Hoop Stress, Radial Stress, Axial Stress and Displacement Analysis on an Open Cylinder in SolidWorks

By Don Althouse
December 28, 2014

Historical development of the theory

The first theoretical analysis of the stress in cylinders was developed by the mid-19th century engineer William Fairbairn, assisted by his mathematical analyst Eaton Hodgkinson. Their first interest was in studying the design and failures of *steam boilers*. Early on Fairbairn realized that the hoop stress was twice the longitudinal stress, an important factor in the assembly of boiler shells from rolled sheets joined by riveting. Later work was applied to bridge building, and the invention of the box girder.

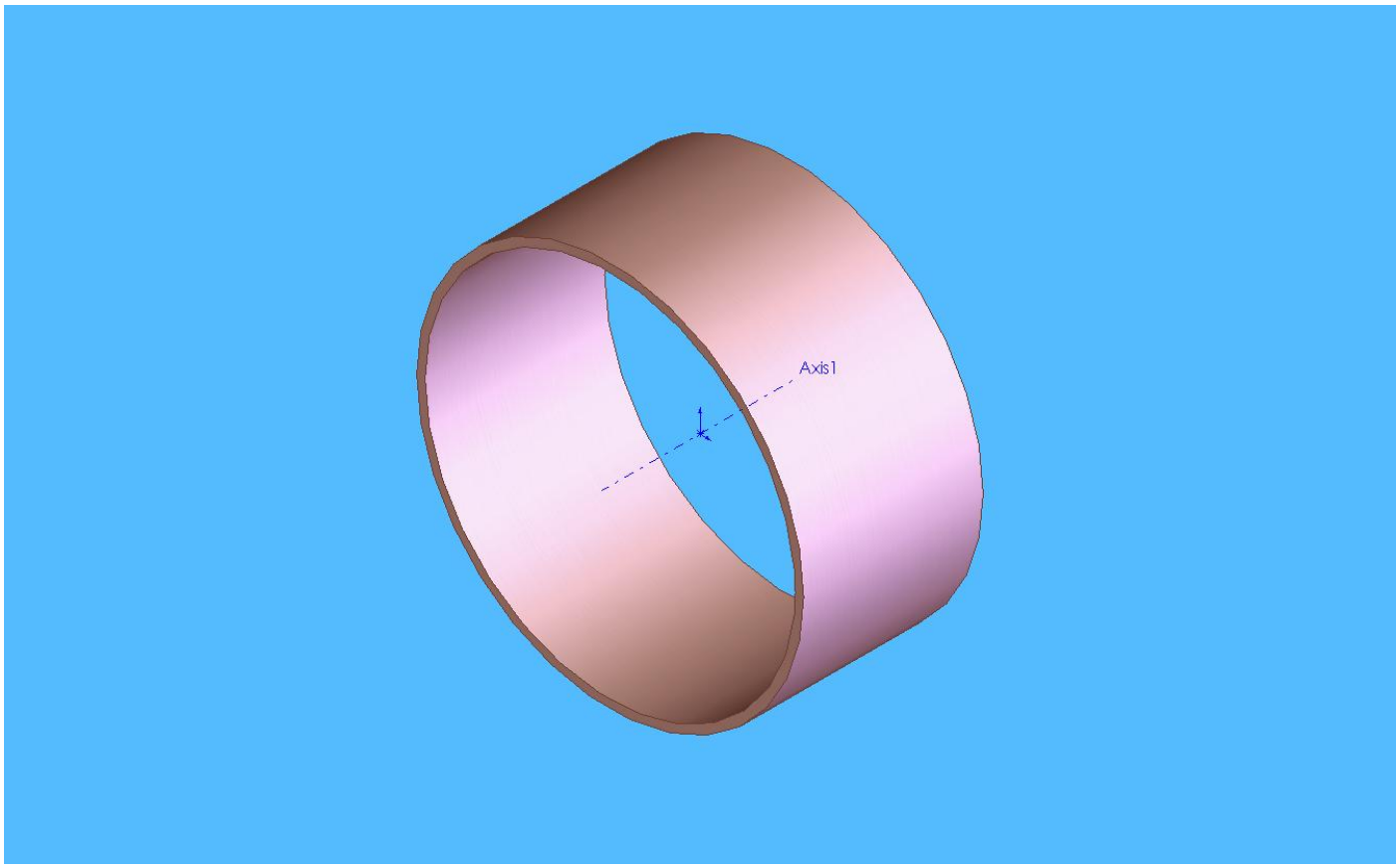


Figure 1: Cylinder for the stress analysis study.

The pioneers of yesterday have led the way for the developments of engineering and design technologies that are in use today; imagine the things they could have accomplished with the use of 21st century tools. Engineers and designers today are very fortunate in the fact that we have so many tools at our disposal and one of those tools is SolidWorks. Not only can it create magnificent 3 dimensional models, but it can also analyze these models to produce graphical representations of stress and displacement as well as other forms of engineering analysis tools it has like thermal and fluids just to name a few.

The cylinder shown in figure 1 represents our graphical stress and displacement study model that I shall use to get a baseline for what I can expect to see in the outer boiler wrap of LBSC's boiler. Its inner radius is 2.0625 inches with a wall thickness of .093 inches. The material is Copper and it has a length of 2 inches. Axis 1 represents the Z axis (which will be used as a reference axis for our cylindrical coordinate system) and the X and Y axis are of standard convention to that.

I shall be looking at the in plane stresses in the x-y plane as well as the out of plane stress in the z and displacement conditions in the normal or rectangular x-y and z planes and also creating a cylindrical coordinate system to view the results in as well as to make comparisons to. In addition to this I will look at the principal stresses in both coordinate systems. I will look at the solid element results and mid-surface shell element results for the Hoop, Radial and Axial Stresses.

There will be a lot of information in this initial study so as to give the reader a good understanding of some of the different ways to view results in SolidWorks and some of the classic stress equations used to check the analysis. Future studies will not go this far.

Thick or Thin Wall Vessel

Before analyzing the part in SolidWorks a good baseline of what we can expect to see as far as stresses go needs to be calculated from some classical stress analysis formulas for a thin walled pressure vessel. Since the dividing line between thin and thick walled pressure vessels is defined differently in various mechanics of materials texts we shall consider several targets for the definition. The first will be $t \leq r_i/10$ (often cited as $r_i/20$), (where r_i = the inner cylinder radius). If the wall thickness t falls in this range then the cylinder will be considered a thin wall vessel.

Equation 1
$r_i = 2.0625$
$t = .093$

$t \leq r_i/10$

$r_i/10$ would give 2.0625/10 = .20625 and $r_i/20$ would give 2.0625/20 = .103125

We can see that indeed $.093 \leq .20625$ and that $.093 \leq .103125$. In both instances, our value for t falls below the requirement set forth for a thin wall pressure vessel.
It may also be noted that some others define a thin wall pressure vessel by the expression $r_i/t \geq 10$, in this case our value would be:
Equation 2

2.0625/.093 = 22.17 and 22.17 is greater than 10.

There is one more equation that states if t/r_i is less than .1 we have a thin wall.
Equation 3

.093/2.0625 = .04509 and again we are well within the limits set for a thin wall.
This result along with the others ensures that indeed we do have a thin wall pressure vessel. Note that all three equations are just variations of each other.

State of Stress Definitions

1.) When the vessel wall is thin the stress distribution through out its thickness will not vary significantly, therefore we will assume that it is uniform or constant. A rule of thumb is that if the thickness is less than a tenth of the vessel radius then actual stresses will vary by less than 5%.

2.) This vessel will also be assumed to be at gauge pressure, since it measures the pressure above atmospheric, which is assumed to exist on the inside and the outside of the vessel's wall.

3.) The weight of the fluid, in this case steam shall also be deemed negligible and we will only deal with the pressure exerted on the inside wall of the vessel.

4.) We will assume the material is linear-elastic, isotropic and homogeneous.

5.) Radial Stress on the outside of the vessel is insignificant compared to tangential stress, thus, $\sigma_{ro} = 0$ and on the inside $\sigma_{ri} = -160$ psi which is the applied internal pressure.

6.) Longitudinal or Axial Stress, σ_L, (Out of Plane Stress)

- Exists for cylinders with capped ends; for open cylinders Axial Stress equals zero.
- Assumed to be uniformly distributed across wall thickness;
- This approximation for the longitudinal stress is only valid far away from the end-caps. i.e., Saint Venant's Principal

7.) No temperature change will occur: i.e. $\Delta_T = 0$

<u>Hoop Stress, classical equations from Mechanics of Materials theory</u>

Cylindrical pressure vessels are typically subjected to two types of stresses: **Hoop (Circumferential) Stress, σ_H** and **Axial (Longitudinal) Stress, σ_L** These stresses are derived by taking cuts through the cylinder and summing the forces that act along that cut, notating of course that due to equilibrium these forces must sum to zero.

For our hoop stress, the **Free Body Diagram** and summation of forces calculation are shown below in *equation 4*.

<u>Equation 4</u>

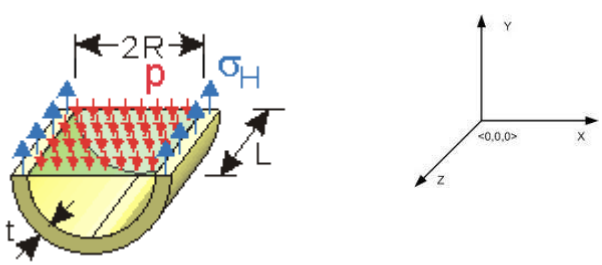


Figure 2: Hoop Stress **FBD**.

Summation of Forces:

$$\Sigma F_Y = 0 \rightarrow 2pRL = 2\sigma_H tL$$

This leads to the equation

$$\boxed{\sigma_H = pR/t}$$

<u>Where</u>
p = internal pressure
R = internal radius
t = thickness of material
L = length of the cylinder

Now that we have our classic mechanics of materials equation for the hoop stress all that is left to do is plug in the variables and solve the equation.

For the cylindrical part of the boiler wrap we have an inner radius of 2.0625 inches, the thickness of the material is .093 inches (13 gauge copper) and the pressure for the first initial test is LBSC's recommended 160 psi. We will later look at the SolidWorks results for the 80 psi running pressure.

<u>Equation 4:</u> $\sigma_H = (160 \text{ lb/in}^2)(2.0625 \text{ in}) / .093 \text{ in}$

$$\sigma_H = 3548.4 \text{ psi}$$

Now we have a good baseline of Hoop Stress to look for when we run our SolidWorks analysis for stress on the round part of the wrap. To make sure things are on the up and up I shall first run an analysis on the round cylinder I created with *no ends* shown in figure 1 and see what SolidWorks gives me for answers, then we can make a comparison of that analysis to the actual boiler wrap part later.

SolidWorks has criteria for doing analysis with sheet metal parts. Number 1 they recommend using shell elements, either mid-surface shells or surface shells. Number 2 they recommend having at least two elements per thickness of the material when using solid elements.

I thought it would be nice to see the actual distribution of stress through out the thickness of the cylinder so I am going to try and use solid elements first, a mid-surface shell will not show this distribution, it will only show the top and bottom surface of the shell and the membrane.

<u>Solid Elements</u>

At first I tried to mesh and analyze the entire cylinder with solid elements and since the wall thickness of my cylinder is .093 inches, I figured a mesh element size of .0465 would be fine to give me my two element requirement for the thickness. My old computer did not like that at all; there were over 766,000 Degrees of Freedom to compute. After waiting awhile for things to get done my results were not quite what I had expected to see in the cylindrical coordinate system I had created. Though we see what appears to be a somewhat uniform distribution of hoop stress with most of the colors going from orange to light blue from inside to outside, we also see patchy splotches of red with high stress concentrations, this is not uniform distribution. Memory issues I suspect are at the heart of this problem, it's time for some magic in the form of symmetry.

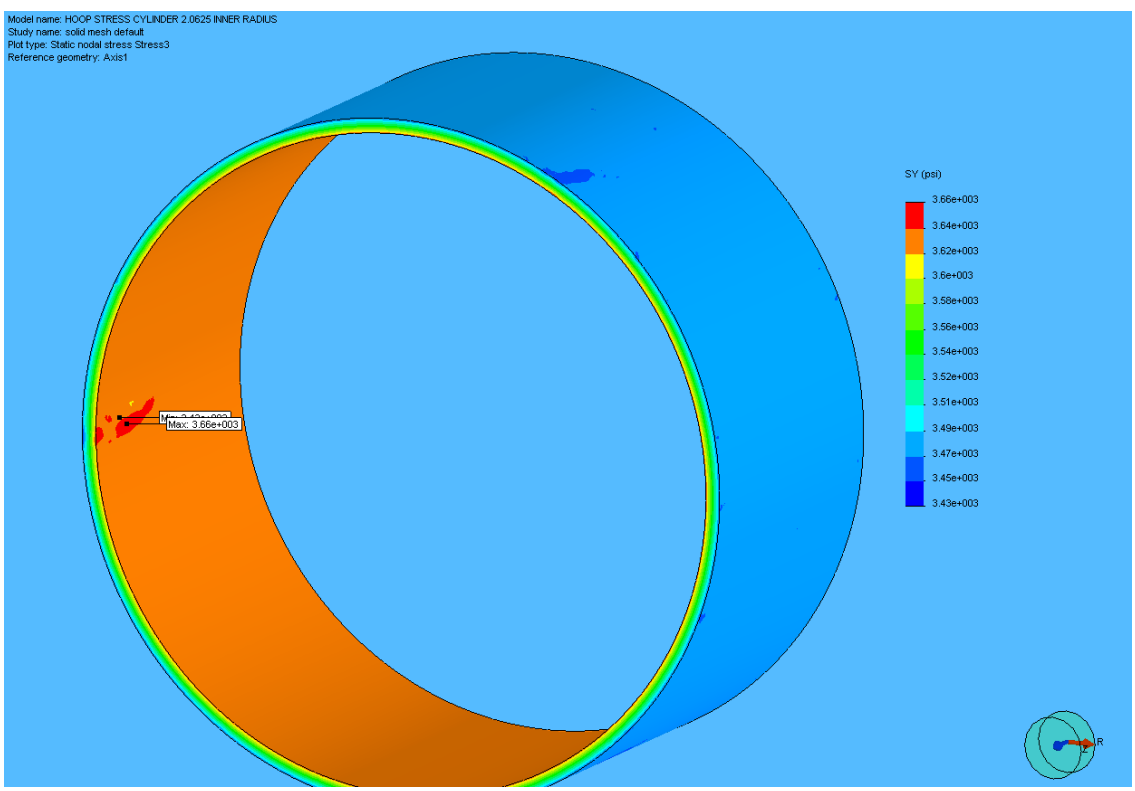


Figure 3: Solid mesh analysis with .0465 inch element size in a cylindrical coordinate system.

After seeing this somewhat expected result I made a 1/8 model of the cylinder, applied ***symmetry restraints*** to the cut edges, applied a ***Flat Face*** 0 translation restraint in the Z or normal to the direction on the back face of the cylinder. I have no choice here with the 0 translation restraint; I must prevent ridged body motion or the analysis will fail. Then I applied my 160 psi pressure to the inside surface.

With confidence I reduced my element size even more to .031 inches, this should give me 3 elements in the thickness now and a more accurate analysis with less error.

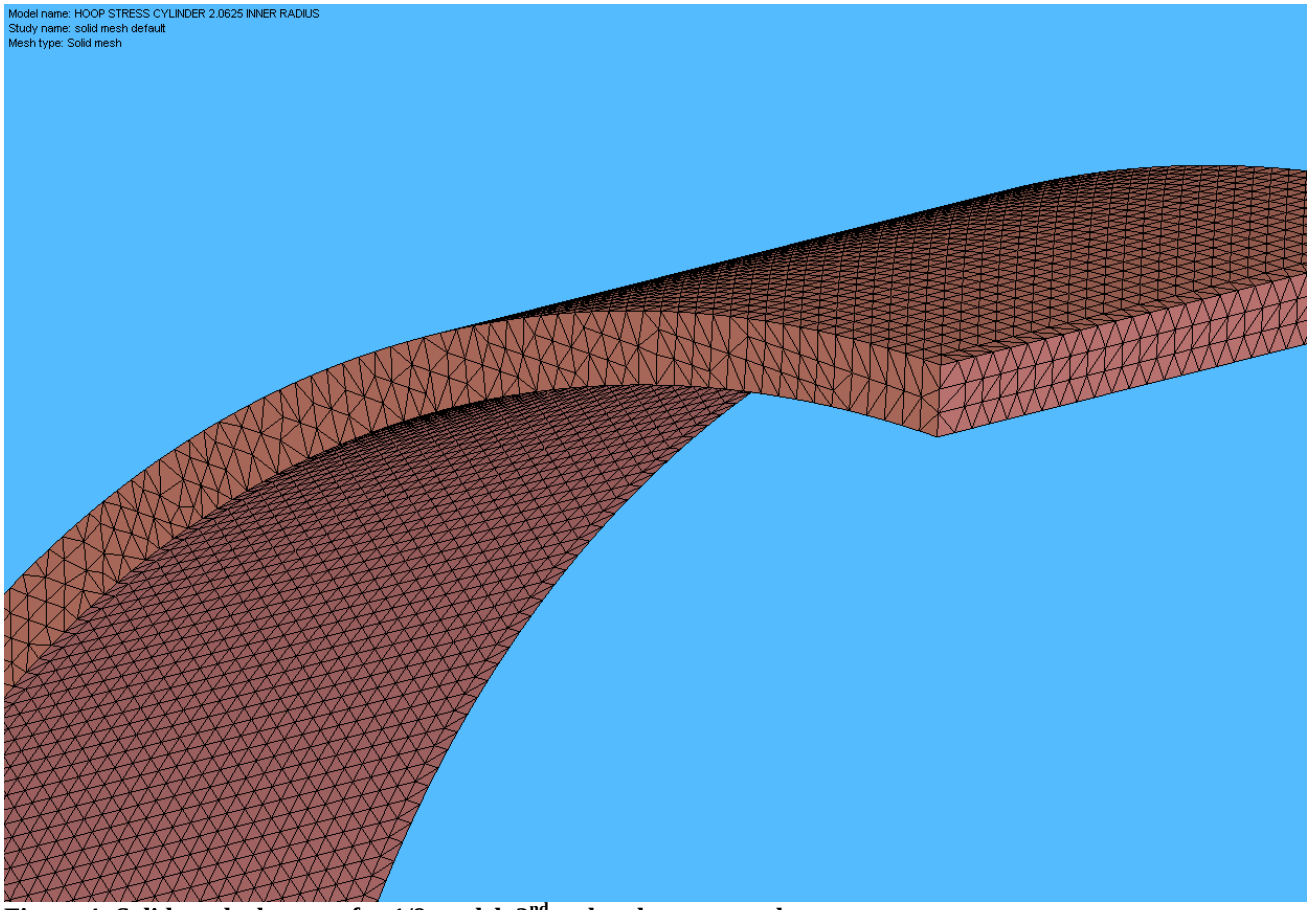


Figure 4: Solid mesh elements for 1/8 model, 2nd order elements used.

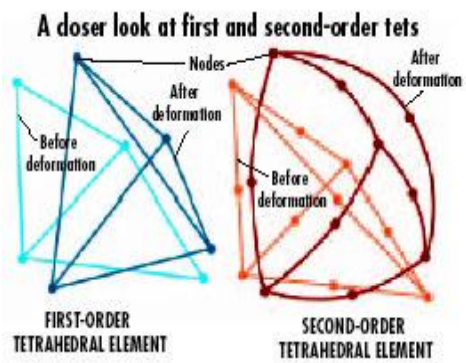


Figure 5; First and second order tetrahedral elements

Figure 5 above shows the tetrahedral elements that SolidWorks uses to map model geometry for analysis with COSMOSWorks. In first order elements, edges are straight and faces are flat. After deformation the edges and faces must retain these properties.

The edges of a second order element before deformation may be either straight or curvilinear, depending on how the element has been mapped to model the actual geometry. After deformation, edges of a 2nd order element may either assume a different curvilinear shape or acquire curvilinear shape if they were initially straight. Consequently, faces of a 2nd order element after deformation can be either flat or curved.

Each degree of freedom of a node in a finite element mesh constitutes an unknown. In structural analysis, nodal degrees of freedom represent displacement components, these are the primary unknowns. Structural analysis finds displacements first then strains and finally stresses, the strains and stresses are based on the nodal displacement results.

I am using 2nd order elements exclusively in all my analysis runs.

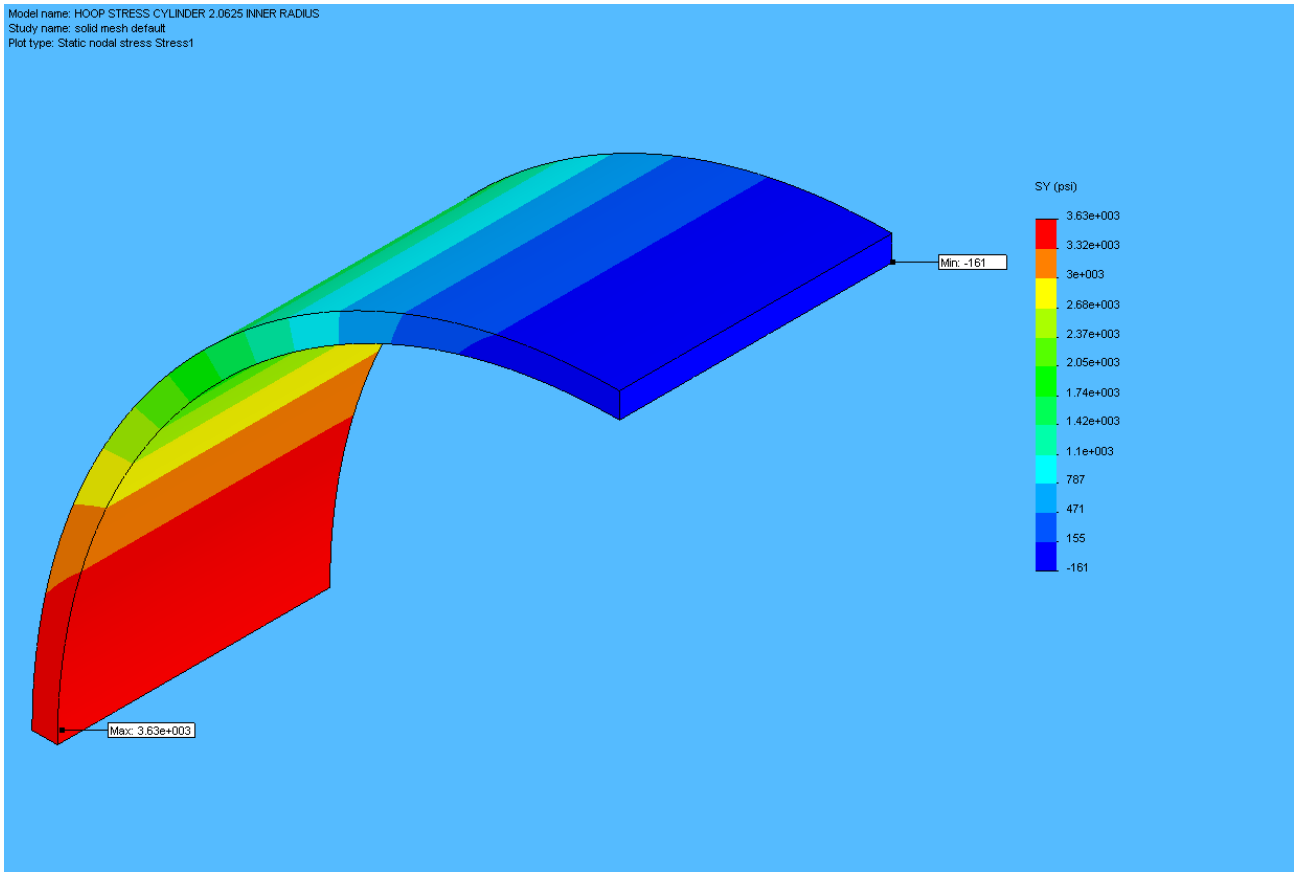


Figure 6: Plane Stress-Sigma Y in SolidWorks, rectangular coords.

