snippet!

To insert a function load in Simulation, simply copy/paste from the log file after reading in the function into a command snippet. It should look something like:



```
*DEL,_FNCNAME
*DEL,_FNCMTID
*DEL,_FNCCSYS
*SET,_FNCNAME,'d_test'
```

```
*SET,_FNCCSYS,11
```

```
*DIM,%_FNCNAME%,TABLE,6,12,1,
,,,_%FNCCSYS%
```

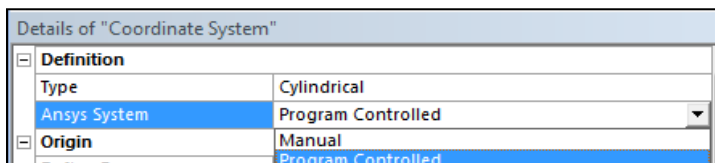
...Table Definition removed for article brevity...

```
!End of equation:
-1*[X]^3-2*[X]^2+10
```

You then need to insert the appropriate loading command and specify the magnitude as being the table (for the example above I used bf,all,hgen,%d_test%).

To really use the full capability of functions in Simulation, you'll need to do the following prepwork:

- 1 – Create named selections to define nodal/element components to load
- 2 – Create coordinate systems so



you can specify appropriate x/y/z interpretation

To use a user-defined coordinate system in your snippet, simply modify the line that says "*set,_FNCCSYS,XX" and replace XX with the appropriate coordinate system. In case you're curious, the

first user-defined coordinate system in Simulation gets passed as csys,12. All subsequent coordinate systems are incremented up by 1 (e.g. the fourth user defined csys = csys,15). You also have the ability to specify the "ANSYS System" in the 'Details' window for your coordinate system. I don't usually do this, I'd rather just follow the default rule listed previously.

Obviously, in order to make full use of this capability you need to have a decent grasp of APDL. You need to know how to select components ('cmsgl'), how to load elements/nodes, and how 'Named Selections' in Simulation are passed over as 'Components' in ANSYS Classic. However, once you get it working, the only things that need modifying in the command snippet are the table name ('_FNCNAME') and coordinate system ('_FNCCSYS'). I've been able to use functions to define a parabolic-shaped nodal displacement field and a radial-decaying heat generation rate to multiple cylinders (all in a single *do loop...had to get a little creative with that one).

After a little practice you'll be powering through hard-boundary conditions like James Dalton through Brad Wesley's cronies (hopefully with fewer explosions

and a little less death). Who knows, you might even become the local 'ANSYS Cooler'. But always remember "Rule 3: *Be Nice*". That's right, one article, two Swayze references! It's also been a personal goal of mine to somehow incorporate *Road House* into the Focus.

(Solid - Shell, cont...)

mesh used with the low-order elements (Figure 3). So, for the remainder of the cases herein we will use only high-order elements which ought to be quite accurate to capture simple bending given that the expected stress distribution is of the same order of the shape function (quadratic).

Shell Elements:

Using only shell elements (93's) we see similar accuracy in Figure 4: Note the length was extended for use later, but in that case the results are similar in Figure 5.