Looking at the in plane stress plot (σ_Y) in figure 6, it is very hard to discern where the hoop stress is in the part and how the hoop stress behaves. The two max and min values you see in plot actually represent two totally different stresses.

The max value we see at the bottom left of the figure is for argument sake a value for the Hoop Stress at that point in the Y direction. The minimum value we see at the top of the part is actually a Radial Stress in the Y direction, notice the negative number indicating a compressive stress approximately equal to the inside applied pressure of 160psi.

Knowing where to look for certain stresses in SolidWorks is as invaluable as is setting forth certain assumptions and classic calculations for a comparison to the values that SolidWorks gives you. Recall equation 4 and in particular figure 2, since we know that the Hoop Stress lies in the cylinder wall and acts normal to face of the cut shown we should be able retrieve a similar value on the face of our model in SolidWorks. All I have to do is to rotate the part so that the bottom face of figure 5 comes into view as a front view and then I can probe for results on that face or get an average listed value for the face.

To make it easier to probe for results I have turned the mesh back on, this will enable me to count the number of elements across the face and get my middle point so as to better adhere to Saint Venant's principal which states;

Uniform stress predicted by classic equations exists only in regions reasonably well removed from points of load application, support locations, or locations of geometric discontinuity. (We have discontinuity at both ends of the cylinder)
External forces or internal reactions at these locations typically introduce significant localized effects. Therefore, in these regions, shortcomings of classical stress equations are at odds with the enhanced predictive capability of finite element results.

When nodes are probed with the probe tool, SW also includes coordinates of the probed point so one could also just probe look at the coords and go from there as well.

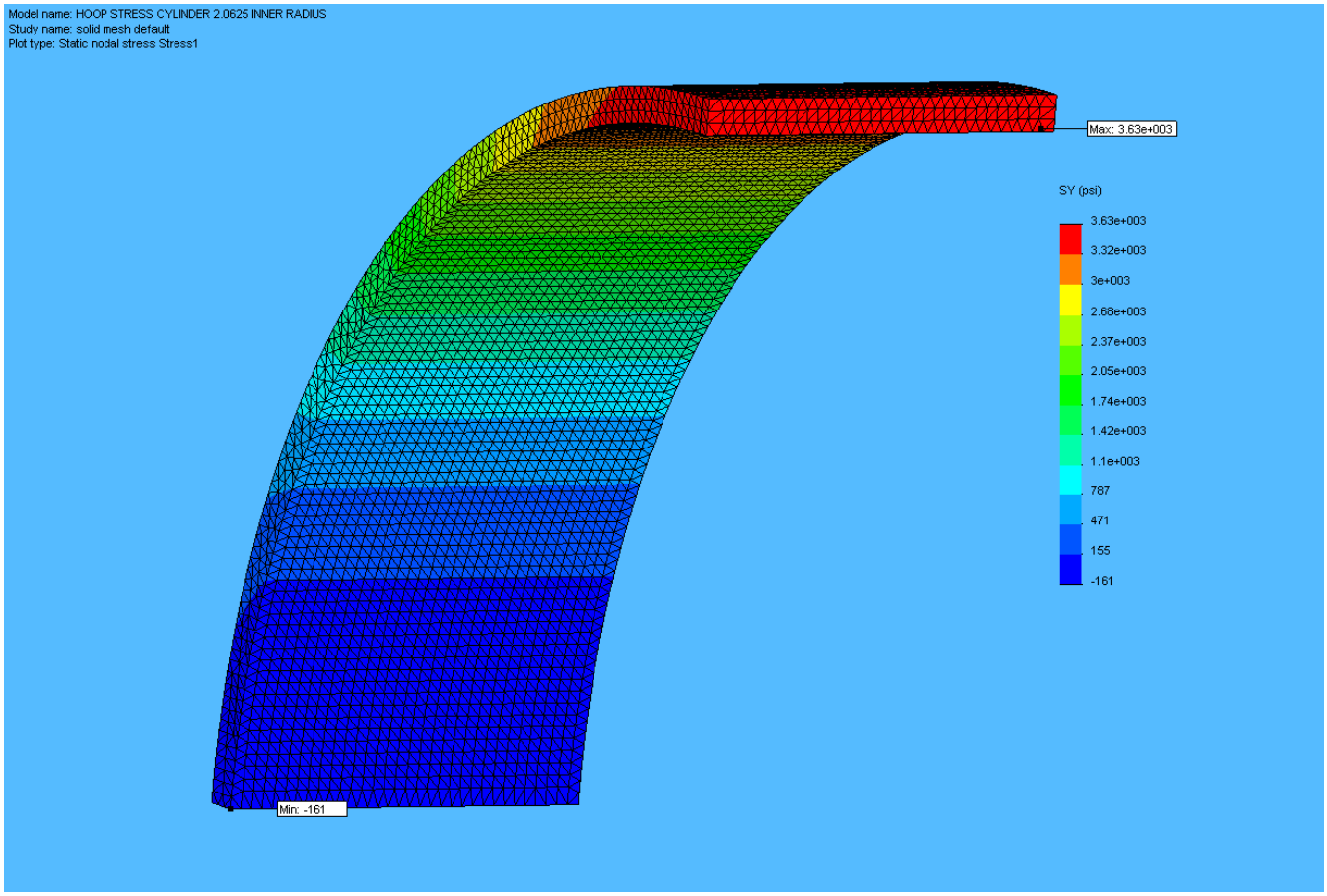


Figure 7: Mesh turned on for accurate probing and part rotated toward intended face.

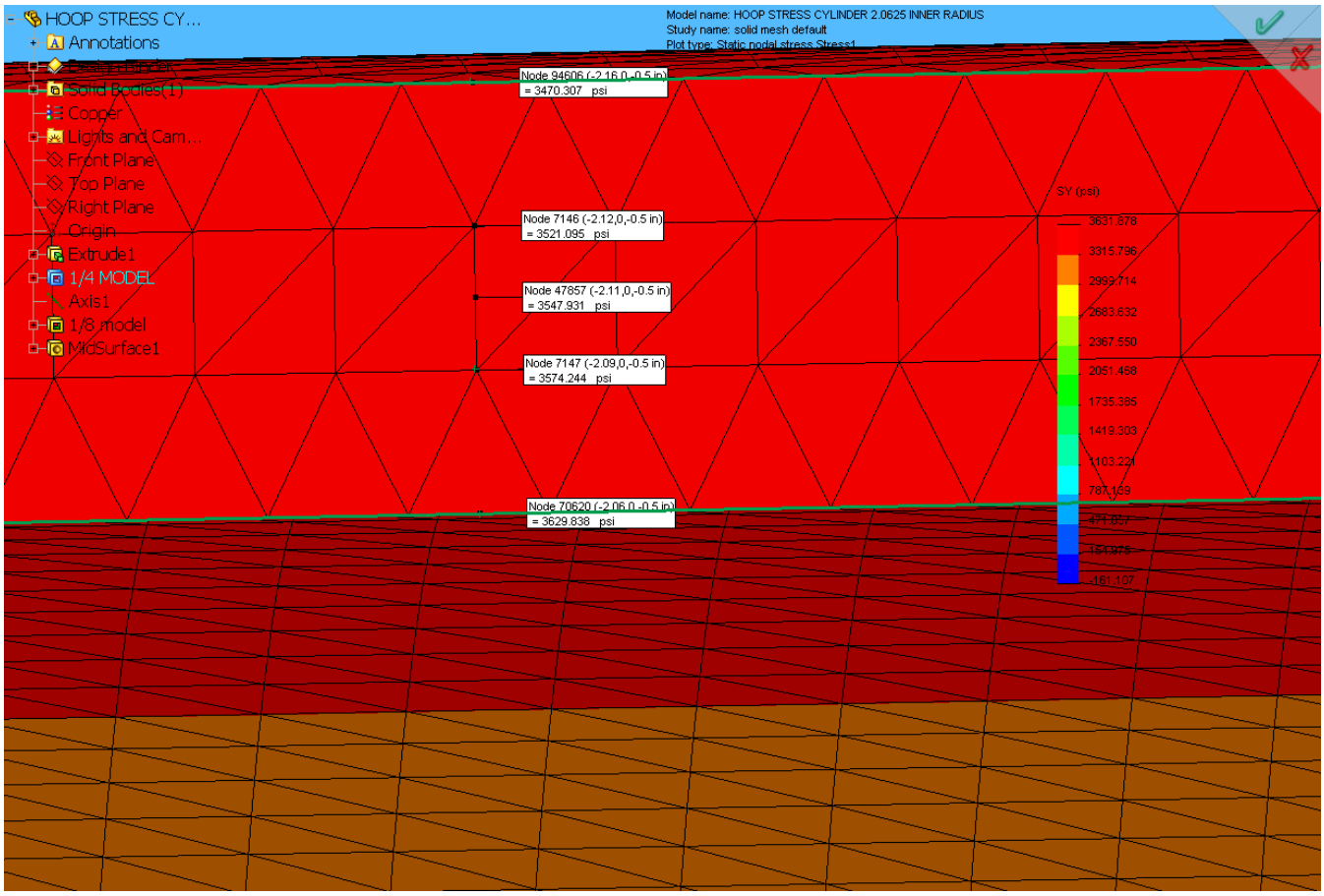


Figure 8: Probed results for the in plane Sigma Y stress thru the thickness of the material (rectangular coords).

From figure 8, we see at the very middle of the wall thickness on node # 47857 we get a value for the Hoop Stress at 3547.931 psi. That is a pretty good value considering our calculated value is 3548.4 psi. Now we can generate a percent difference or error difference between our classic and SW analyzed results.

Equation 5

% difference = [(FEA Result – Classical Result) / (FEA Result)] * 100

[(3547.931 – 3548.4) / (3547.931)] * 100 = .013218% or .013%

Hoop Stress Error (mid-point probe) = .013% Acceptable

We can also use a tool called selected entities to measure the average stress across the entire face. The selected entity tool calculates all the nodal values on the selected face then divides that number by the amount of nodes it got values for and gives an average.

The average value is 3548.8 psi.

Equation 5

% difference = [(FEA Result – Classical Result) / (FEA Result)] * 100

[(3548.8 – 3548.4) / (3548.8)] * 100 = .011271% or .011%

Hoop Stress Error (face average) = .011% Acceptable

As I stated in my assumptions at the beginning in order for us to assume a state of uniformly distributed stress through out the thickness of the wall I stated it should not vary more than 5% and as we see below it does not. Looking at the inner and outer values for the stress in figure 8, we get:

From *Equation 5*

% difference = [(FEA Result – Classical Result) / (FEA Result)] * 100

[(3629.838 – 3470.307) / (3629.838)] * 100 = 4.39498% or 4.39%

% variation of stress thru wall = 4.39% Acceptable

Pretty good results and with solid elements, now just for fun let's see what Sigma X gives us for the hoop stress; I would think it would just be directly opposite of what we saw before.

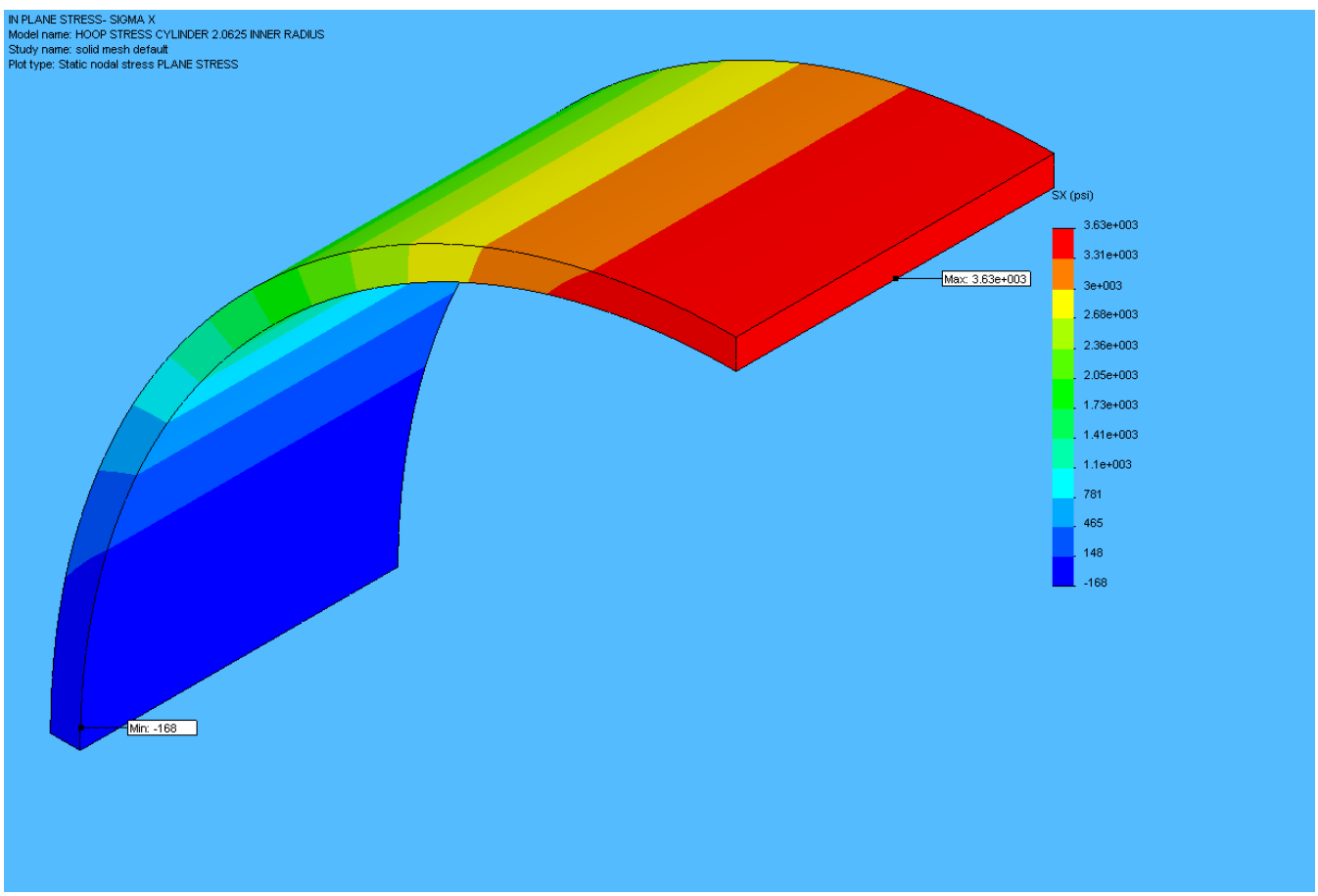


Figure 9: Plane Stress Sigma X in SolidWorks (rectangular coords).

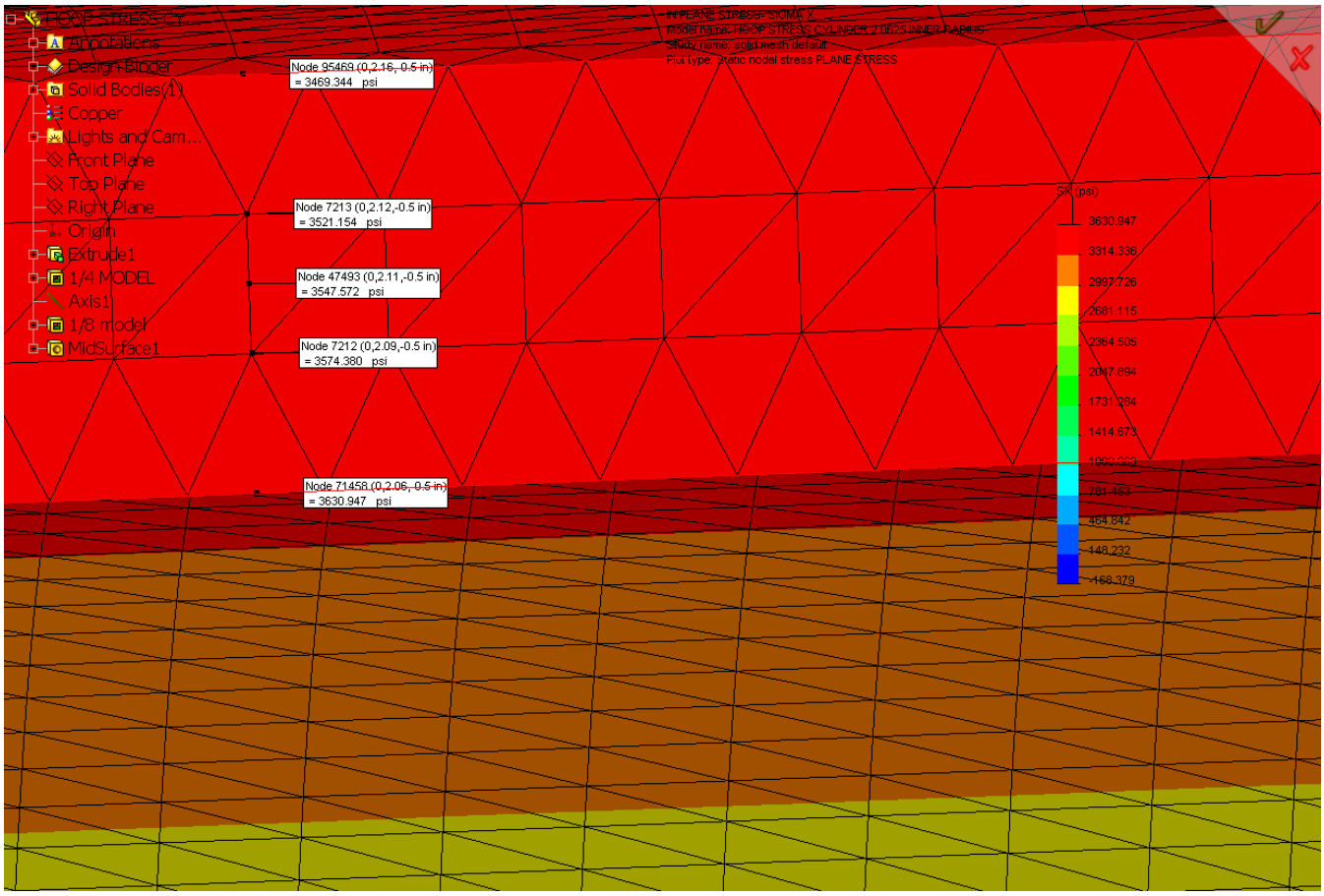


Figure 10: Probe results for the in plane Sigma X plane (rectangular coords).

Well sure enough, our hoop stress and radial stress are both in opposing places as figure 9 and 10 show, but notice how the radial stress values have increased a bit. Before it was -161psi and now it is -168 psi. I also checked the Hoop Stress again and although the list selected average value remained the same at 3548.8 psi, the individual probe results did not; they were actually a little bit lower than the ones before. Our mid point probe in figure 10 is 3547.572 psi where as the mid point probe in figure 8 was 3547.931psi.

I realize the differences are quite small but as I laid out before in my assumptions prior, this stress should be uniform through out the thickness of the material. I suspect an old computer, old program and memory issues could be the culprit here, or rounding errors in the program, but all in all they are good results and the wall variations in stress do not vary by more than 5%. Also note that for this in plane stress condition $\sigma_x = \sigma_y = \sigma_H$, which is our Hoop stress.

Now let's take a moment and look at the blue area at the top of the model from figure 6 where we have the -161 psi reading. As is clear in figure 11 we have a compressive stress at the inside of the cylinder that measures -159.827 psi and decreases to what should be a zero value (in this case we have -.235 psi) on the outside, this is a Radial Stress as the probed results show.

There are much better ways to view these stresses in SolidWorks as we will see momentarily.

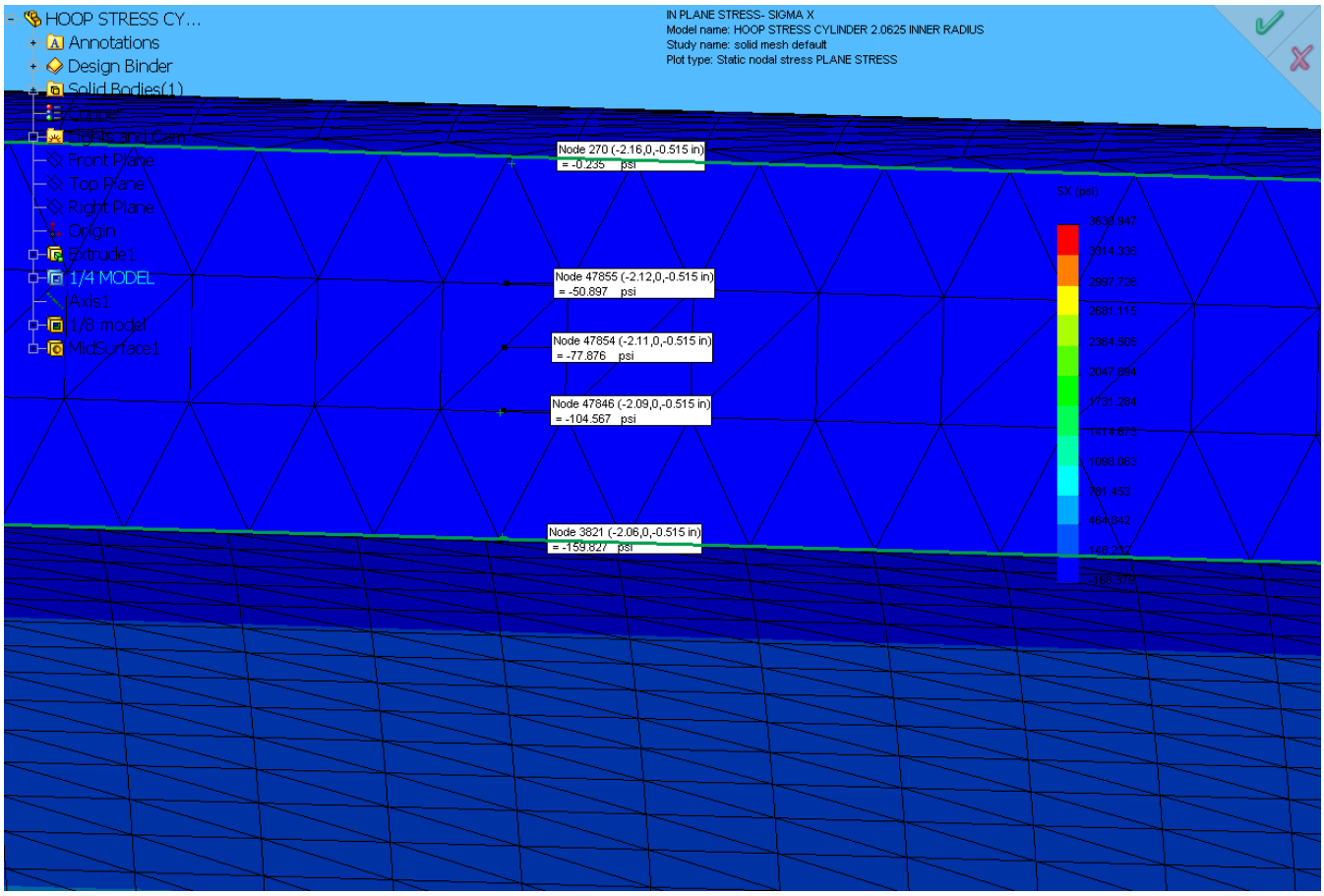


Figure 11: Probed results for the blue wall thickness of Figure 6, plane stress Sigma X shows radial stress.

Now let's look at the Hoop Stress in a cylindrical coordinate system and see what we find.

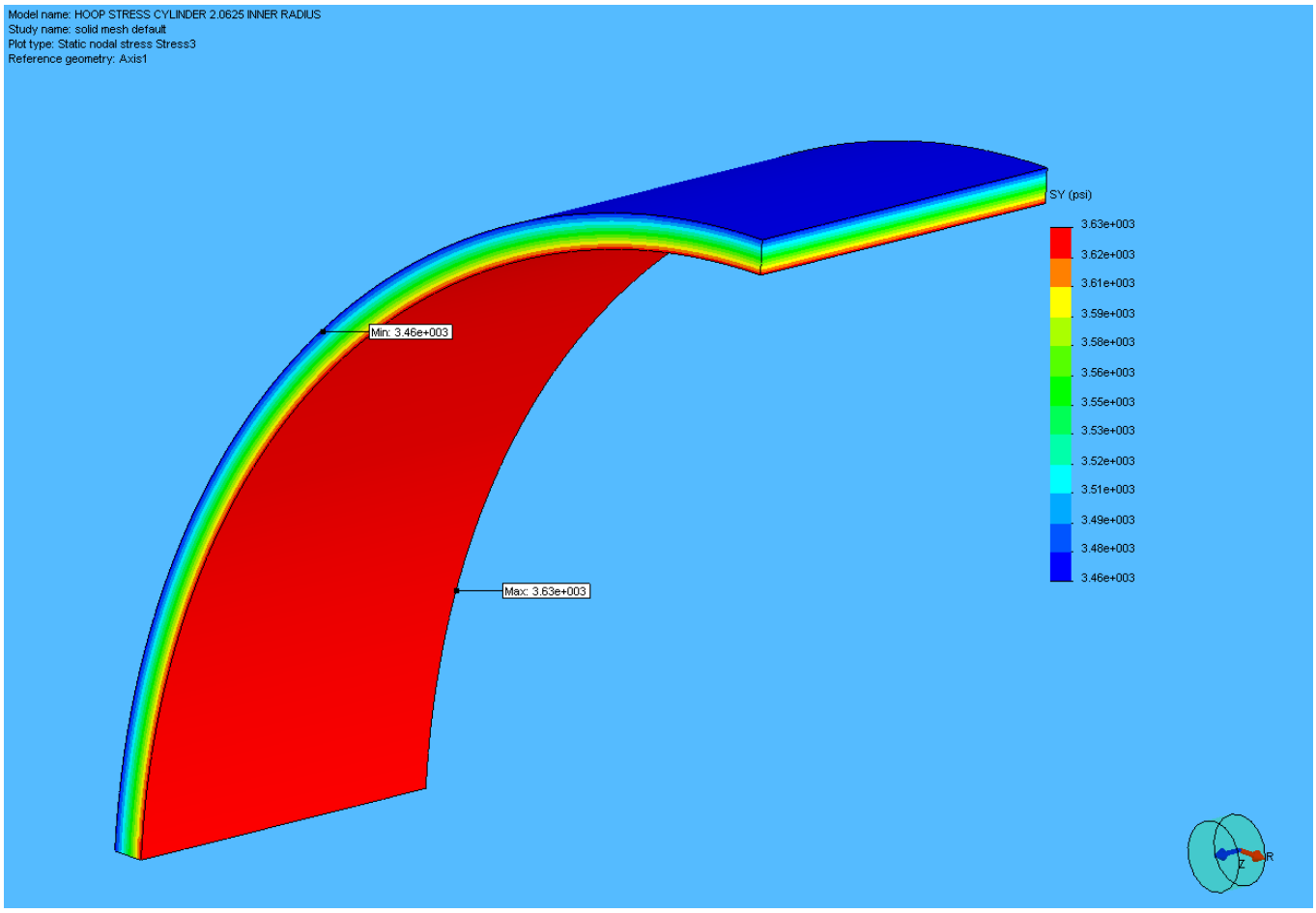


Figure 12: Hoop Stress distribution from inside to the outside of the cylinder with an asymmetric internal load of 160 psi in cylindrical coords.

Much better plot this time, we can clearly see the radiating hoop stress all in tension as shown by the positive values in figure 12. Note that the stress is plotted in cylindrical coordinates as shown by the light green cylinder at the bottom right of the screen and is in the SY plane which is the new circumferential direction for stress in a cylindrical coordinate system in SolidWorks with the Z axis as the reference axis for the cylinder. The max and min stresses you see on the plot are where the hoop stress is a maximum on the inside of the vessel and the minimum value on the outside of the vessel at those points.

To find the calculated hoop stress we will have to probe the thickness of the material again in the middle as we did before, but this time it should not matter what face we choose to probe, the stress should be the same, all Hoop Stress, not a combination of stresses like those that we had before. This is where the difference lies between a rectangular and cylindrical coordinate system.

To get a good probe location I shall turn the mesh back on and zoom into the top front face.

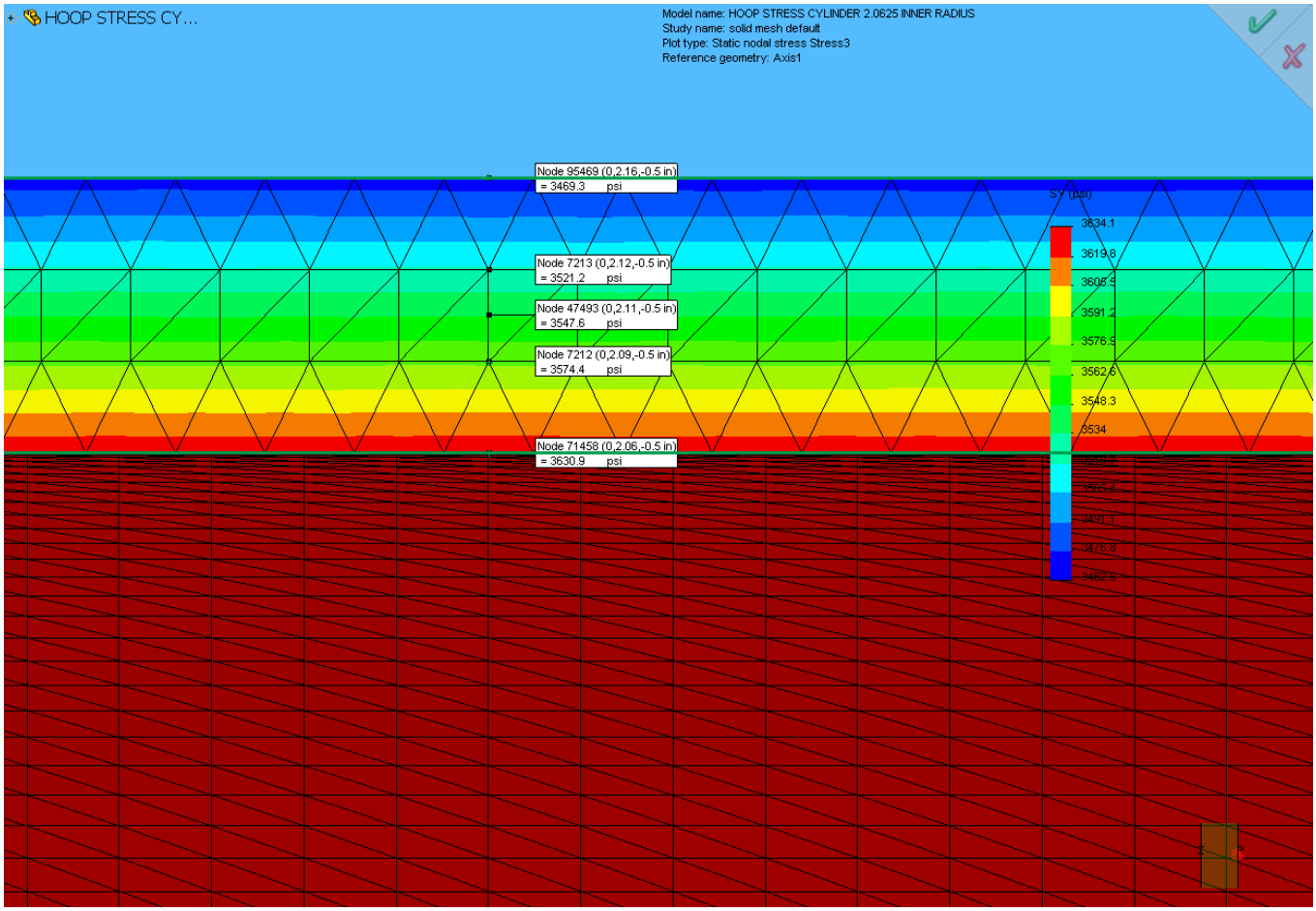


Figure 13: Probe results for hoop stress on the top middle face of the 1/8 model, first face or top face, cylindrical coords SY.

A check of the 5% variation of stress thru the thickness reveals:

[(3630.9 – 3469.3) / 3630.9] * 100 = 4.45068%

% variation of stress thru wall – top face = 4.451% Acceptable

<u>From Equation 5</u>

% difference = [(FEA Result – Classical Result) / (FEA Result)] * 100

[(3547.6 – 3548.4) / (3547.6)] * 100 = .02255% or .023%

Hoop Stress Error – top face (mid-point probe) $_{\text{cyl coords}}$ = .023% Acceptable

As a matter of curiosity, since we saw we had a variation in hoop stress as well as the radial in rectangular coordinates, I thought it best to check the other face of the model and verify that the hoop stress is the same.

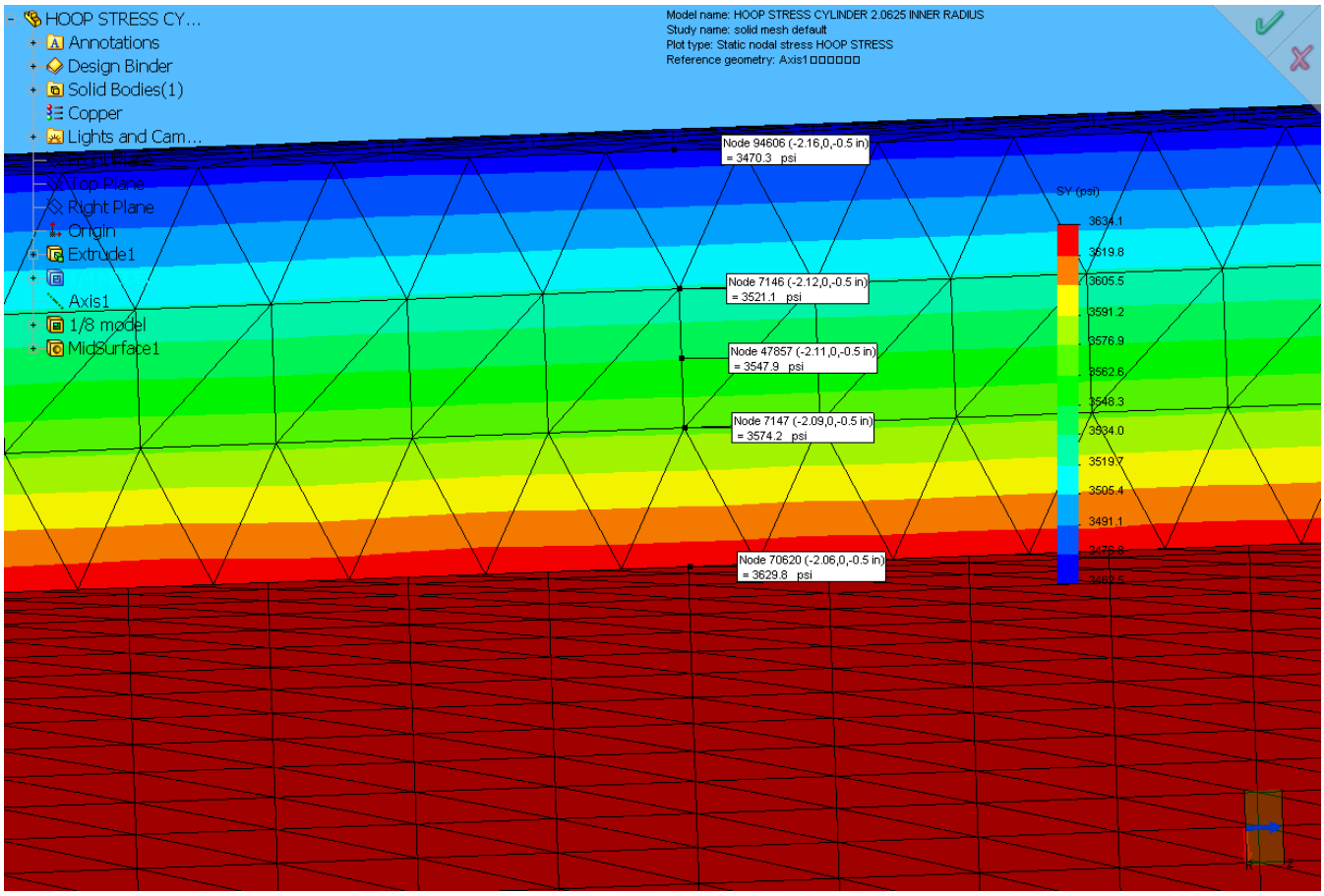


Figure 14: Hoop Stress probe, second face or bottom face in cylindrical coords.

Well this is interesting; the values are slightly off between the two probed faces. The mid-point value for the first face is 3547.6 psi and the value for the second face mid-point is 3547.9 psi. I would suspect rounding errors and an old computer for this discrepancy. Though I will not go this far on every analysis I will err on the side of caution for this one and take the higher stress value for the error calculation.

Our calculated value for the hoop stress in the cylinder was 3548.4 psi and we got a probe result of 3547.9-psi right in the middle. Now we can look at the percent error in our cylindrical result.

<u>Equation 5</u>

% difference = [(FEA Result – Classical Result) / (FEA Result)] * 100

[(3547.9 – 3548.4) / (3547.9)] * 100 = .01409% or .0141%

Hoop Stress Error – bottom face (mid-point) $_{\text{cylindrical coords}}$ = .014% Acceptable

At this point in time I can not for the life of me figure out how to capture the list selected data and put it into this report, so you will just have to take my word that the average listed stress for the face was calculated to be 3548.8 psi, the max was 3630.9 psi, the min was 3469.3 psi and the RMS mean was 3549.2 psi.

Looking at the average listed value for the Hoop Stress Error now, we will get;

[(3548.8 – 3548.4) / (3548.8)] * 100 = .0127% or .011%

Hoop Stress Error – bottom face (face-average) $_{\text{cylindrical coords}}$ = .011% Acceptable

I am very impressed with all those result errors, considering they are solid elements, which are not the recommended element to use when analyzing sheet metal components. I think a lot can be said about symmetry for this part in the analysis. It drastically cut down my computing time and memory usage.

A quick look at both results

	Cylindrical Coordinates	Rectangular coordinates
	psi	psi
Upper face stress @MP	3547.6	3547.572
Lower face stress @ MP	3547.9	3547.931
Upper face avg	3548.8	3548.8
Lower face avg	3548.8	3548.8

MP = Mid-Point

The reason there is a slight variation of the errors between the two studies is because in the rectangular coordinate system I had my decimal points set to a higher degree than I did in my cylindrical coordinate system. If it were not for that all the errors and stresses would be exactly the same. So inadvertently I have shown you how rounding errors come about, I shall endeavor to do a better job in the future.

Now let us look at the recommended mid-surface shell elements and see how they fair in our hoop analysis.

Mid-Surface Shell Elements

A mid-surface shell is just that, a mid-surface between the thickness of the material and in our case the mid-surface would lie at .093/2 = .0465 inches from the inner radius of the material. The radius from the axis to the mid-surface would equal 2.0625(our inner radius of the cylinder) + .0465 = 2.109 (see figure 15 & 16).

Shell elements appear two dimensional with straight sides, a flat surface and one node at each of its three corners and it also has mid side nodes when the second order shell is used. Shell elements actually have more degree's of freedom than do solid elements. In addition to the three displacements x, y and z they also have three rotations about each of those axis. My shell element size for this run is what I used for the solid elements at .031 inches.

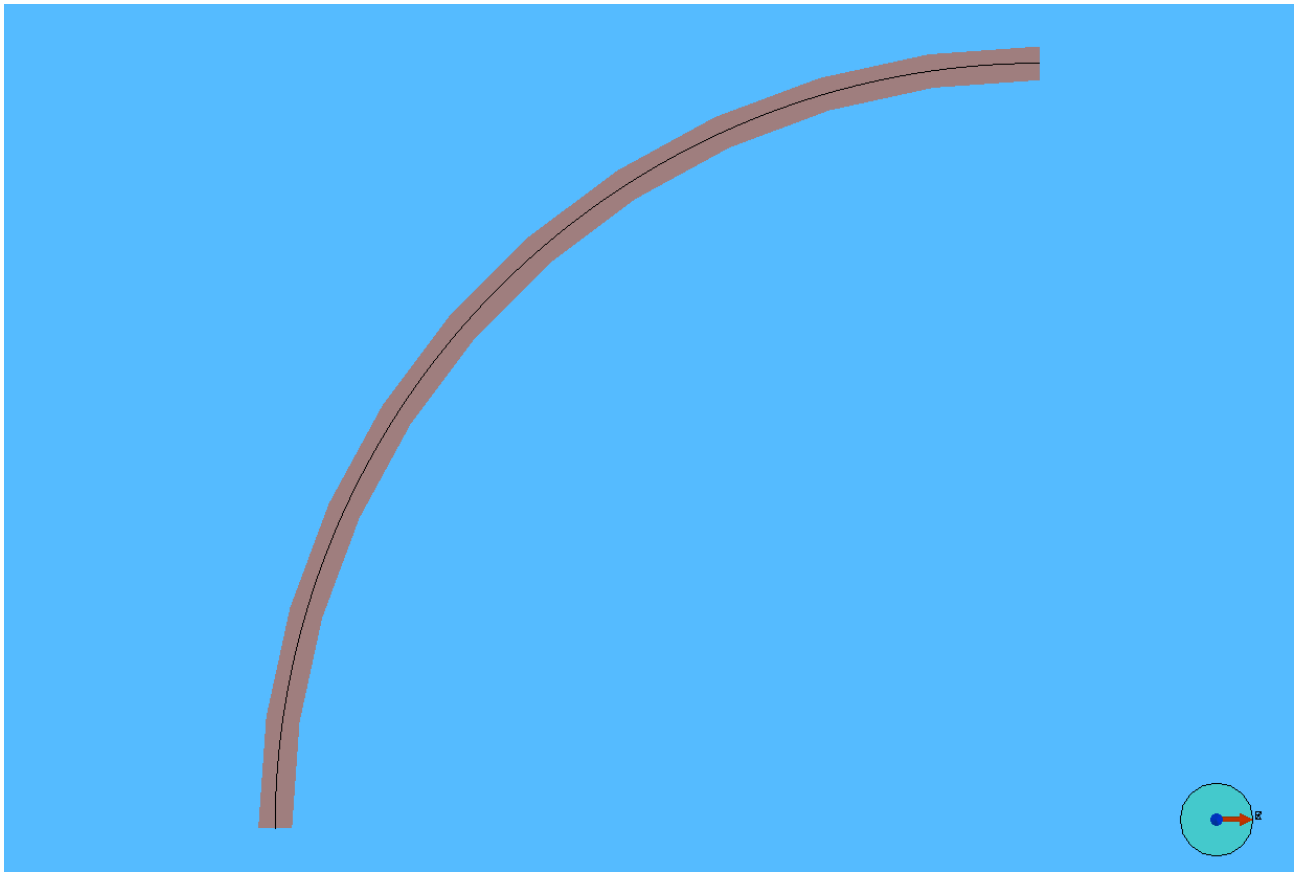


Figure 15: Line thru the middle represents the Mid- Surface shell with a radius value of 2.109 inches.

One of the things I will not be able to check with a shell element is the 5% variation of stress thru the wall thickness since a mid surface shell has no depth.