Shell-to Solid Geometry

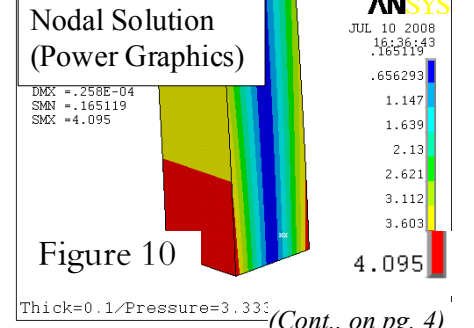
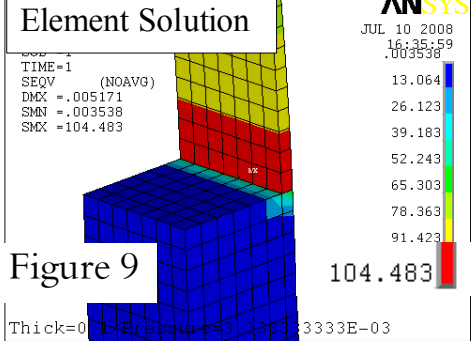
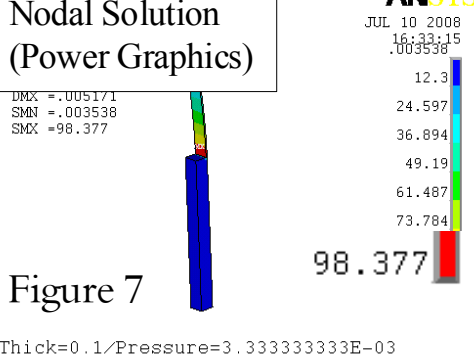
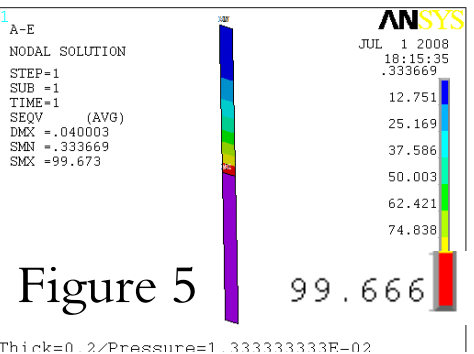
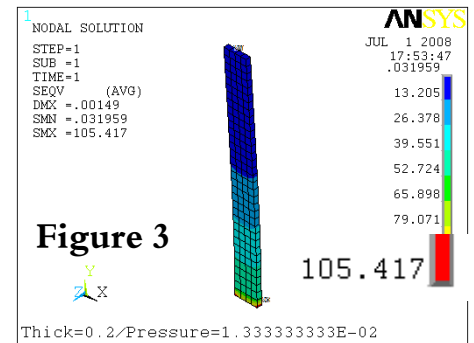
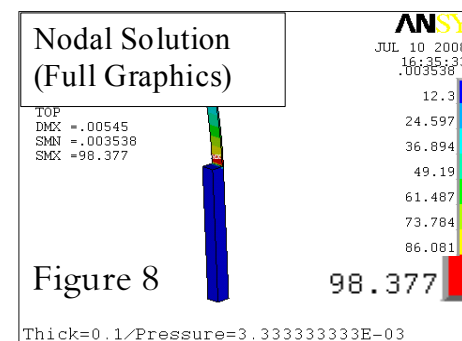
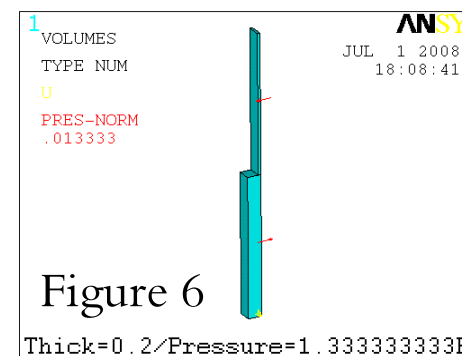
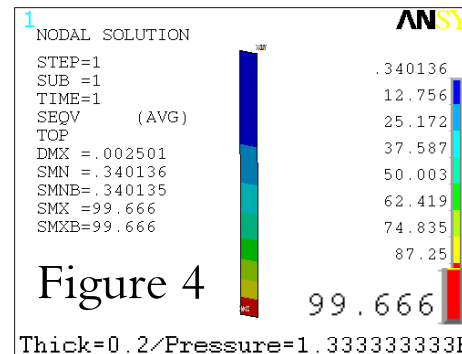
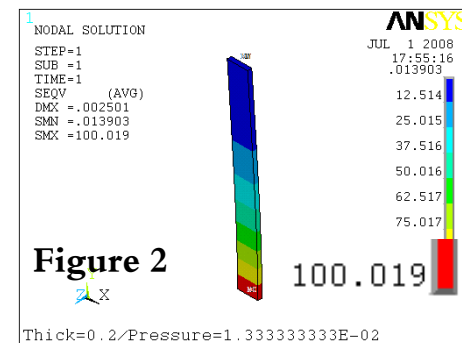
Because we are concerned with solid-shell connections, we are going to include a larger solid region as shown in Figure 6.

Solid Elements Only:

For this case, the nodal solution (PLNSOL,S,EQV) stresses are quite a bit lower than theoretical due to averaging of the nodes (Figure 7 and 8). In some cases an element solution (Figure 9) is more accurate and is often more appropriate at boundaries of differing materials, geometric singularities, etc. With refinement the nodal solution and element solution would approach the theoretical value of 100 at the base of the thin region and 4.00 at the base of the solid (Figure 10).

Solid Painted with Shell Elements:

Note that without some way to link the shell ROTX deflections, the shell is free to spin around the X-axis because the shared (or coupled) nodes do not have ROTXYZ DOF's on the solid element. In that case we have a hinge joint and no transfer of moment to the solid elements.