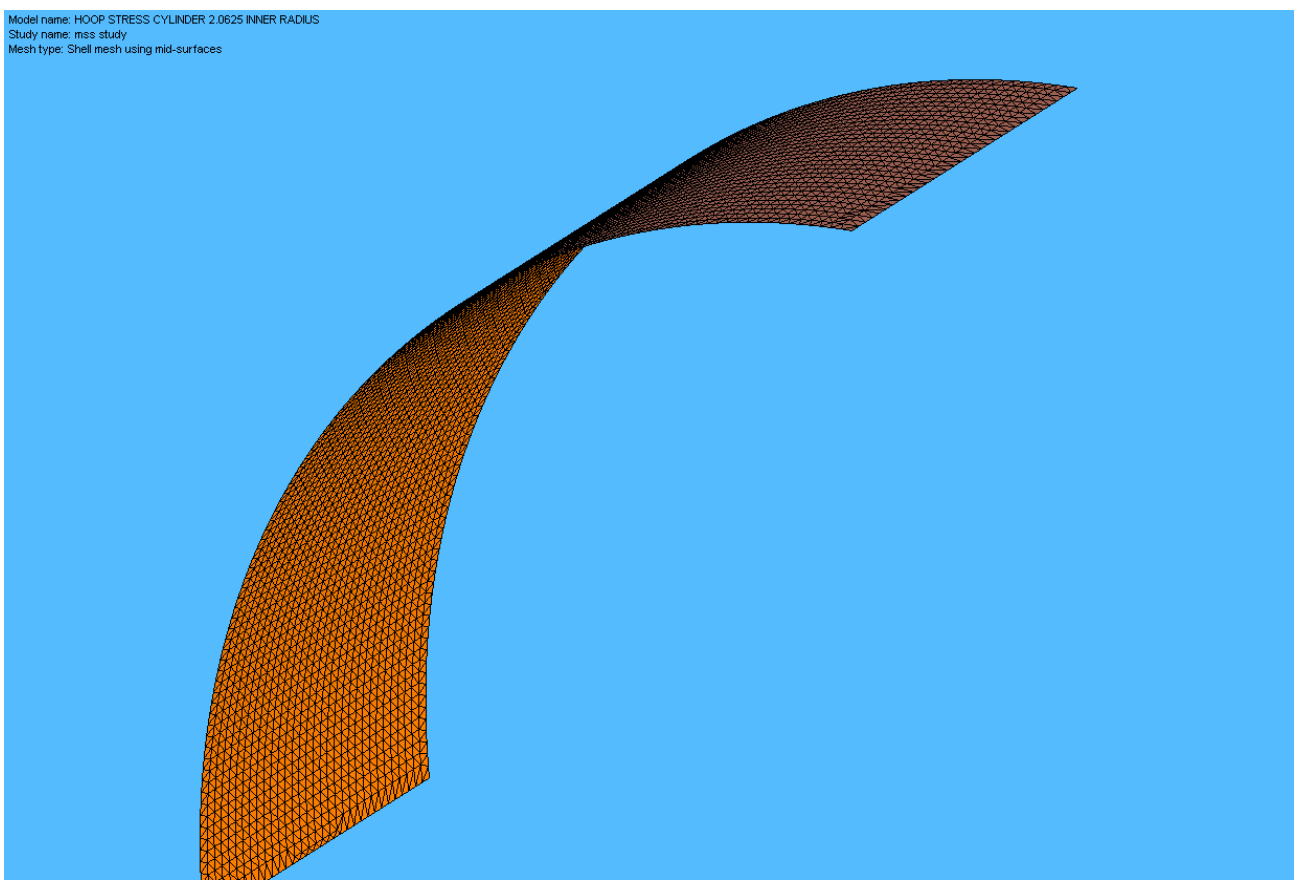


Figure 16: Bottom designation of shell element by orange color.

Shell elements have default color designations for the top and bottom surface of the shell (figure 16). The default for the bottom is usually orange and the bottom of the shell is usually where the load is applied. In our case the load or pressure is applied to the inside of the cylinder thus the orange color of the shell on the inside and the brown color on the outside. Note; when I first created the mesh the orange color was on the outside and I had to use the flip shell element command located as a toggle in the mesh folder to bring it to the inside.

Figure 17 shows the shell element analysis for plane stress in the Y direction in rectangular coordinates. Again we have basically the same color distribution as we did before back in figure 6 for the solid elements and again it shows two different stresses. This time however we get some substantially different numbers. For the hoop stress we have 3638.178 psi and for the radial stress we are at .049 psi.

SolidWorks gives the values for shell element stresses a little differently than it does for solid elements because of the addition of the three other axes for rotation as I noted before, in addition to this you can get values for the top or bottom of the shell as well as just the membrane values. The membrane values are calculated with the meridian radius value of 2.109 inches as opposed to the 2.0625 inch inner radius value for the solid elements calculations.

You can get values of the top and bottom of the shell but these values also include bending stresses. There are 4 values that SolidWorks gives for a shell element face stress.

<u>Shell Face Stresse Value Designations</u>

Top - Total stresses (bending + membrane) at the top face.
Bottom - Total stresses (bending + membrane) at the bottom face.
Membrane – membrane stress component.
Bending - Bending stress component.

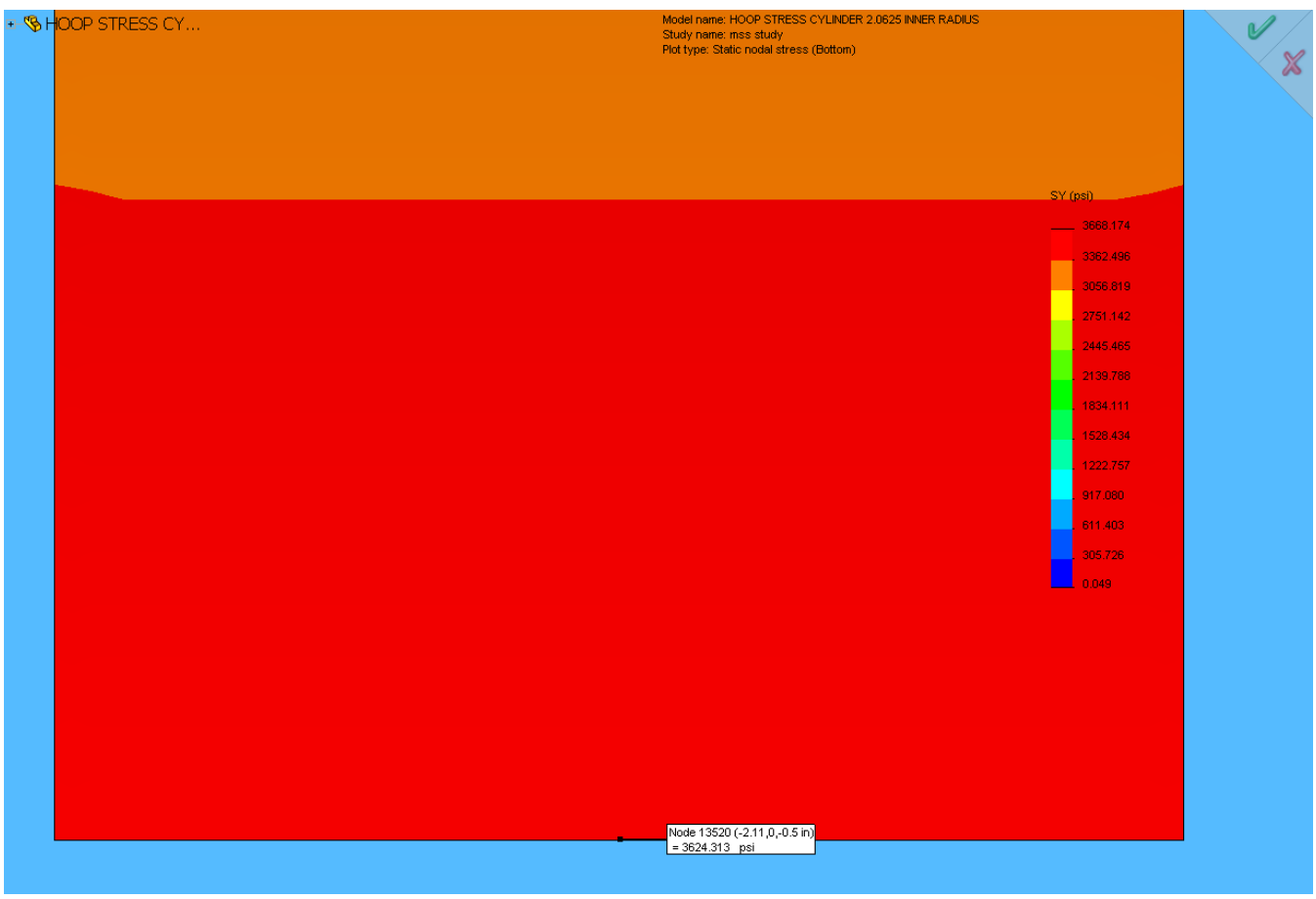


Figure 17: Hoop and Radial Stress- Sigma Y with shell elements, calculated values are from the Bottom surface (Orange color) of the shell and shown in rectangular coords, probed at mid-point.

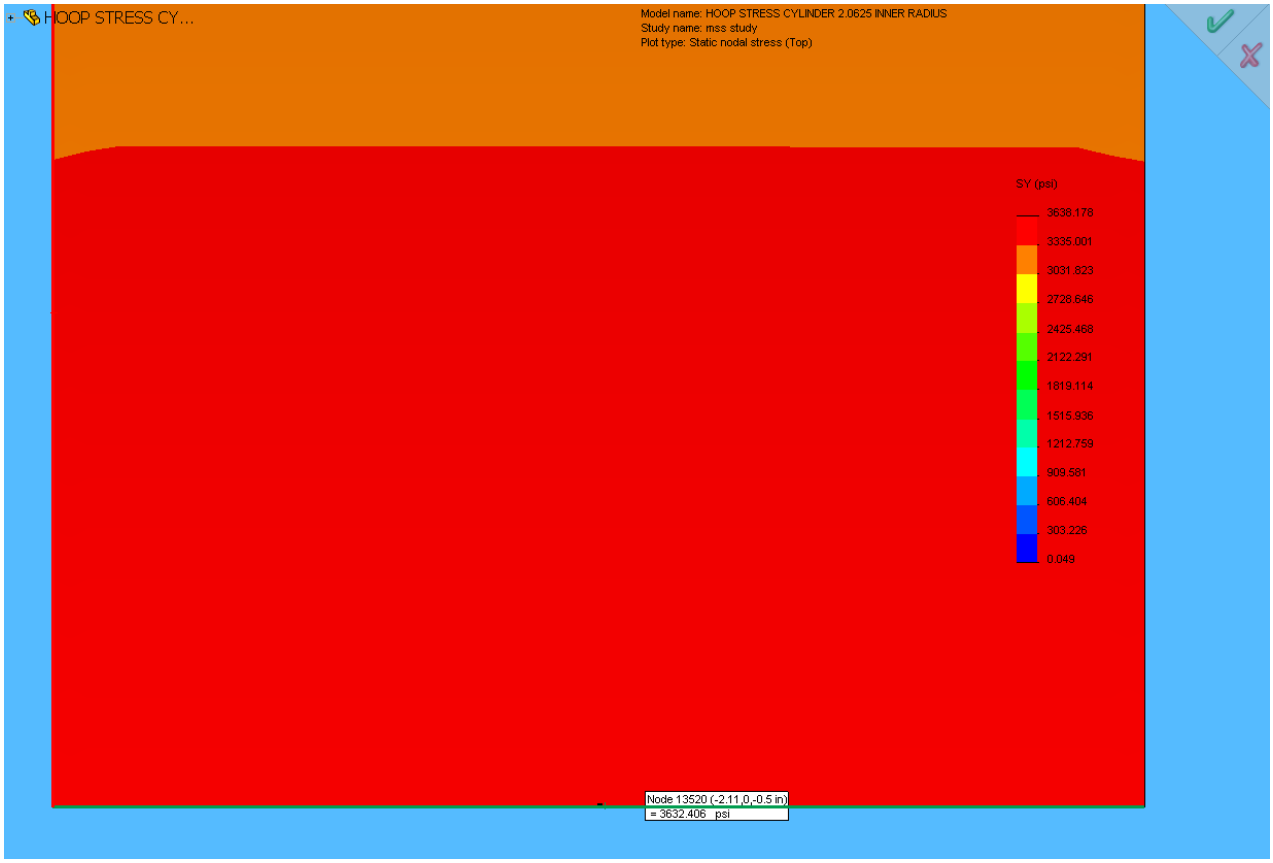


Figure 18: Hoop Stress - Sigma Y with shell elements, calculated values are from the Top surface shown in rectangular coords and probed at the mid-point.

The values for the hoop stress probe went up from 3624.313 psi (fig 17), bottom of the shell to 3632.406 psi for the mid-point probe value at the top of the shell (figure 18), and a difference of 8.093 psi. Now let's check the membrane measurement.

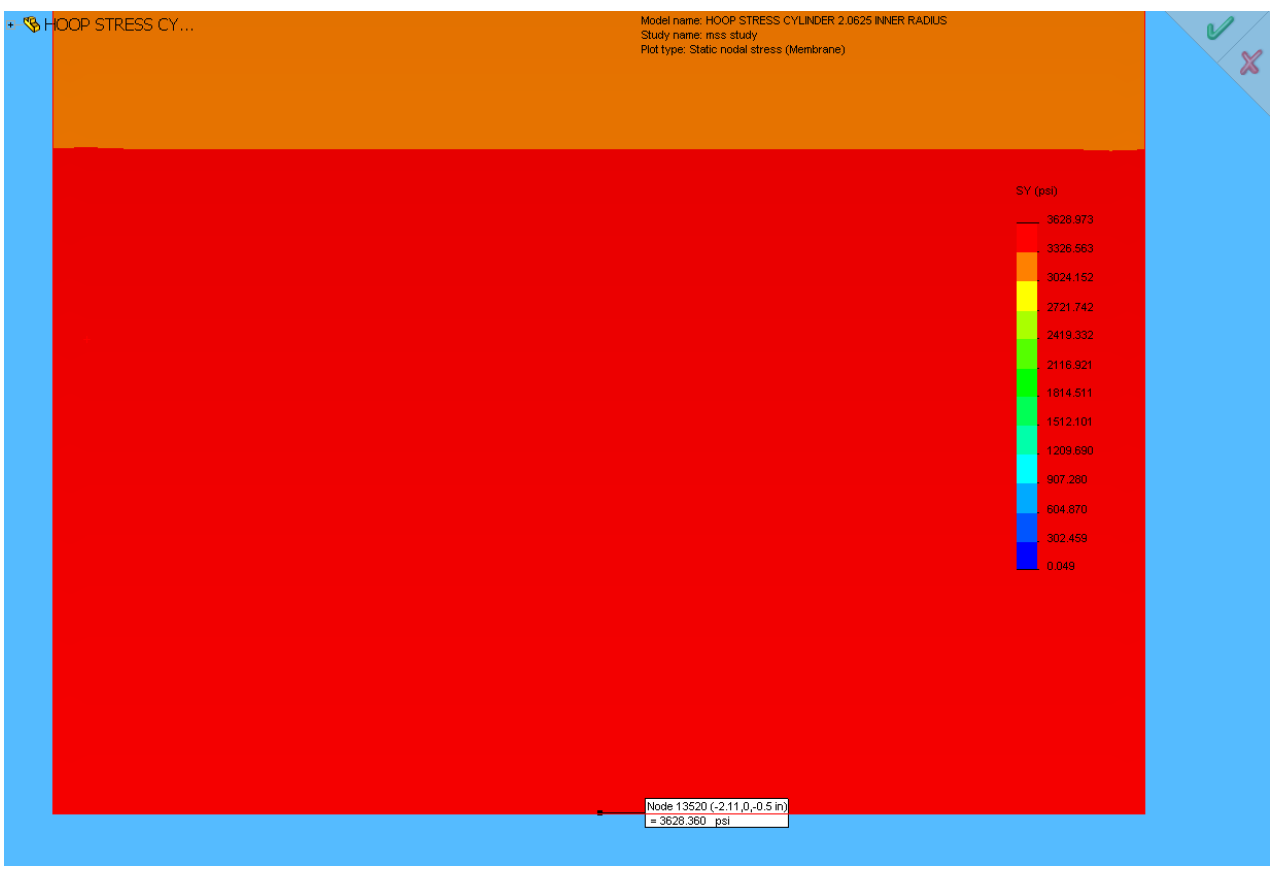


Figure 19: Probed Membrane value for hoop stress, Sigma Y rectangular coords.

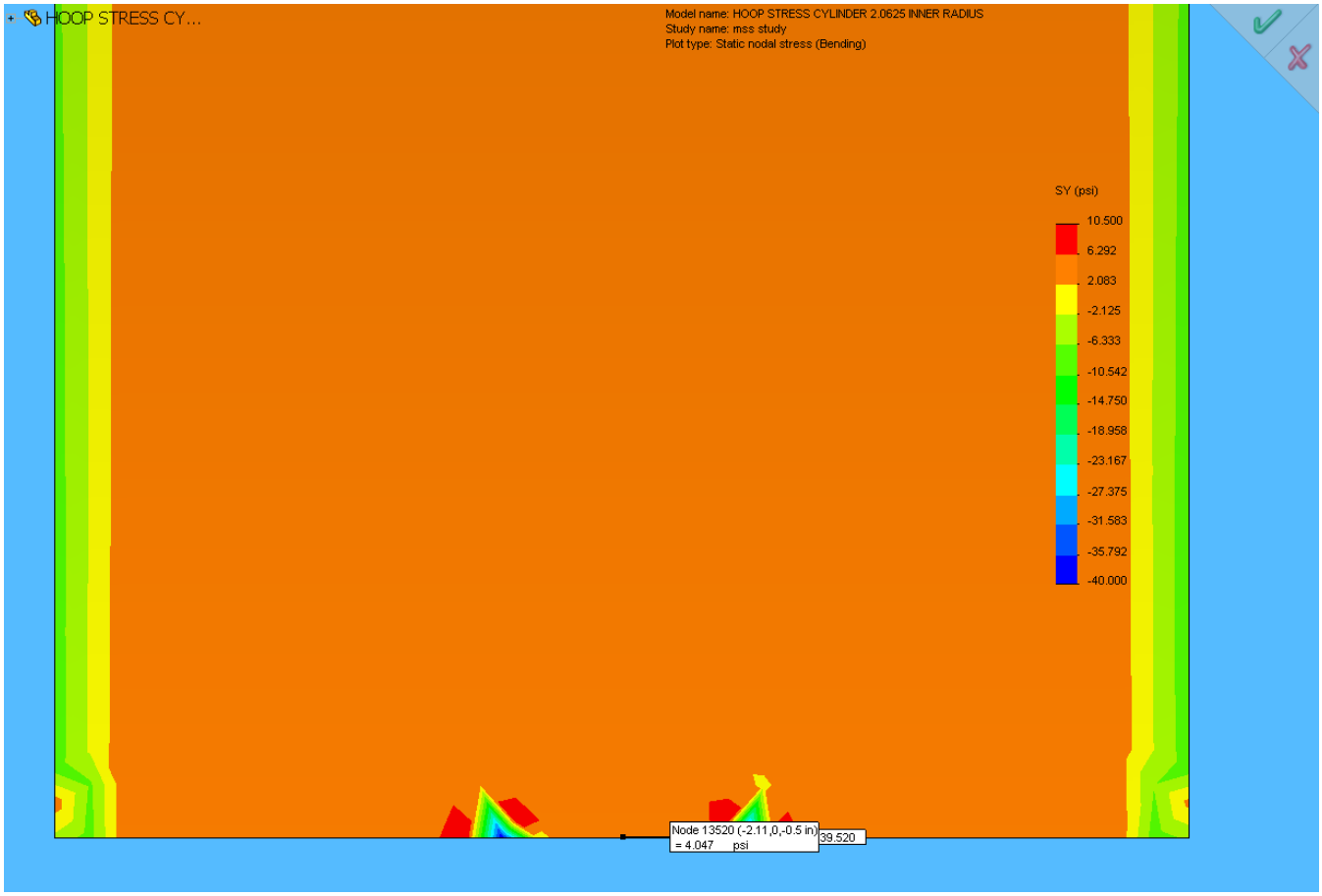


Figure 20: Bending stress values in rectangular coords SY.

The probed Membrane value is 3628.360 psi and our probed bending value is 4.047 psi.

3628.360 + 4.047 = 3632.407 for the top our probed value in figure 18 was 3632.406 psi

3628.360 – 4.047 = 3624.313 for the bottom our probed value in figure 17 was 3624.313 psi

It would appear we have a bending stress present in the hoop stress albeit very low.

There is a .001 psi difference in the top value number calculation, a strong case for significant figures. I am sure had I set my decimal point one or two more digits out I would have nailed the number as I did for the bottom of the shell calculation. The hand calculation for the hoop stress needs to be redone with the meridian value for radius to get an accurate measurement. From equation 4 we get:

Equation 4: $\sigma_H = (160\ lb/in^2)\ (2.109\ in) / .093\ in = 3628.38709$

$\sigma_H = 3628.387\ psi$

% error Bottom of shell surface = ((3624.31 – 3628.39) / (3624.31)) * 100 = .11257 or .113%

Mid-Surface Shell element Hoop Stress % error at the bottom (mid point value) $_{rect.\ coords}$ = .113 %
Acceptable

% error Top of shell surface = ((3632.41 – 3628.39) / (3632.41)) * 100 = .11067 or .111 %

Mid-Surface Shell element Hoop Stress % error at the top (mid point value) $_{rect.\ coords}$ = .111 %
Acceptable

The above errors are for argument sake not really accurate, they represent the membrane stress and the bending stress within the element. Though the percent error in each listing above, from the top reading to the bottom is quite low the true error is for the membrane value of 3628.36 psi and should be looked at.

% error membrane surface = ((3628.36 – 3628.387) / (3628.36)) * 100 = .0007441378 or .00074%

Mid-Surface Shell element Hoop Stress % error at the membrane (mid point value) $_{\text{rect. coords}}$ = .0007 %
Extremely Acceptable

I guess we can see why shell elements are the preferred method of analysis for sheet metal and thin walled vessels, the error percentage here is extremely low and if one wanted to adjust the significant figures to one decimal point we would get identical numbers. However we must keep in mind that these are ***membrane or meridian*** values of Hoop stress in the wall of the cylinder.

Recall from the solid element analysis where we calculated a hand value of 3548.4 with a radius value at the ***inner*** wall of 2.0625 and then compared the SolidWorks values at the mid-point to that. In both cases here we have very acceptable error values, but one analysis does show a higher value of stress than does the other at the mid-plane of the cylinder wall, there is almost an 80 psi difference in value and there is a significantly lower error value as well. This difference can make a significant impact on your Factor of Safety calculations and the reason for this difference is the different ***radius values*** that are used in the calculations.

Next we can again look at the cylindrical coordinate system values for the hoop stress.

Hoop Stress with Mid-Surface Shells in Cylindrical Coordinates

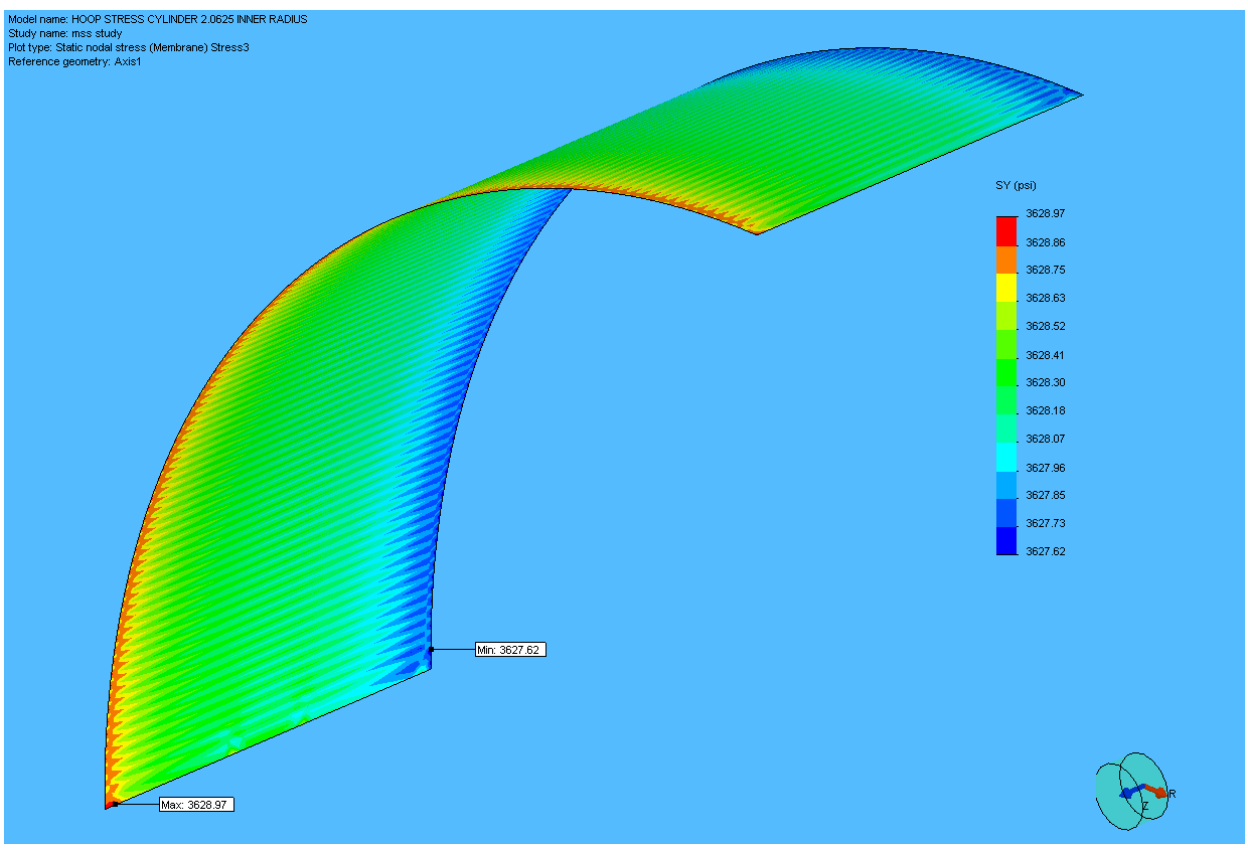


Figure 21: Hoop Stress SY at membrane in cyl coords

As we see in figure 21 the hoop stress appears to not be uniform, going from an upper value of 3628.97 psi to a lower value of 3627.62 psi. The values for the charts are done automatically in SolidWorks for you and sometimes they need a little refinement. If I look at the upper and lower values I can set up the

chart a little bit better. If I set my lower boundary to 3600 and my upper boundary to 3630 we should get a much better graphical representation of the hoop stress.

What I need to do is to extend my chart boundaries just a bit to encompass all of the upper and lower values of the stress. In other words I adjusted the stress magnitude display parameters. Figure 22 shows the results of the adjustment and we see a much better uniform graphical representation of the hoop stress. Since this hoop stress is measured at the membrane I can either probe for the value or I can get a listed value of the entire flat face. It doesn't matter what side you select here as this is the membrane and both sides are equal. (And yes, I checked) The listed average value for the hoop stress on the entire flat face was 3628.1 psi. The max was 3629 psi, the min was 3627.6 psi, the RMS mean was 3628.2 psi and the redefined sigma H value for the membrane radius of 2.109 was 3628.387psi, so our error is:

% error = ((3628.1 – 3628.387) / (3628.1)) * 100 = .00791 or .008%

Mid-Surface Shell element Hoop Stress % error at the membrane (avg listed value) $_{\text{cyl coords}}$ = .008 %
Acceptable

Note the errors are the same both in cylindrical and rectangular and with different methods used to find the stress. The variation in the stress at the membrane from cylindrical to rectangular was only .26 psi.

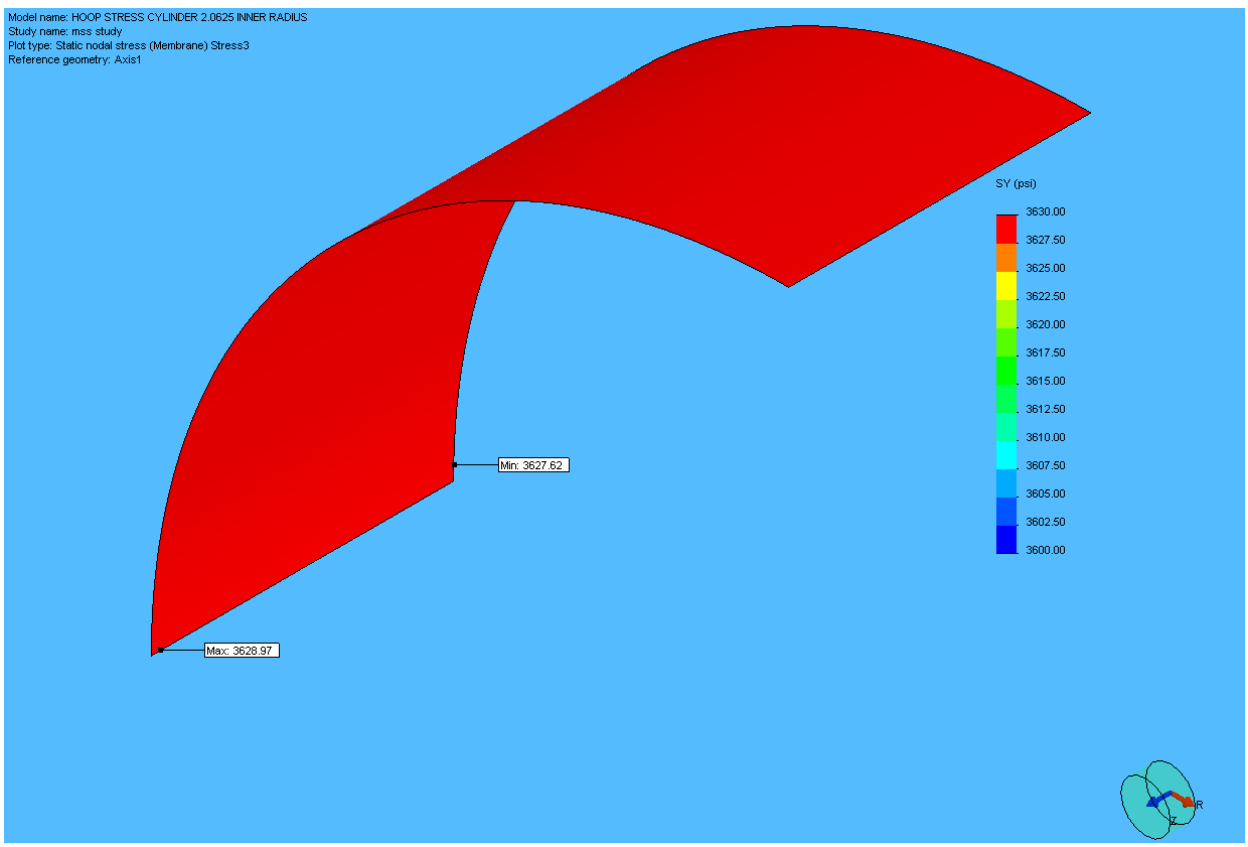


Figure 22: Hoop Stress at membrane with redefined chart boundary's.

Figure 23 shows the probe mid-point value of the shell and its value is 3628.28 psi, so our average listed value of 3628.1psi for the hoop stress was pretty good and our % error is reasonable.

Figure 24 shows the probe results for the bending stress all the way up the shell surface which could not have been done in rectangular coords. Note the value of the bottom mid-point probe value is exactly the same as the one in rectangular coords.

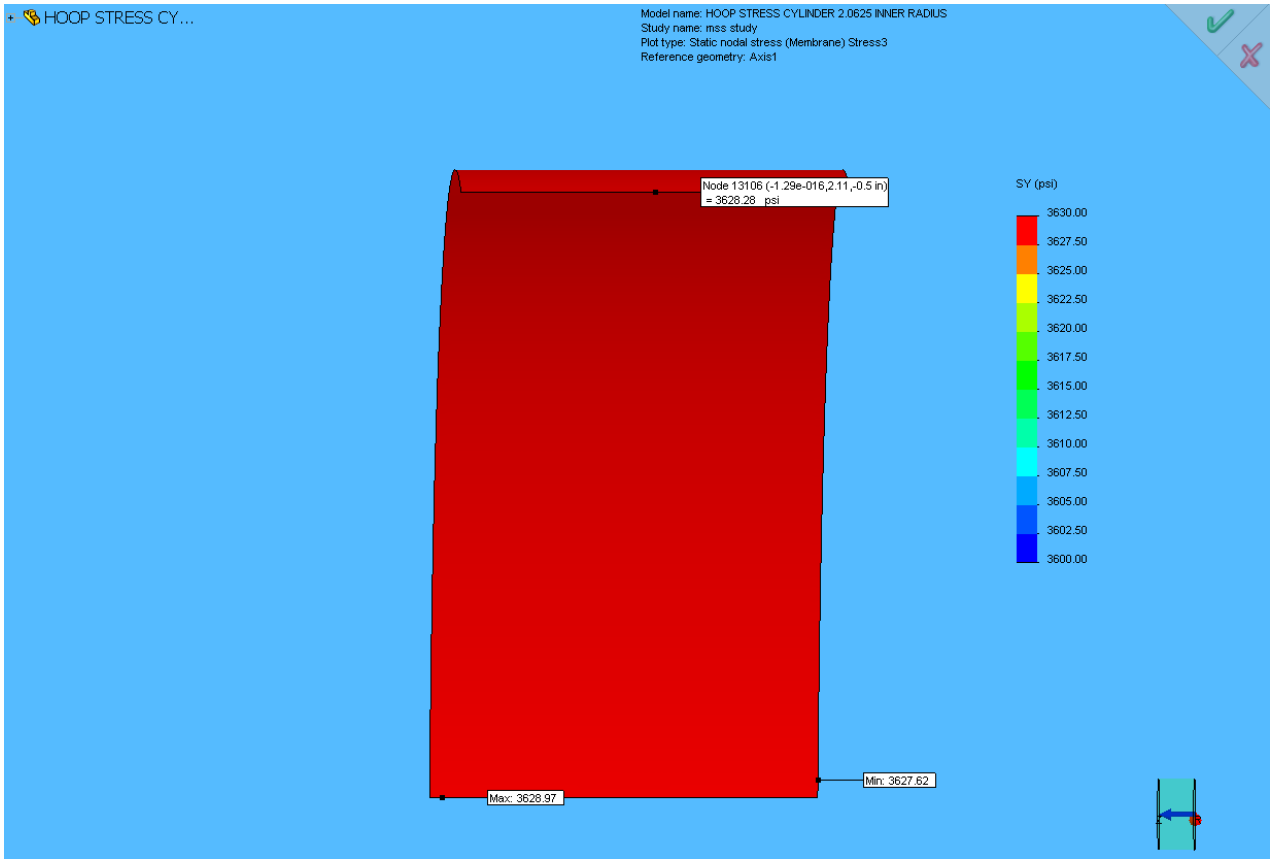


Figure 23: Probed value (node # 13106) for hoop stress at mid-point of shell at the membrane, cyl coords.

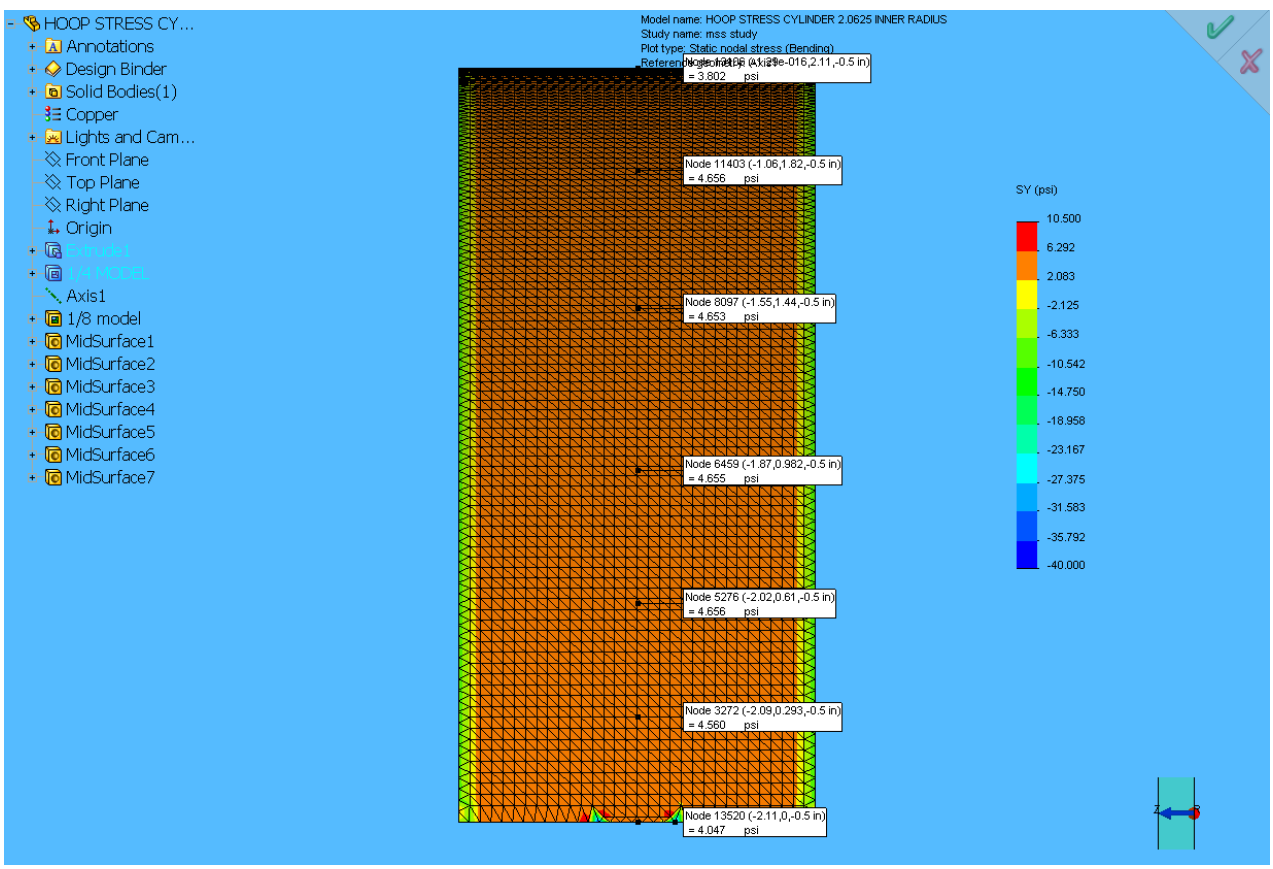


Figure 24: Probed values for bending stress in cylindrical coords

One thing that I happen to notice while looking back at the solid element analysis from either figure 13 or 14 is that the mid-surface shell element membrane value seems to line up better with the values found at the inner part of the cylinder wall for the hoop stress in the solid study, one would think that they would be more in line with the values we had at the mid-points.

Example: figure 13 has a solid element value for the hoop stress at the inner part of the cylinder at 3630.9 psi and this does make sense since this is the point where the 160 psi internal pressure is applied, then the values decrease thru the wall to the value of 3548 at the mid-point and decreased further to the outer wall.
Our hoop stress listed average value for the shell membrane was 3628.1 psi, which is a far cry from the probed solid element mid-point value of 3547.6 psi. I would think the value for the membrane would be more in line with the mid point probe value of the solid element since at the membrane there is not any bending stress added. However, as I surmised before, I believe the reason for this difference is due to the different radius values used in the calculations. (I am not positive about this it is only a guess)

Although the error percentage is well within the acceptable range for the shell element membrane hoop stress mid-point error value (.0007%), there is a significant difference in value for the listed average face error value (.008%) I am a bit baffled by the difference in values above. The fact that shell elements use those three extra degrees of freedom for rotation must somehow add a significant amount to the displacements as a whole as opposed to the solid elements. Remember SW calculates displacements first, strains second and then stresses.

The difference in values between the solid elements and the shells I believe is due to different radius values used for the calculations. One is based on a meridian wall radius value (2.109 inches) while the other is based on an inner wall radius value (2.0625 inches). The difference in value we saw was almost 80 psi for the hoop stress. The values for the errors were virtually non existent for the shell analysis (.0007%) while the solid elements came in higher at around .011%.

<u>Principal Stresses</u>

Cylinder devices can be open ended or closed ended. If open ended, a two dimensional state of stress is said to exists in the cylinder wall with radial and tangential (Hoop) stress components. If closed ended, a third dimensional stress is present called longitudinal (Axial*).These three applied stresses are mutually orthogonal and are* ***Principal****, since there is no applied shear from the uniformly distributed pressure.*

Shell membrane

We can also use the principal stresses to check our work. The first Principal stress or σ_1 should correspond with our Hoop Stress values we had before whether it be in rectangular coordinates or cylindrical, the values for the 1st principal stress should be the same. The membrane shell element in figure 25 shows almost the identical plot we had in figure 21. The only difference is that the max value is slightly higher. Adjusting the chart again (figure 26) gives the exact same plot we had in figure 22 and for all intensive purposes practically the same number values. Plotting Sigma 1 in cylindrical coordinates is shown in figure 27.

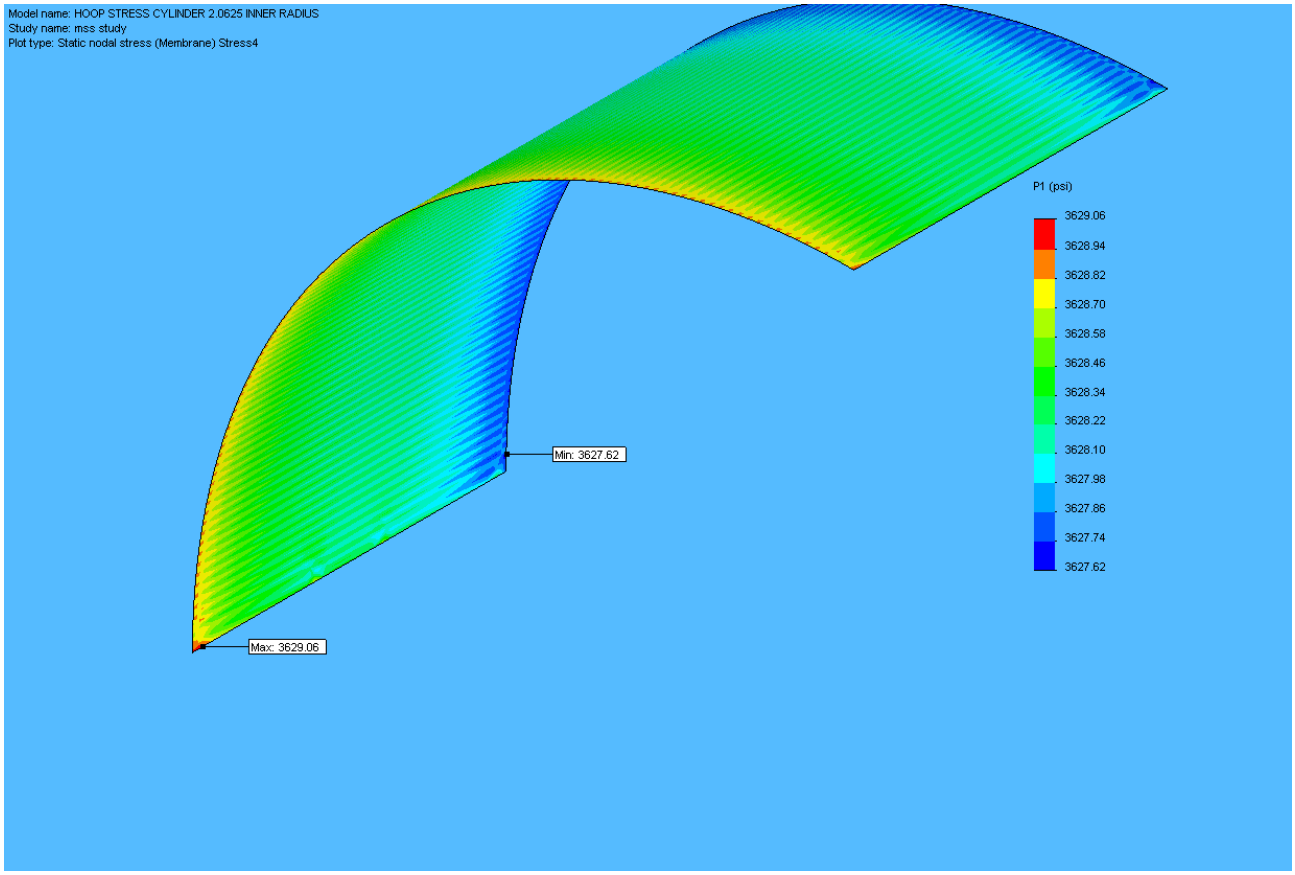


Figure 25: First Principal Stress, shell element membrane.