(Cont.. on pg. 4)

(Solid-Shell. cont...) degrees of freedom (ROTX, ROTY, ROTZ) at the interface nodes, and allows the moments from the beams to transfer into the solid as shown in Figure 11 and Figure 12.

With this “painting” method the element solution will give fairly accurate results as seen in Figure 13 and Figure 14. (The nodal solution will take great refinement to reach the same accuracy). The stresses in the solid are different for the two cases because the moment is transferred to the solid over different regions. We’ll discuss this more below.

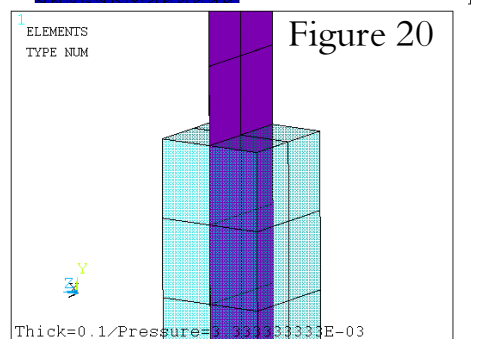
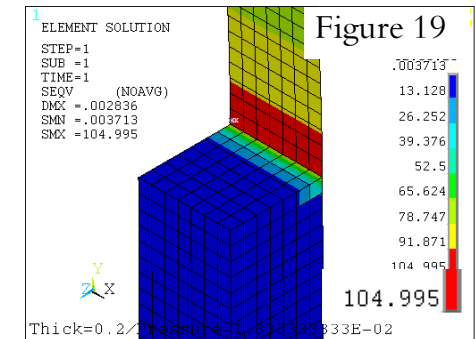
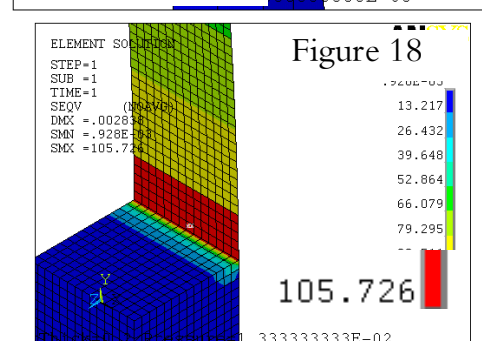
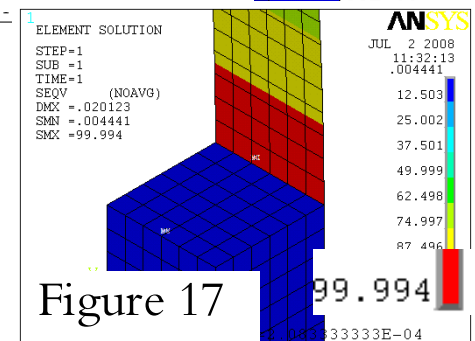
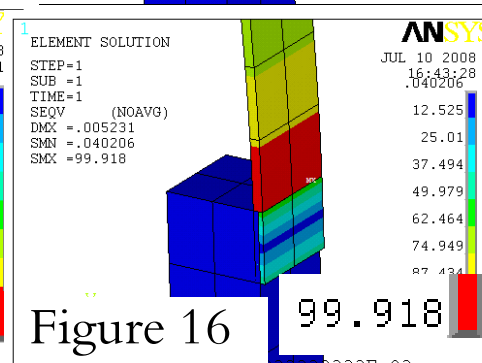
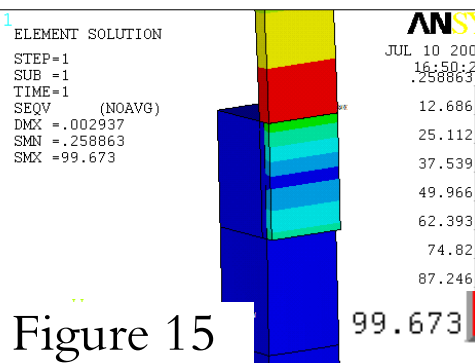
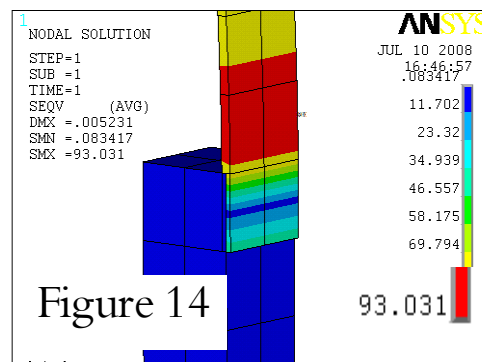
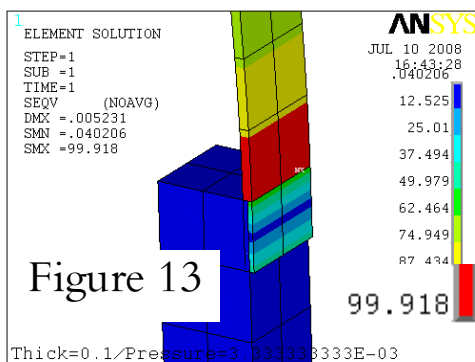
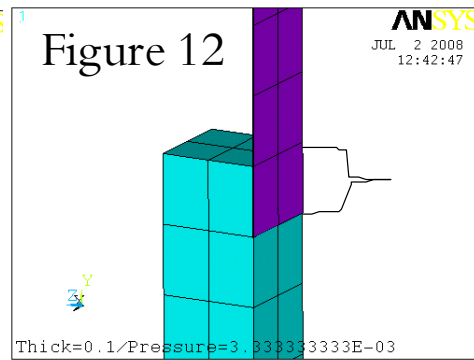
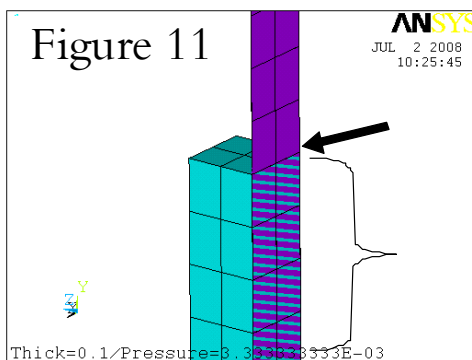
For thinner beams of a more shell-suitable nature the results (pressures appropriately reduced to allow for the thinner section) the element solution will also approach the theoretical value of 100 as demonstrated in Figures 15-17.