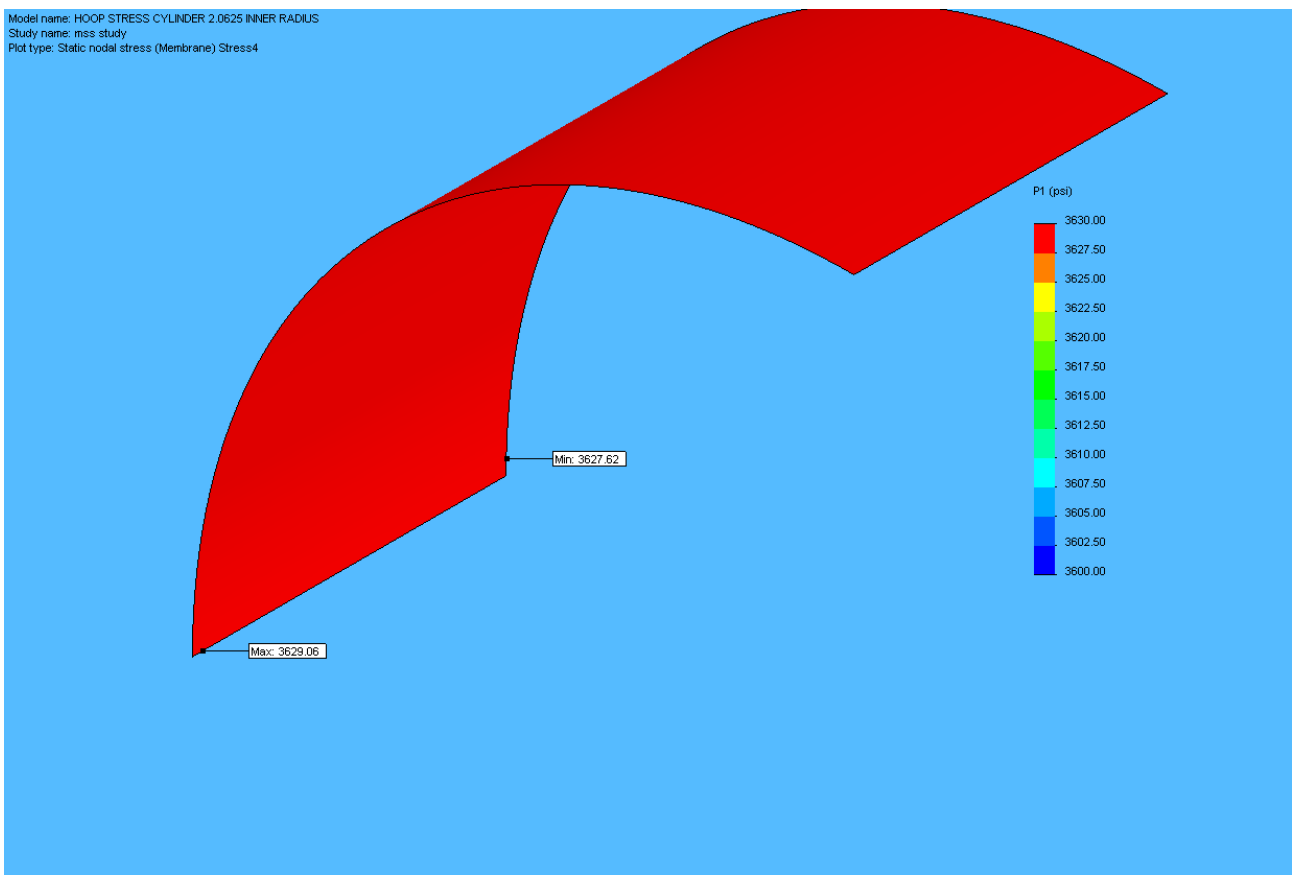


Figure 26: First principal stress (rectangular coords) with chart adjustment.

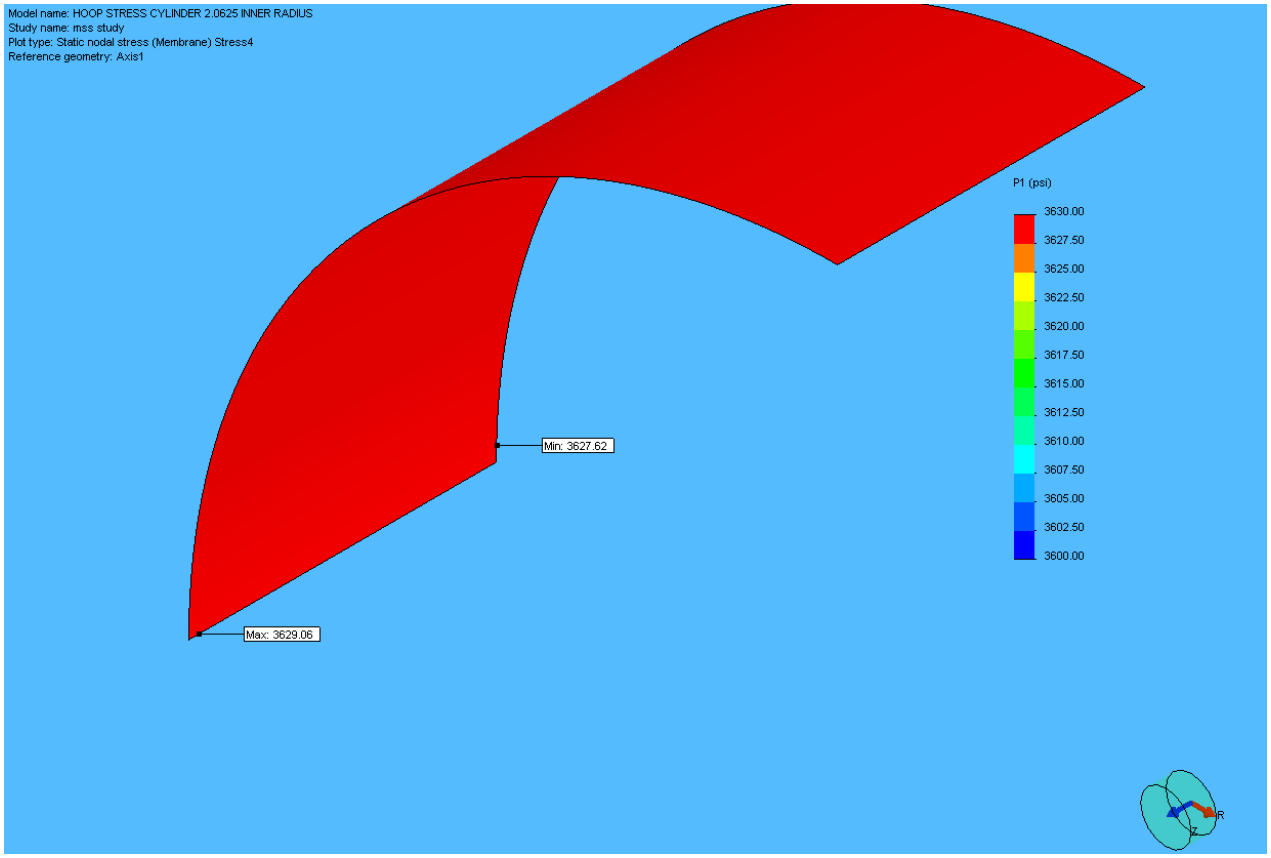


Figure 27: First Principal Stress in cylindrical coords with chart adjustment.

The error for all of these is still .0007% the exact same as we calculated for the membrane Hoop stress in cylindrical coords.

Principal Stresses

Solid Elements

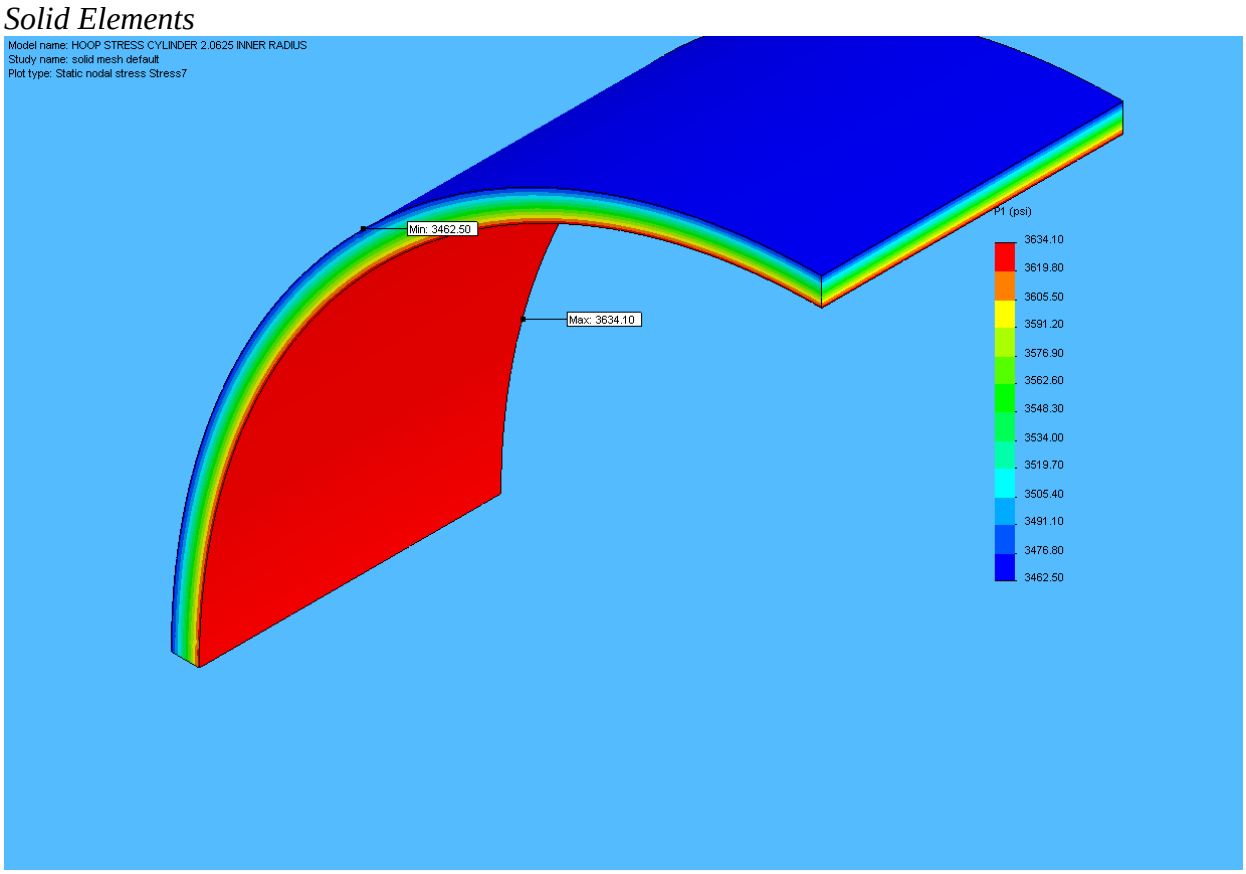


Figure 28: Solid Elements first principal stress (rectangular coords)

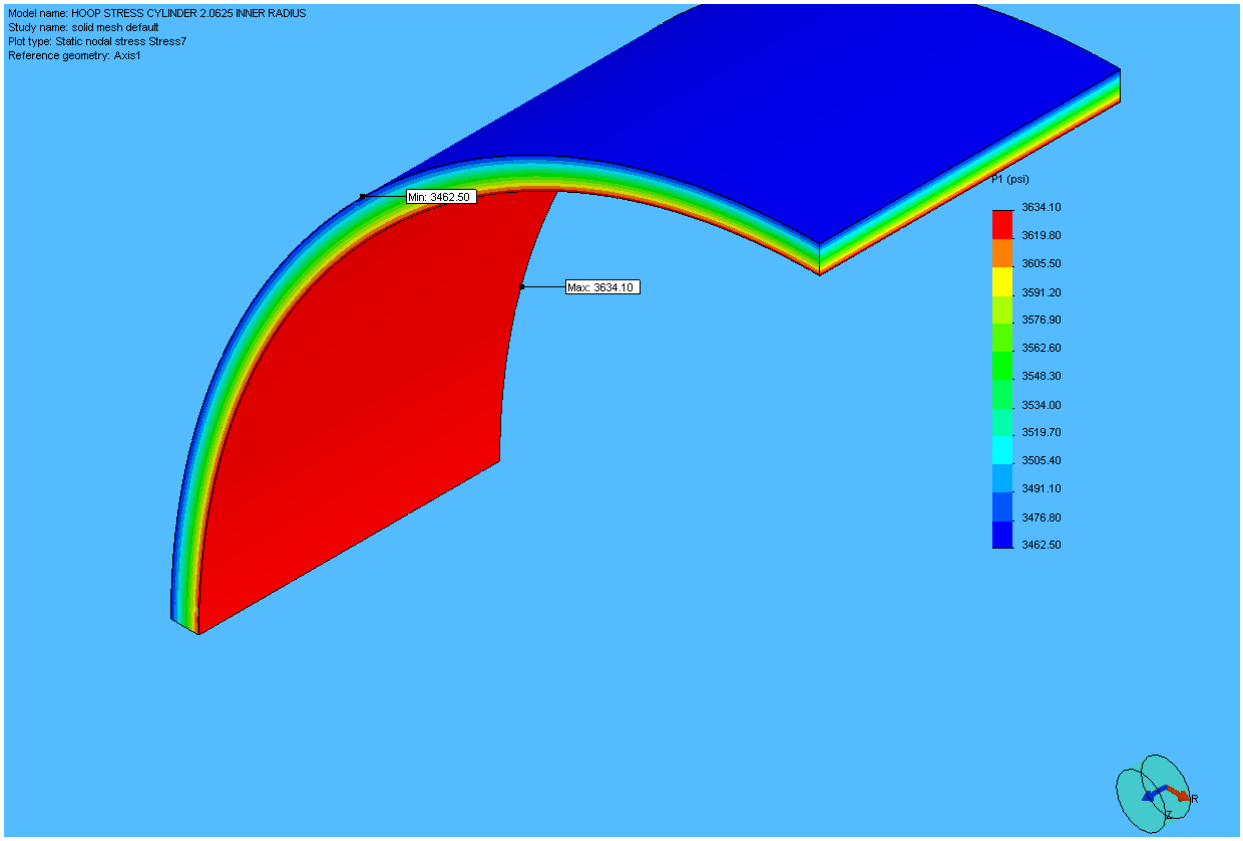


Figure 29: Solid Elements first principal stress (cylindrical coords)

Figures 28 and 29 show the first principal stress with solid elements plotted in both rectangular (figure 28) and cylindrical (figure 29) both are identical to the SY (sigma Y) plot in figure 12 that shows the Hoop Stress plot in Cylindrical Coords. When we look at the probe results as shown in figure 30 they are identical to the probe results in figure 13.

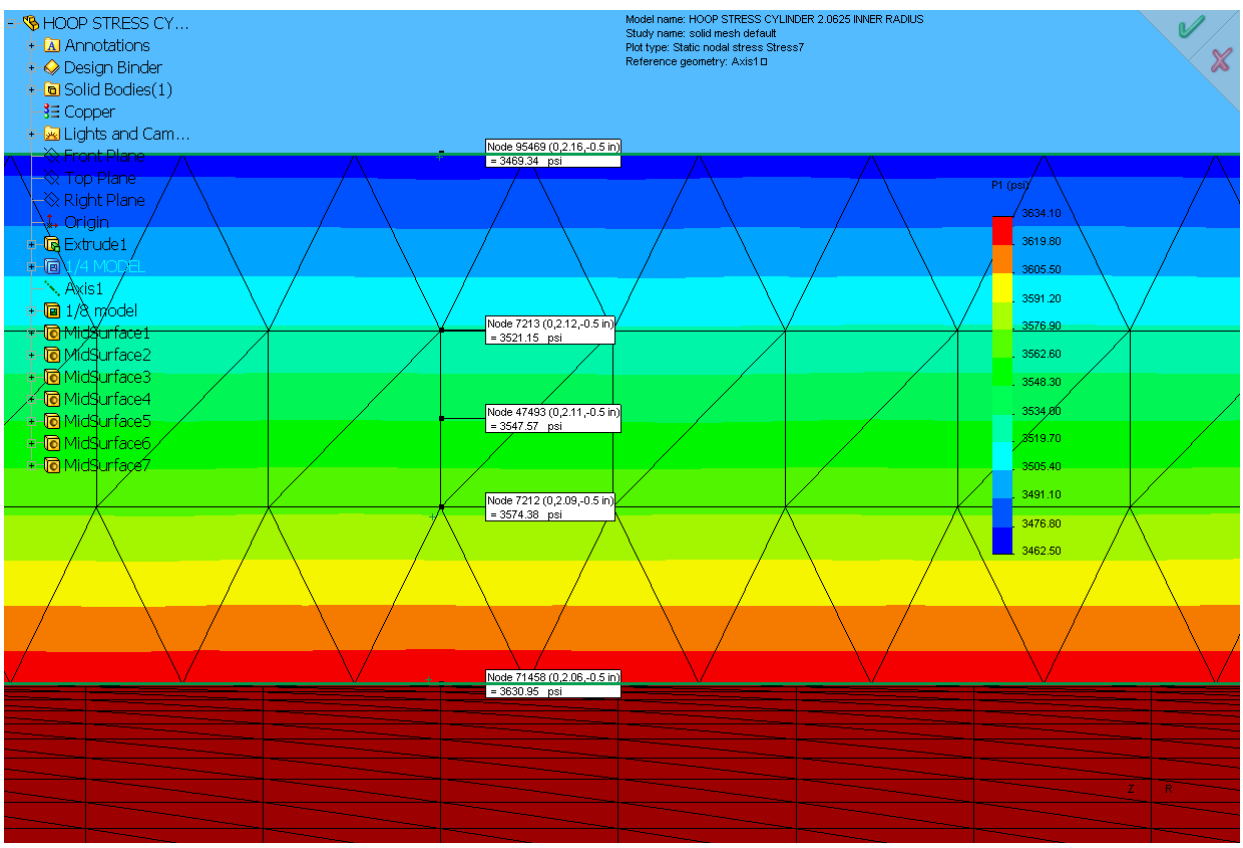


Figure 30: Solid Elements first principal stress probe results for Hoop stress. (Cylindrical coords)

So obviously there are quite a few different ways to look at Hoop Stress in SolidWorks and as I showed we do get some discrepancy in values depending on what coordinate system you use as well as what element type you use. If it were not for the fact that I had done my homework in calculating the classic stress results though we would not have know which values were better.

For me personally, I like the way the solid elements show the hoop stress distribution thru the wall. Did I have to put the cylinder in a cylindrical coordinate system? No I did not. The 1st principal stress would have been all that was needed to gain a good insight to the hoop stress and as I showed, the values for the 1st principal were exactly the same as the ones in the cylindrical coordinate system for SY. However with that said I would not have known that there was a bending stress going on that the shell elements showed me, it pays to look both ways.

As far as the plane stresses go in the rectangular system those were hard to get information from considering when we looked at stress in the X direction we found Hoop and Radial Stress. The same can be said for the Y direction except the hoop and radial stresses were at opposite spectrums, which we expected but what was not expected were the different values in the radial stress.

The shell elements did give us much better error numbers than the solids; .013% to .014 % for the solids as opposed to .0007% to .008% for the shells. Since this was such a simple part to analyze I suspect that when or if I do a more complicated part or an assembly I may have to use shells. Knowing this error range between the two element types however will come in handy later when making a call about safety. I will also be curious to see the differences for the other stresses as well.

The 5% rule of thumb for the stress variation thru the wall held for all cases were measurable, i.e. solid elements only.

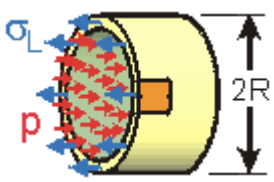
When making a call on safety I find it better to err on the side of caution and since the hoop stress is the major contributing stress to failure in a thin wall cylinder I will probably use the higher shell value of 3628 psi to make that decision.

Axial or Longitudinal Stress,(out of plane) classical equations from Mechanics of Materials Theory

For our axial stress, the **Free Body Diagram** and summation of forces calculation are shown below in equation 6.

Equation 6 **Axial Stress or Longitudinal Stress:** ***For capped ends only***

$$\Sigma F = 0 \rightarrow p\pi R^2 = \sigma L\,(2\pi R t)$$



This leads to the equation

$$\boxed{\sigma_L = pR/2t}$$

Figure 31: Axial Stress **FBD**.

Where

p = internal pressure
R = internal radius
t = thickness of material
L = length of the cylinder

Equation 6: σ_L or σ_A = (160 lb/in^2) (2.0625 in) / 2 (.093 in)

σ_L =1774.2 psi

Almost all pressure vessel design standards contain variations of these two above formulas (Sigma H and Sigma L) with additional empirical terms to account for wall thickness tolerances, quality control of welds and in-service corrosion allowances.

For example, the ASME Boiler and Pressure Vessel Code (BPVC) (UG-27) formulas are:

Spherical shells:

$$\sigma_\theta = \sigma_{\text{long}} = \frac{p(r + 0.2t)}{2tE}$$

Cylindrical shells:

$$\sigma_\theta = \frac{p(r + 0.6t)}{tE}$$
$$\sigma_{\text{long}} = \frac{p(r - 0.4t)}{2tE}$$

Where *E* is the joint efficient and all others variables are as stated above.

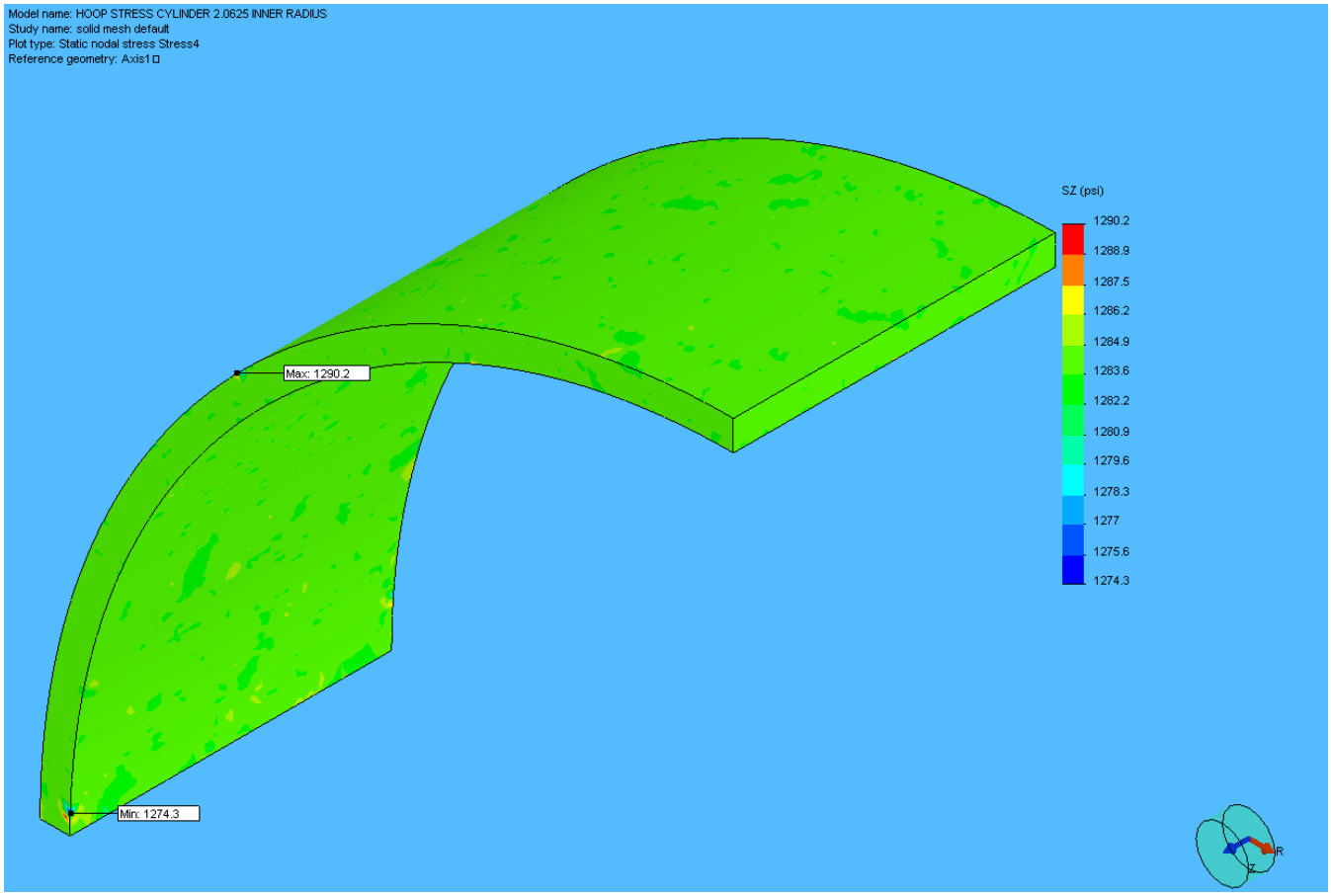


Figure 32: Longitudinal or Axial Stress (shown in cylindrical coords) Solid Elements

Solid Elements

Figure 32 shows the Longitudinal or Axial stress in our model, Sigma Z or SZ. Our predicted value for this was half the hoop stress or 1774.2 psi and as we see we get a varying value from a minimum on the inside of 1274.3 psi to a maximum value of 1290.2 psi on the outside, this value should be uniform and our automatic graphics do basically show that it is. The listed average value for the curved face is 1284 psi, probe results were basically the same. This value is the same in the SZ direction (rectangular), or the SZ (cylindrical) or 2nd principal P2.