Don't Make this Mistake!

Often people use a fine mesh on the shell regions trying to increase accuracy! (Figures 18-19) This creates element aspect ratios outside the accuracy regime of the shell formulation. Keeping sizes closer to 10:1 for edge length:thickness will best support accuracy. Note that with high order elements, structural stresses will be very accurate even for coarse meshes as long as the shells represent the geometry (more may be needed to capture curvature of the actual part).

Mid-Plane Extraction Considerations:

A slightly different configuration imbeds the shells within the solid (Figure 20). Within the thin region, the results are identical as shown in Figures 21 - 22. (Cont.. on pg. 5)



(Solid - Shell, cont...)

Thus one can conclude that offsetting surfaces (and mid-plane extraction methods) are more accurate only from a geometric standpoint, but the resulting shape functions and solution are not changed.

Modeling the Physical Attachment:

We can see that the stresses within the thin member don't vary much given our methods of attachment. But what about stresses within the solid? They are changed for different scenarios as shown in Figure 23.

Even if we use a non-imbedded scenario, which likely is closer to reality for most bolted and butt or lap-type welds schemes, we must make the decision to paint the top, vertical, or both surfaces with shell elements. For each of these cases the stresses within the solid are impacted as shown in Figures 24-26. Note that the difference between the two methodologies transmits all the way to the base of the solid structure!!

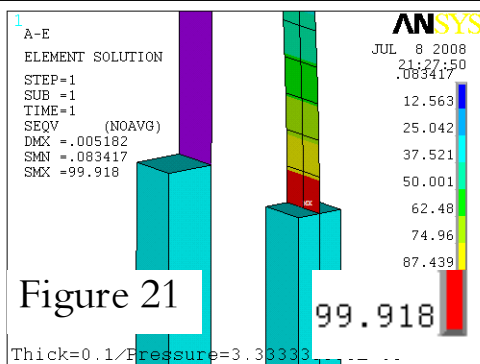


Figure 21

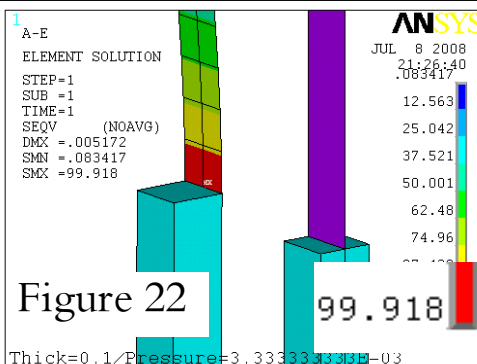


Figure 22

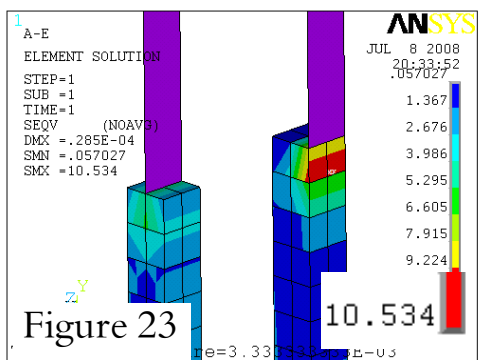


Figure 23

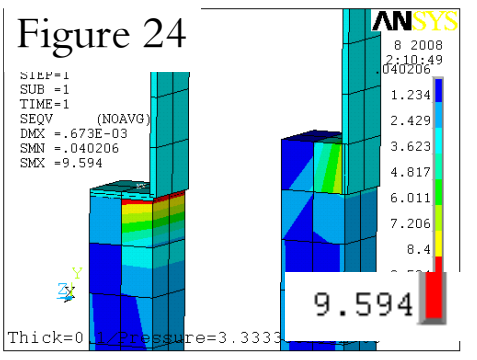


Figure 24

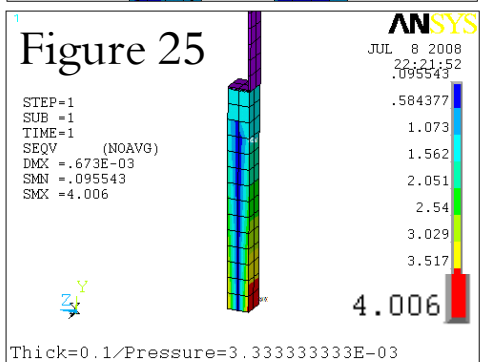


Figure 25

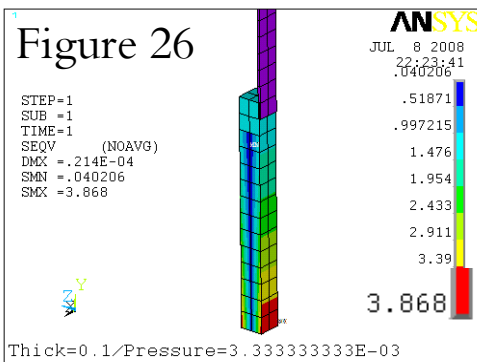


Figure 26

Recommendation for Bolted Connections:

If the connection is a bolted one, in this case too an overlap should be included. Also, one should check that the separation force and bending moment result in a less than zero force separating the thin member from the solid. Consider if a net force check is sufficient, or if a less than zero pressure condition is necessary over the entire face. Finally, to check for rotation about the bolt, check moment reaction, and verify that it is less than the resultant tensile load within the joint (after considering bolt preload) multiplied by a conservative estimate for friction.

Recommendation for Welded Connections:

Of course the above-detailed methods are meant to increase accuracy around the connection, but are not meant to capture stresses with a high degree of accuracy near the welded region. (This **should** be evident by the dramatic coarseness of mesh typical of shell elements.) The actual welding process implements residual stresses, has changed geometry and stress concentrations and is typically not included in FEA. That being said, we might as well have the moments and overlaps be as accurate as possible as they transfer

into the solid given that little extra effort is required to implement the most appropriate connection. (We might also cautiously consider the off-hand experience that properly welded joints are usually stronger than a model with no weld bead included.) We'll look at three cases.

(Cont.. on pg. 6)

1) Welded Vertical Overlap:

Covered above.