Though the value variation is quite small from the inside to the outside and within our 5%, we can notice that we have all positive numbers in the chart indicating a state of tension. Also notice what appear to be quite a few splotchy yellow marks through out the model; I believe these are rounding errors again within the program.

Axial Stress % error at the axial face (avg listed value) = ((1284 -1774.2) / (1284)) * 100 = 38.18%
Not Acceptable

Shell Elements

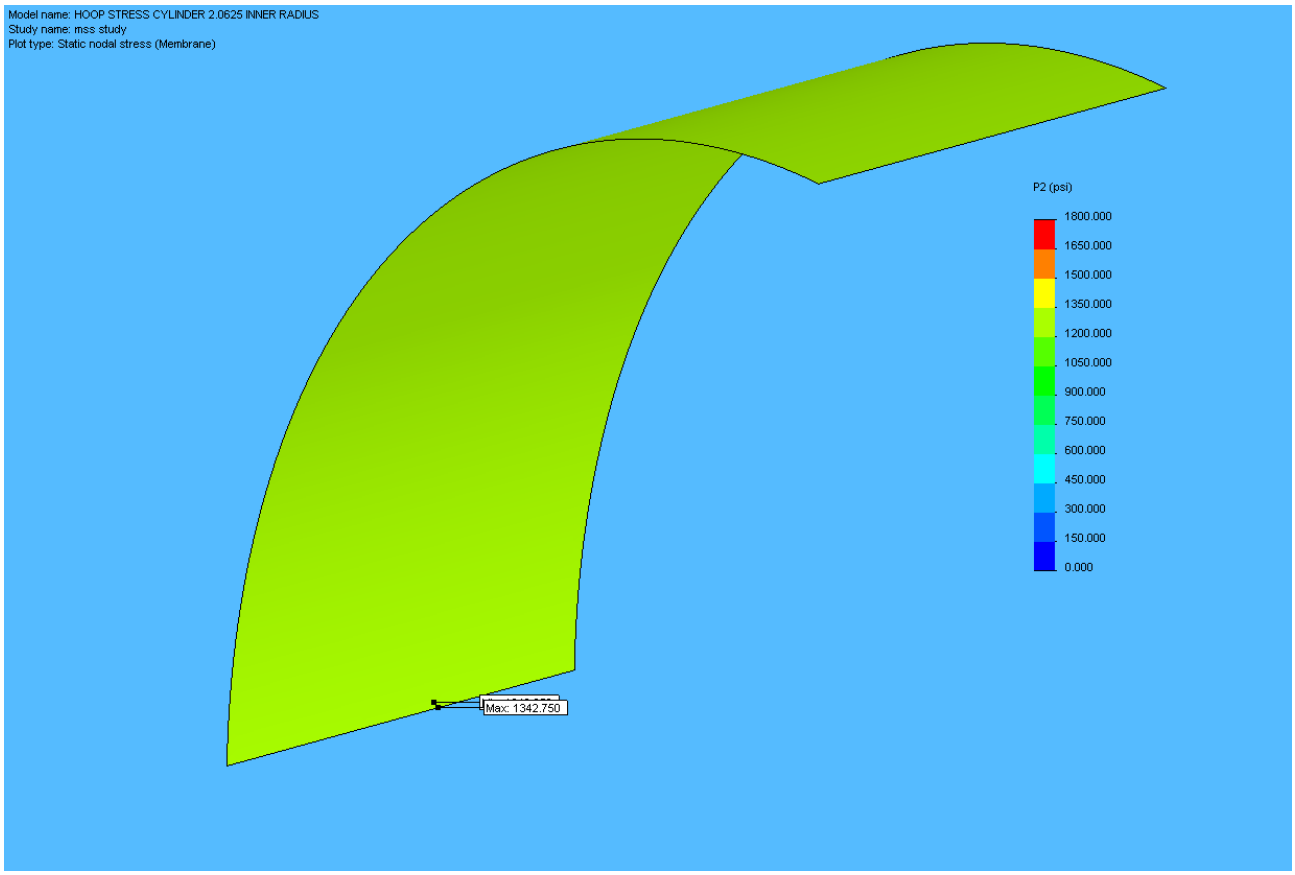


Figure 33: Shell element 2nd principal stress at the membrane.

Figure 33 shows the 2nd principal stress in rectangular coordinates; I changed the chart display again to go from 0 to 1800 psi just to give a different look. You can see the max value is 1342.75 psi. The listed average value for the entire face was 1342.3 psi. Again I checked all the coordinate systems and axis and the membrane, they are all the same values. There is a difference between the two measured values for solids and shells of 58.3 psi, the shell value being the higher value. This difference might be due to the fact of that extra rotational axis I spoke of before or as before a different radius value. We probably also have a bending stress as well.

Calculated meridian value; (160(2.109))/ (2 (.093)) = 1814.1935

Axial Stress % error at the membrane (avg listed value) = ((1342.3 -1814.2) / (1342.3)) * 100 = 35.16%
Not Acceptable

Comments about the Longitudinal or Axial Stress

As I mentioned before I had to restrain the model on the flat face of the back side of the cylinder to prevent ridged body motion in the Z or axial direction. In doing this I am not sure if we have a true representation of an open cylinder, for all intensive purposes the axial stress for a cylinder with no end caps should be 0. However, this may not be all that is going on.

The material is in a state of plane stress (in-plane), X-Y if no stress components act in the third dimension (out of plane), Z. This occurs commonly in thin sheets loaded in their plane. However, a state of plane stress is not a state of plane strain. The sheet will experience a strain in the z direction equal to the Poisson strain contributed by the x and y stresses. In other words we can calculate the out of plane strain (Z) from the in plane stresses (X-Y). From Hooke's Law and Poisson's ratio we get:

Equation 7

$$\varepsilon_z = (-\nu/E)\ (\sigma_x + \sigma_y)$$

Where;

Sigma X = 3548.4
Sigma Y = 3548.4
Poisson ratio (ν) = .37 (SolidWorks value for Copper)
Modulus of Elasticity (E) = 15.954 X 10^6 (SolidWorks value)

However, since $\sigma_x = \sigma_y =$ the Hoop Stress, then only one stress should be used; it is all the same stress.

So our strain in the Z direction would be

$$\varepsilon_z = (-.37/15.954(10^6))\ (3548.4)$$

$$\varepsilon_z = -8.6(10^{-5}) = -.00008229 \quad \text{a negative strain indicates contraction.}$$

From Hooke's Law the definition between stress and strain is;

$$\varepsilon_z = \sigma_z / E$$

Then

$$\sigma_z = \varepsilon_z E$$

$$\sigma_z = .00008229 * (\ 15.954(10^6)$$

$$\sigma_z = 1312.91 \text{ psi}$$

The results from the average listed value SolidWorks gave for axial stress were 1342.3psi

% error = [(1342.3 – 1312.91) / 1342.3] * 100 = 2.189%

% error for Axial Stress based on Z strain calculation = 2.2% Acceptable

Axial Bending Stress

Again we will have to probe the center of the shell at the membrane to get an acceptable axial stress like we did before and then we can probe the bending stress plot and see if our numbers line up with the upper and lower shell plots for the axial stress.

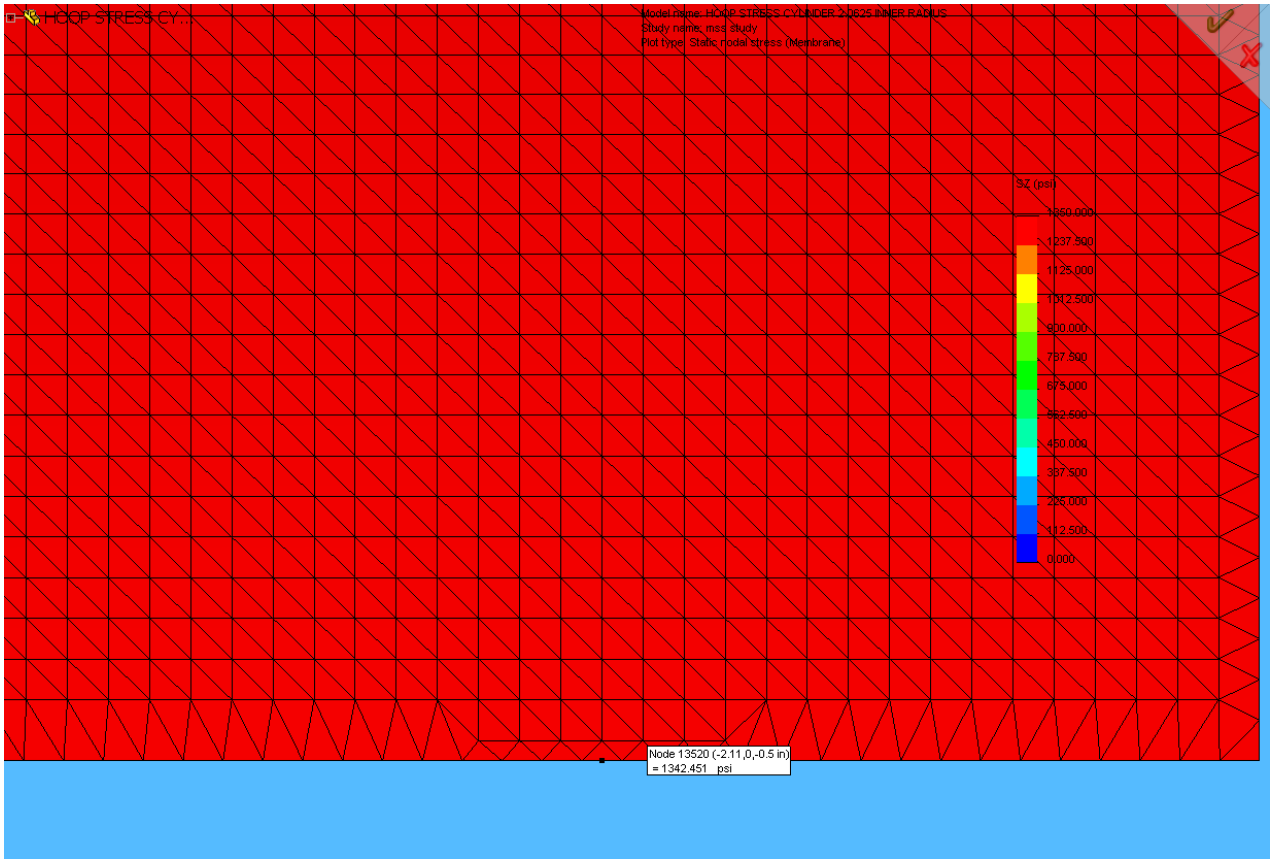


Figure 34: Probed shell value for axial stress at the membrane

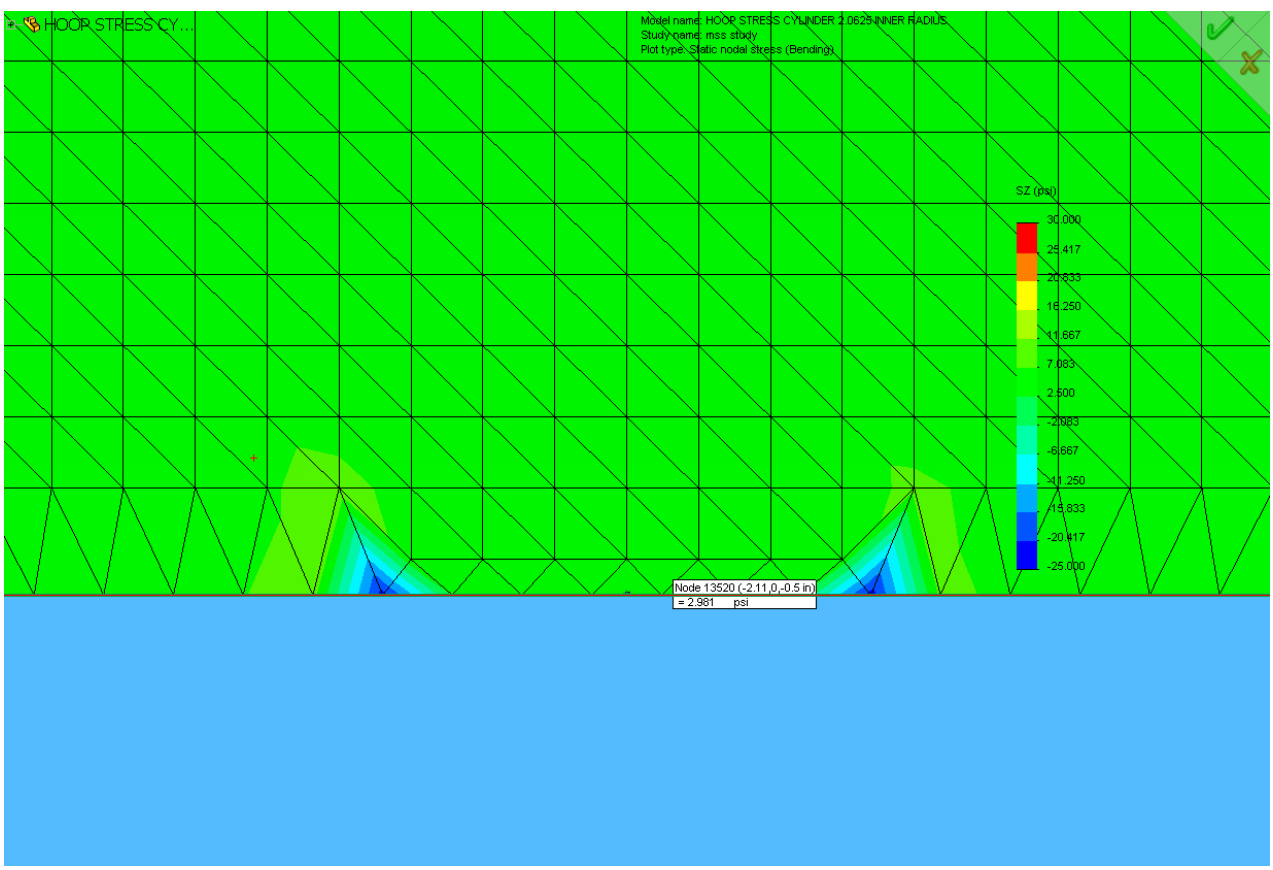


Figure 35: Probed result for axial bending stress.

Membrane + Bending Stress = 1342.451 + 2.981 = 1345.432
Membrane – Bending Stress = 1342.451 – 2.981 = 1339.47

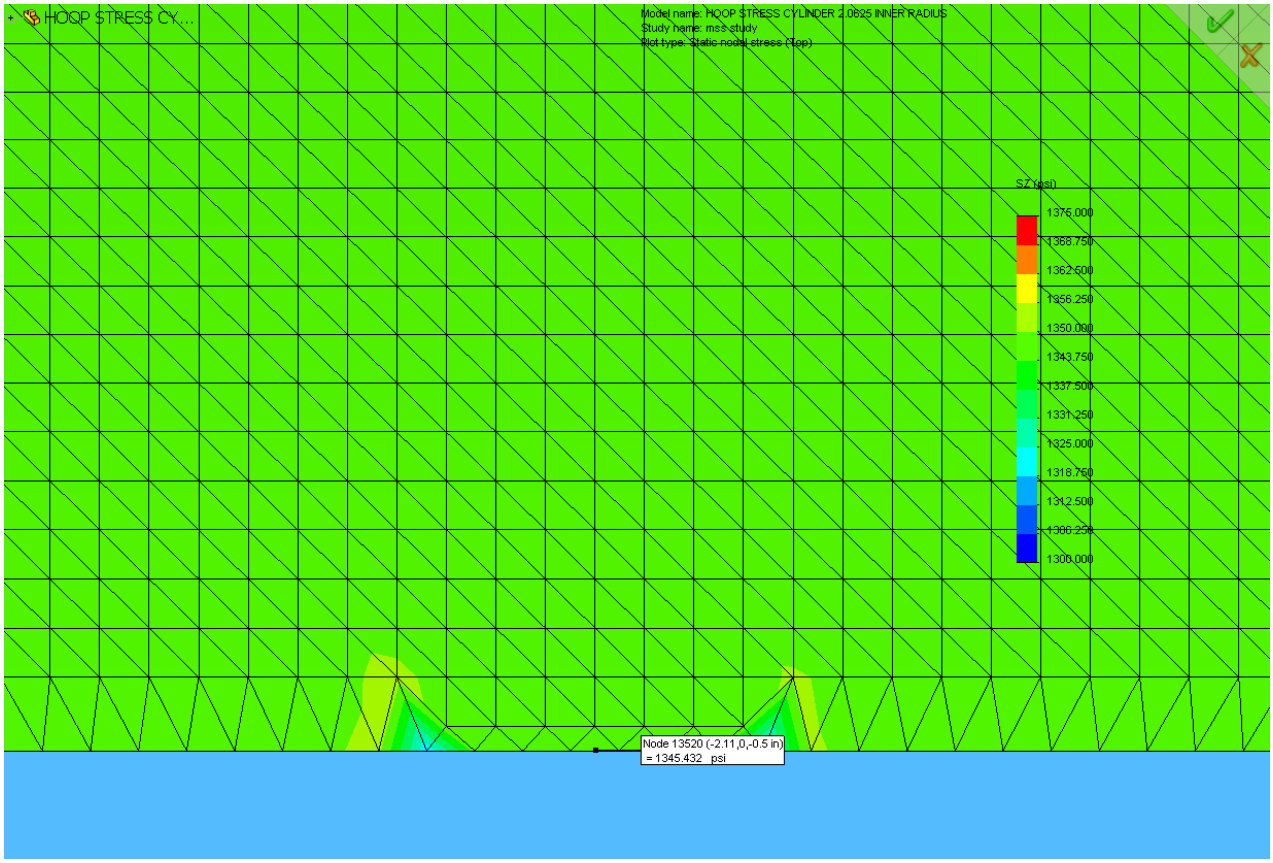


Figure 36: Probed result on top of shell for axial stress

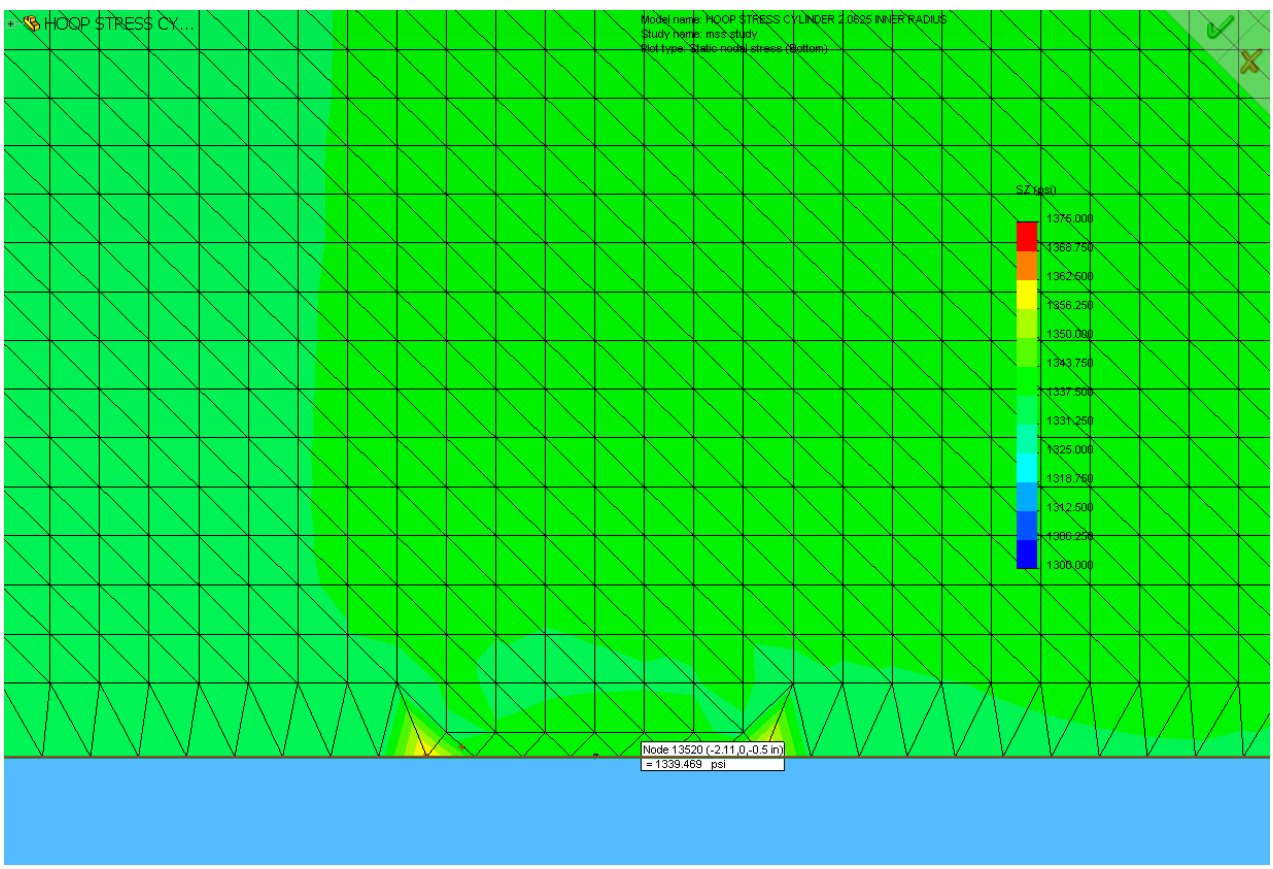


Figure 37: Probed result for axial stress on the bottom of the shell.

For the top we have 1345.342 and the membrane plus bending was 1345.432, right on the money. For the bottom we have 1339.469 and the membrane minus bending was 1339.47, off by .001 again. 2.981 psi bending stress is present with the axial stress.

Again we have a bending stress present and please note that it is a different magnitude. The one present in the hoop stress was 4.047 psi as opposed to this one which is 2.981 psi. I see no need to check the 2nd principal stress as these values will be exactly the same.