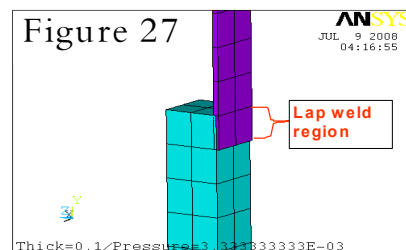
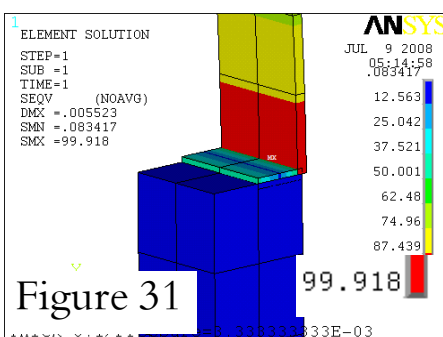
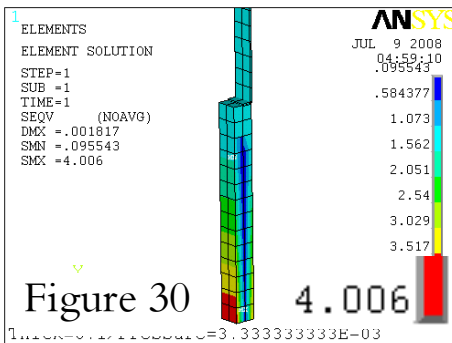
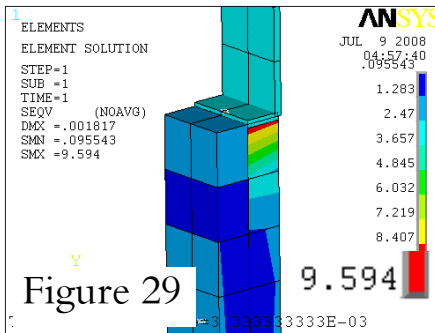
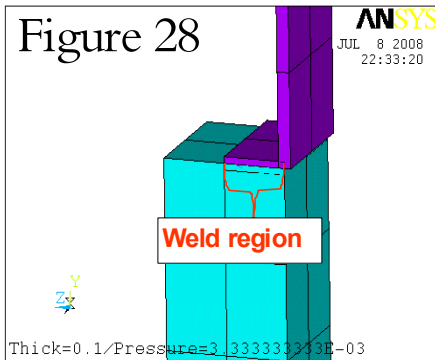


Figure 27

(Solid - Shell, cont...)

2) Angle Steel Butt Joint:

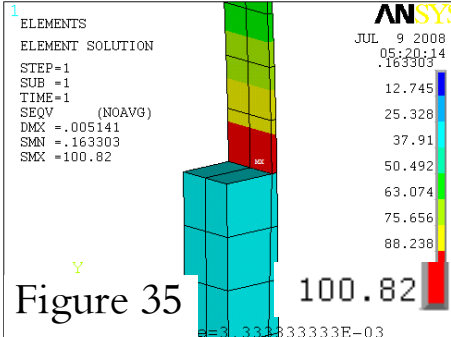
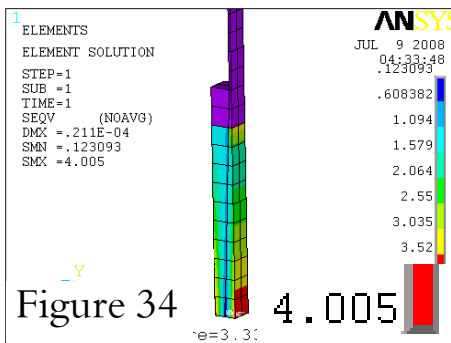
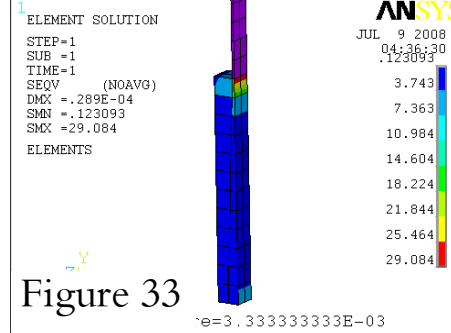
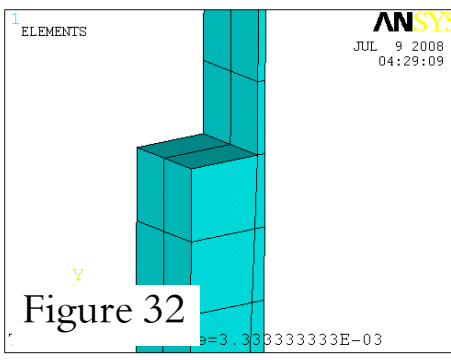
See the setup and results of the Angle Steel Butt Joint in Figures 28-31. We'll use these for comparison with the Butt Weld, up next.



3A) Butt Weld (Solid Elements):

Let's first look at the solid element version in Figures 32 – 35.

Here, if we compare to the Angle Steel Butt Joint results (left) we can see that for this case with the very



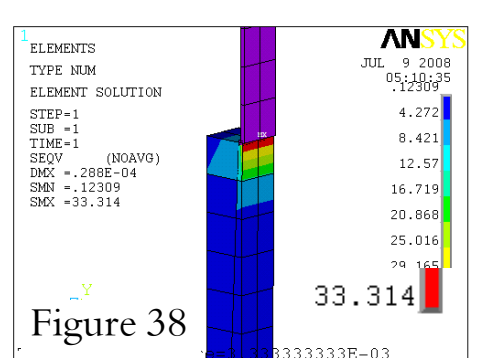
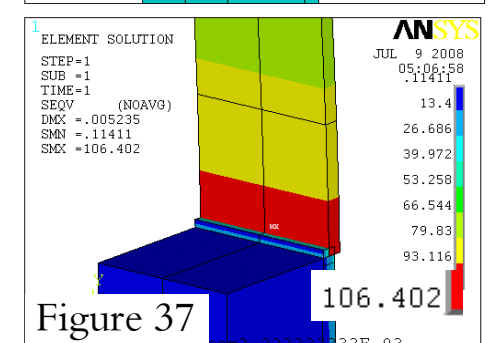
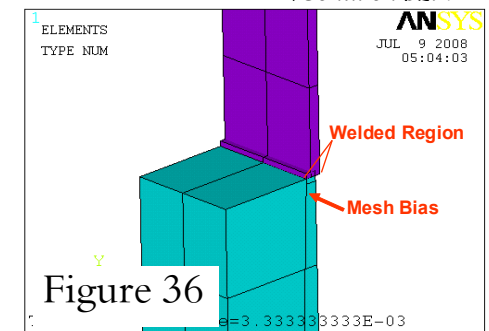
narrow butt-interface the local stresses in the solid are affected dramatically as expected. Yet the stresses at the base in the region removed from the weld location are still essentially unchanged.

3B) Butt Weld (Shell Elements):

Now we're getting somewhere. (I know, finally).

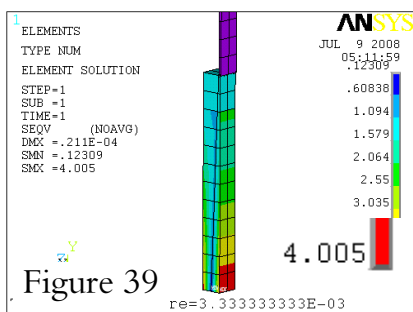
Let's look at the shell-to-solid version of the Butt Joint, (Figures 36-39), we find that the thin region's stress has 6% inaccuracy due to approximation of the short overlapping shells in the corner! This is quite the sacrifice after going through the pain of biasing the mesh to approximate the butt weld width!

(Cont.. on pg. 7)



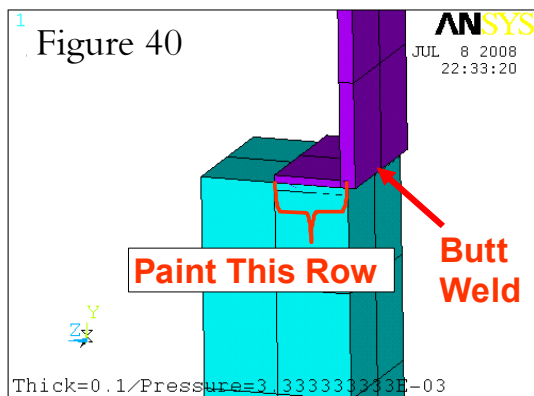
(Shell - Sol-

This is an important conclusion (albeit painfully earned): For shell-solid



joints, one can laboriously bias the mesh of a butt welded region so it resembles Figure 36, if increased accuracy in the solid's immediate region is desired (given great caveats about accuracy near welded joints). Yet, expect notable error in the shell elements.

If, however, accuracy in the thin region is higher priority, it is best to leave the mesh unchanged/unbiased. With this approach a suitable approximation at removed regions is obtained while not sacrificing the stress accuracy in the thin region near the joint! **Thus, often, all we have to do for a butt weld is to paint the first row of elements as shown below in Figure 40.**



ANSYS User Conference 2008

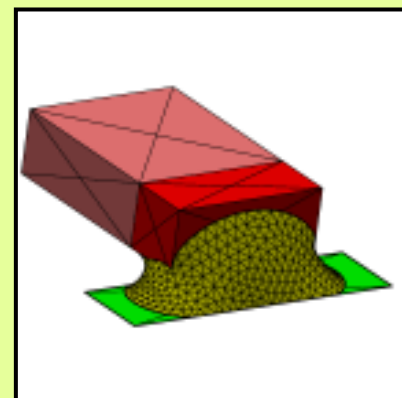
- Technical Sessions (of course)
- A second roundtable discussion featuring thought leaders in high-performance computing (HPC) will examine how significant advances in HPC create an opportunity for ANSYS customers...
- ...this panel discussion, themed "Inspiring Engineering with Simulation," is a chance for attendees to hear from -- and pose questions to -- a diverse group of managers from leading global organizations...
- and more....

Upcoming Training Classes

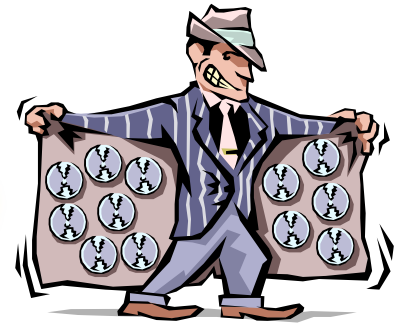
Month	Start	End	#	Title	Location
Aug '08	8/4	8/5	104	ANSYS WB Simulation - Intro	Tempe, AZ
	8/6	8/8	152	ICEM CFD?AI*Environment	Tempe, AZ
	8/11	8/12	202	Advanced Structural Nonlinearities	Tempe, AZ
	8/14	8/15	501	ANSYS / LS-DYNA	Tempe, AZ
	8/18	8/19	604	Introduction to CFX	Tempe, AZ
	8/28	8/29	701	Design Optimization and Probabilistic Design	Tempe, AZ
Sep '08	9/4	9/5	301	Heat Transfer	Tempe, AZ
	9/8	9/10	101	Intro. to ANSYS, Part I	Tempe, AZ
	9/11	9/12	102	Intro. to ANSYS, Part II	Tempe, AZ
	9/15	9/18	802	Advanced APDL and Customization	Las Vegas
	9/22	9/23	201	Basic Structural Non-Linearities	Tempe, AZ
	9/24	9/25	204	Advanced Contact and Fasteners	Tempe, AZ
Oct '08	10/2	10/3	104	ANSYS Workbench Simulation - Intro	Las Vegas
	10/6	10/7	205	ANSYS Workbench Simulation - Dynamics	Tempe, AZ
	10/9	10/10	100	Engineering with Finite Element Analysis	Tempe, AZ
	10/14	10/15	207	WB Simulation - Structural Non-Linearities	Tempe, AZ

Online Resources

- If you use Workbench, you might want to subscribe to the [ANSYS inc. Forum](#) for WB-specific discussion.
- Convert 3-D scans / images into models for analysis using [Simpleware](#) software.
- You like textbooks? After getting your head checked, try this [book written by Uof A professors](#).
- You probably already know this -- but [this is the new location / name](#) for what used to be Mil-HDBK-5
- Want to get the latest news on CAD/CAM/CAE companies? Get the latest news daily. Subscribe to [www.tenlinks.com](#) by filling out the "Subscribe to E-Mail" form.
- This is just plain cool... Okay, maybe not "cool" -- but dang neat. Its a tool for calculating the shape of liquids due to surface tension. Called [Surface Evolver](#). It is perfect for simulating solder balls and such. It is from the Susquehanna University and is free.



The Shameless Advertising Page



Need a guide?



**APDL Manual:
\$75**



Your Ad Here... Free!

Get Details.

**ANSYS Training from
Engineers for Engineers**

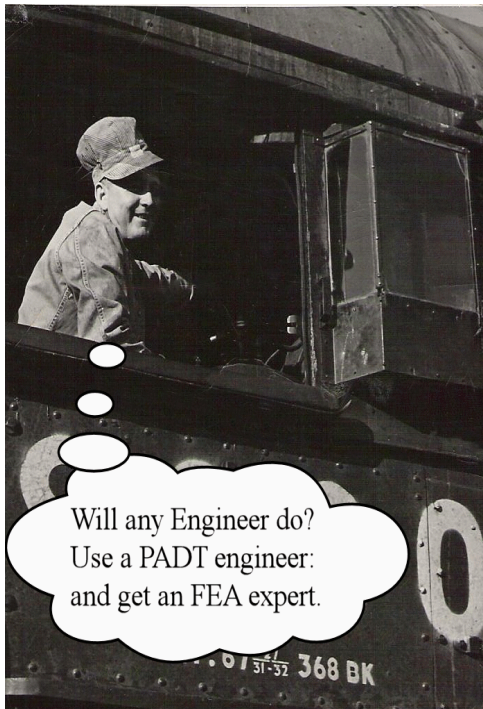


PADT

Faster Prototypes!



Ask about our turnaround time...



Will any Engineer do?
Use a PADT engineer:
and get an FEA expert.