Radial Stress and Saint Venant's Principal

An element in a cylindrical pressure vessel is said to be subjected to biaxial stress (i.e., a normal stress existing in only two directions).
For thin walled pressure vessels, the radial component is assumed to equal zero throughout the wall since the limiting assumption of r/t = 10 results in σ_H being 10 times greater than $\sigma_r = p$ and σ_A being 5 times greater than $\sigma_r = p$.
In reality, the element is subjected to a radial stress, σ_r which acts along a radial line. The stress has a compressive value equal to the pressure, p, at the inner wall,(boundary condition 1) and decreases through the wall to zero(boundary condition 2) at the outer wall (plane stress condition) since the gage pressure there is zero.

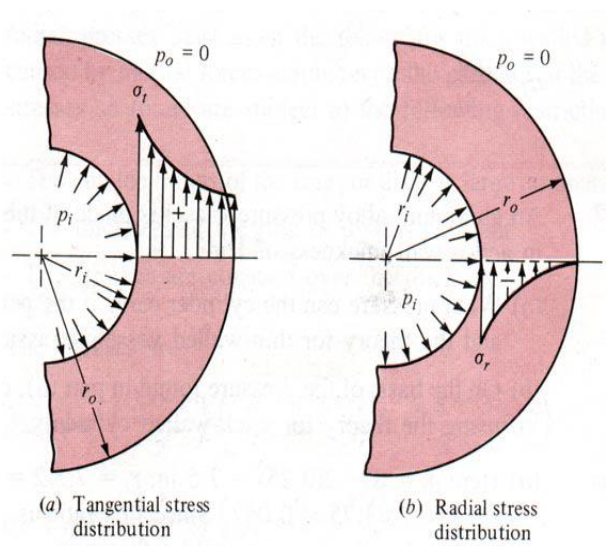


Figure 38: Stress distributions of Hoop and Radial Stress.

Equation 8 **Radial Stress**

$\sigma_r = [(r_i p) / (r_o - r_i)] * [1- (r_o / r_i)]$

Where:
r_i = 2.0625 in.
t = thickness of the material = .093 in.
r_o = outer radius = inner radius plus the thickness of the material = 2.0625 +.093 = 2.1555 in.
p = internal pressure = 160 psi.

σ_r =[(2.0625 * 160) / (2.1555 – 2.0625)] * [1 – (2.1555/2.0625)]
3548.4 = pr_i/t = hoop stress -.04509

σ_r = -159.999 psi

The number above is negative which indicates compression at the inner wall of the cylinder just like we expect and for all intensive purposes is equal to the inner pressure of 160 psi.
The equation above is for any wall thickness and is not restricted to a particular r/t ratio as are *Equations 4* and *5 for hoop and axial stresses.*

Note that the hoop and radial stresses (σ_h and σ_r) are functions of r (i.e., they vary through the wall thickness) and that the axial stress, σ_A, is independent of r (i.e., is constant through the wall thickness.) Figure 38 shows the stress distributions through the wall thickness for the hoop and radial stresses. Note that for the radial stress distributions, the maximum and minimum values occur, respectively, at the outer wall ($\sigma_r = 0$) and at the ($\sigma_r = -p$) as noted already for the thin walled pressure vessel.

Now let's see what SolidWorks gives us.

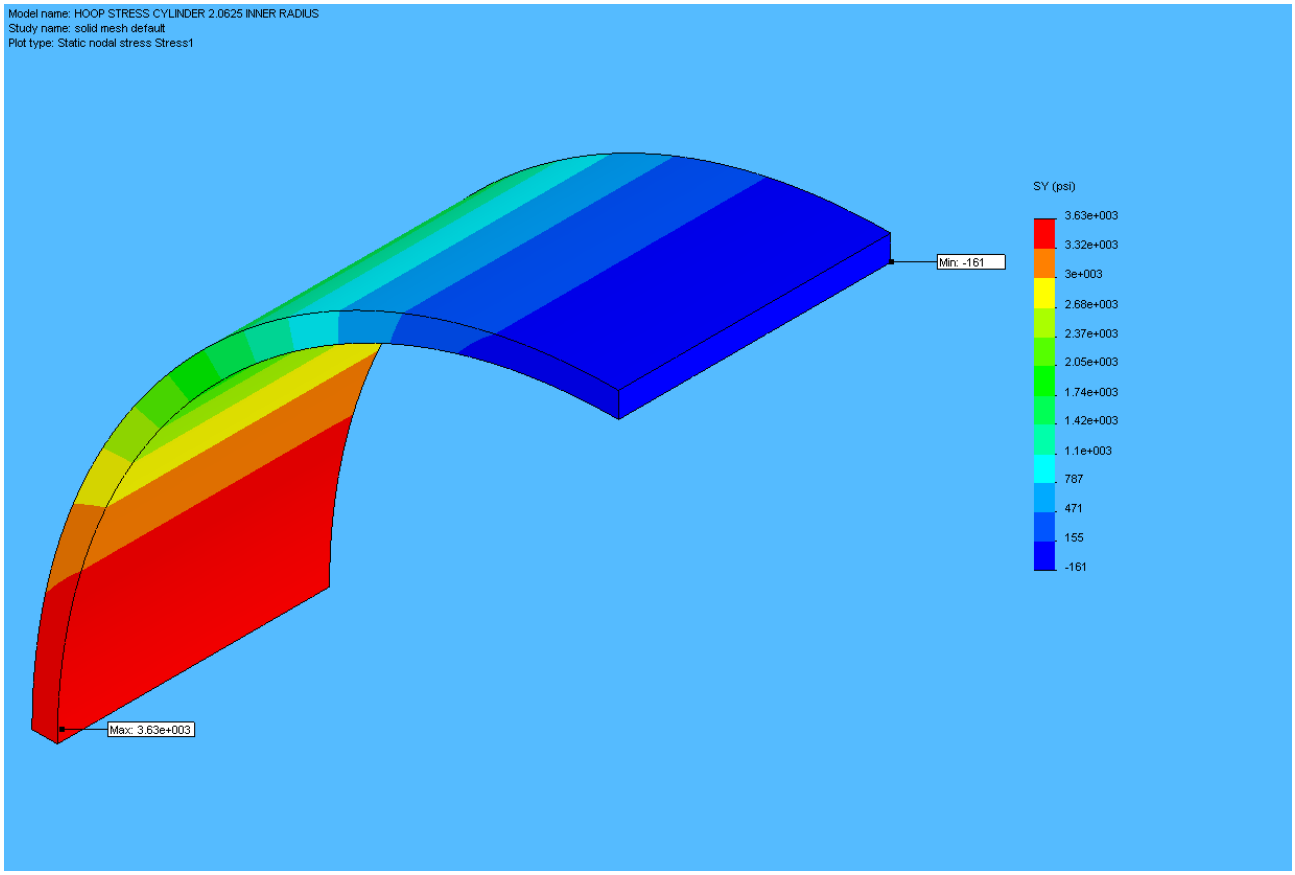


Figure 39: Figure 5 revisited for a look at radial stress in rectangular coords.

Figure 39 shows the plot I created in rectangular coords when we were looking at the in plane hoop stress at the beginning of this report. Again we see we have two different stresses in the figure, at the top is the radial stress and at the bottom is the hoop stress, both are in the Y direction.

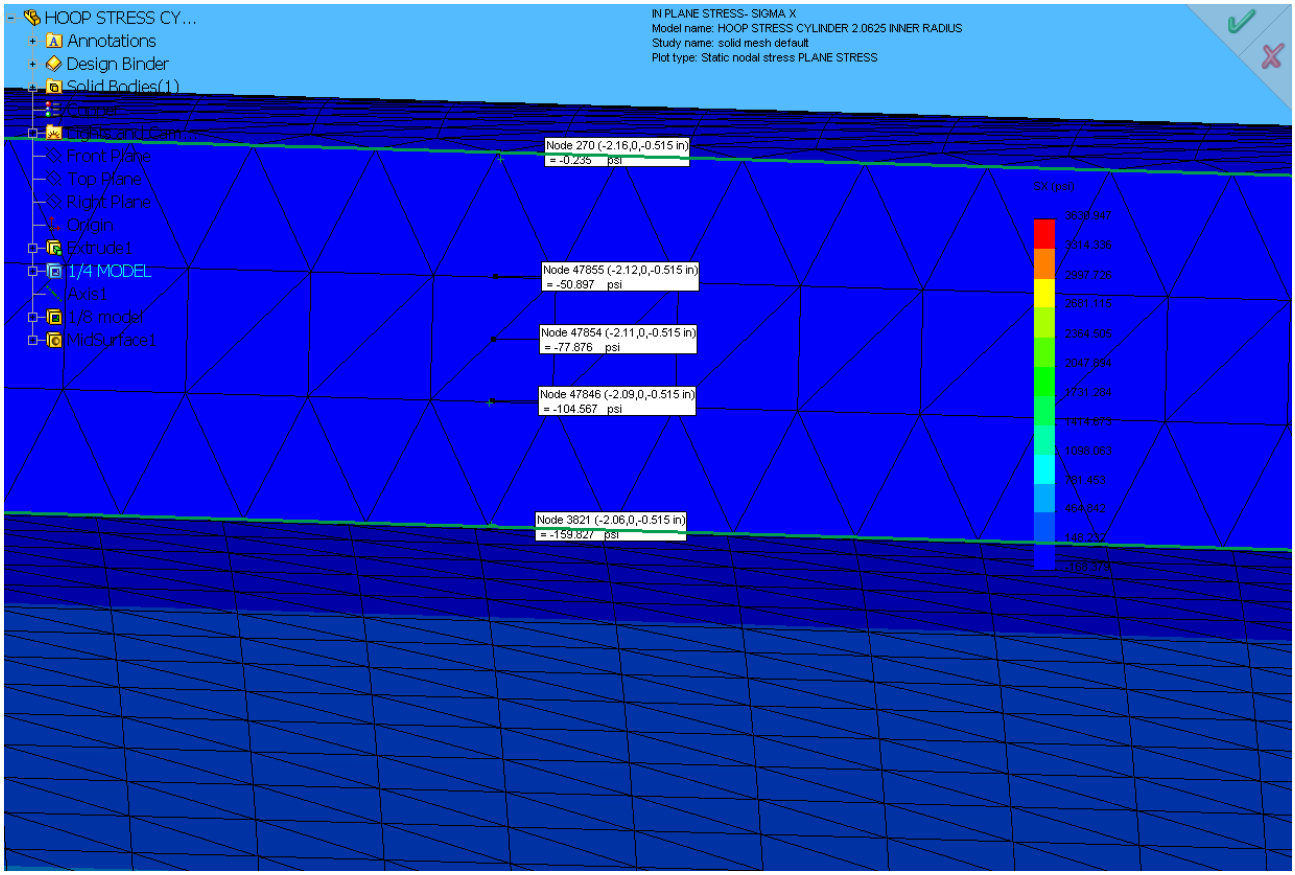


Figure 40: Probe results for the radial stress in rectangular coords.

As we can see from figure 40 the radial stress is for all intensive purposes equal to the internal applied pressure of 160 psi and is of a compressive nature due to the negative value. Also notice how the value thru the wall from the inside to the outside decreases and pretty much goes to zero, this is just what we expect to see. However, the color distribution in the plot does not represent the actual radial distribution of the stress thru the wall. To see that we have to redefine our plot again and put it in cylindrical coordinates to view this distribution.

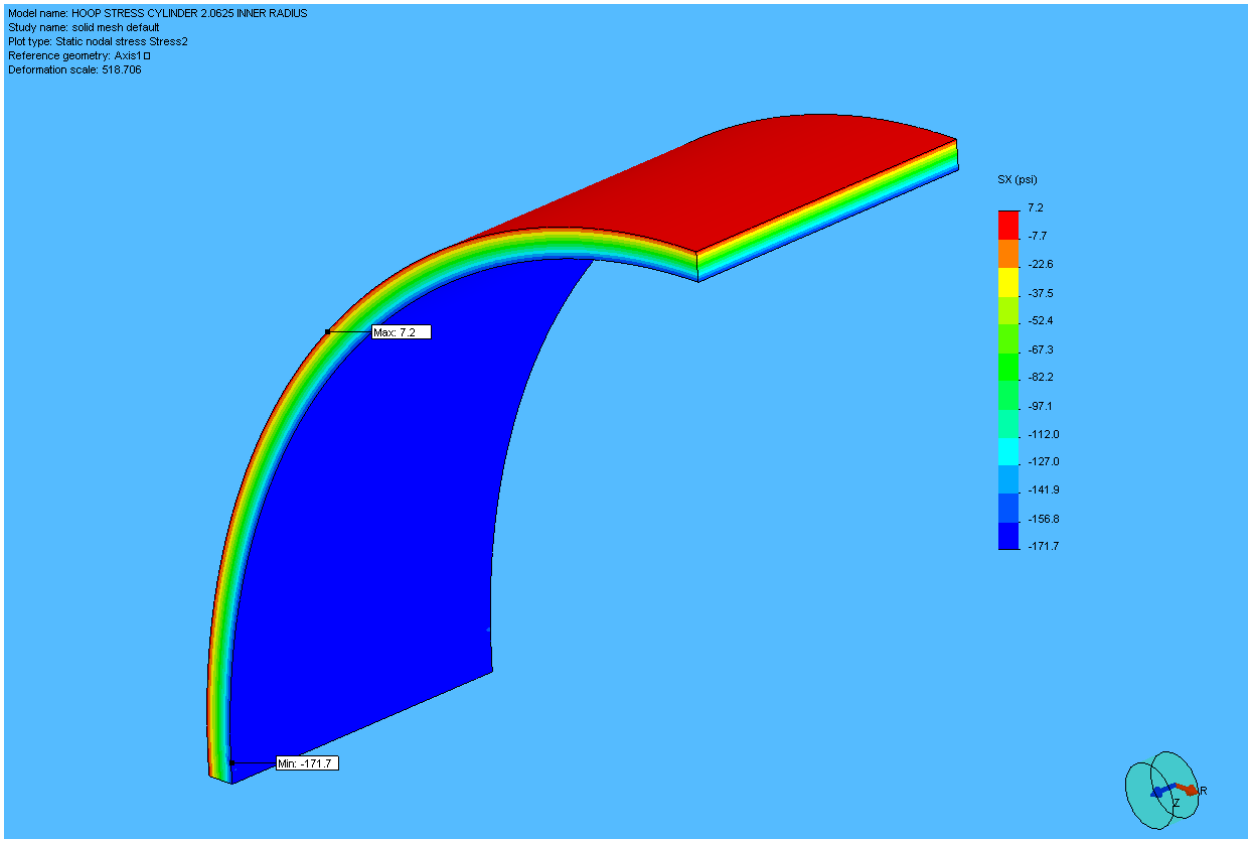


Figure 41: Radial Stress distribution in the cylinder (cylindrical coords).

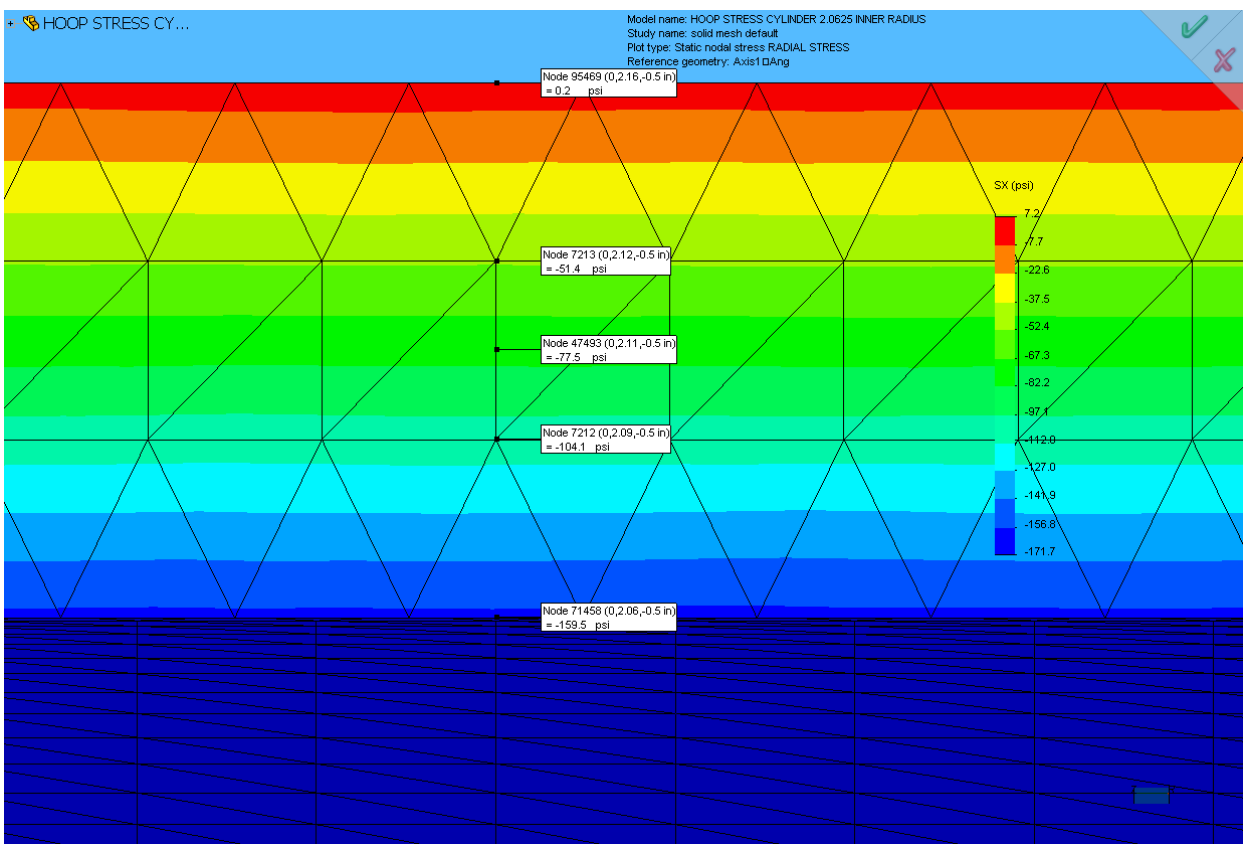


Figure 42: Probe results for the radial stress at the Top middle face in cylindrical coords.

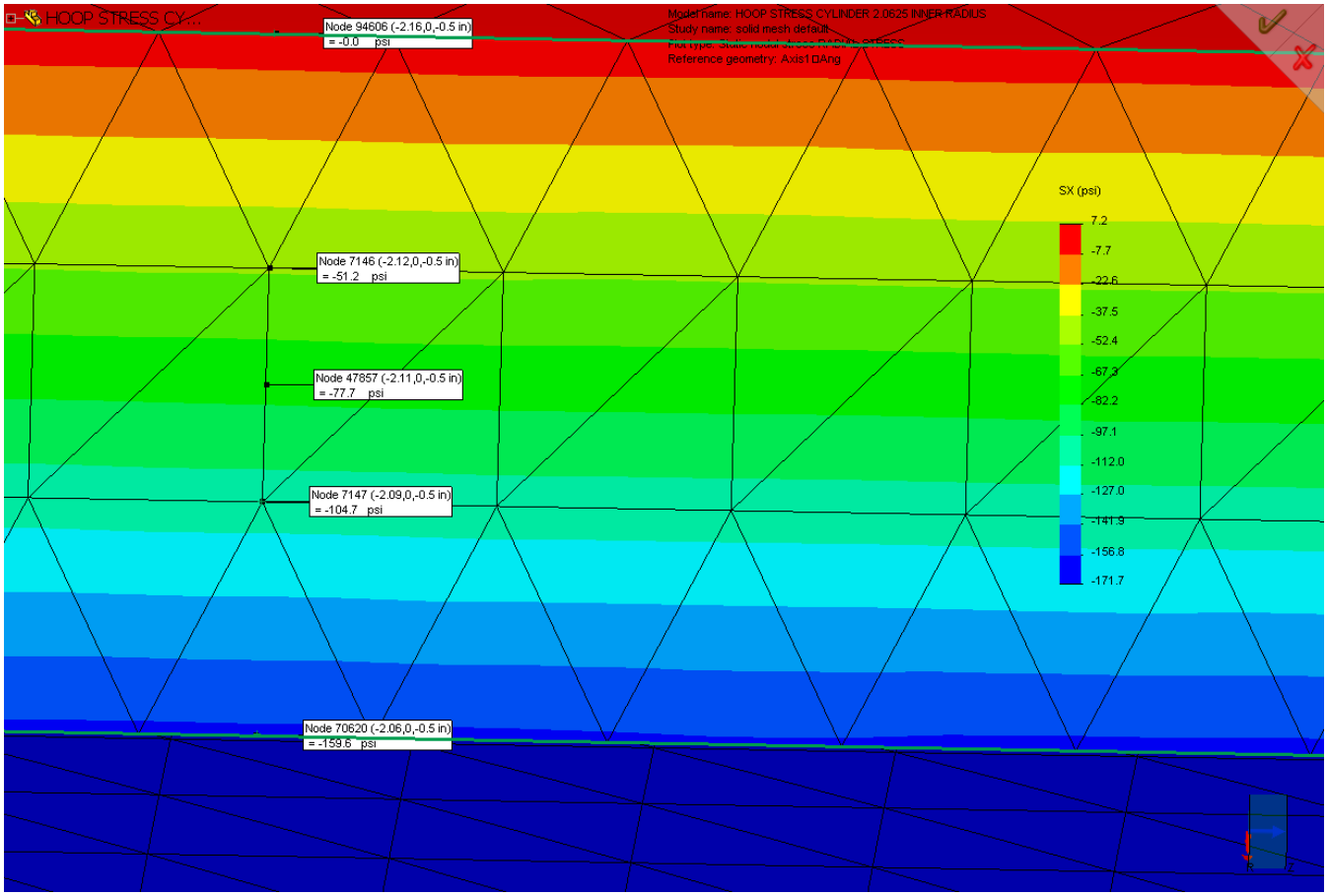


Figure 43: Probe results for the radial stress at the bottom middle face.

As expected, we see in figure 41 a relatively close match to the predicted values of the 160-psi compressive radial stress radiating outward, to a positive tensile stress or in effect to zero stress.

Some interesting observations can be made from figures 41, 42 and 43. If we look at figure 41 we notice we get a range of values from a max of 7.2 to a min of -171.7, but more importantly notice where the values are located, they are on the edge of the object. If we were to take these values and try and find our error percentage, we would get:

From *equation 5* for the percent error results we get:

For the inner radial stress
[(171.7 – 159.9)/ 171.7] * 100 = 6.872%

% error inner radial stress = 6.9% Acceptable

For the outer radial stress
[(7.2-0) / 7.2)] * 100 = 100%

% error outer radial stress = 100% Not Acceptable

100 % error in the outer radial stress, obviously not good. No matter how small I get that outer radial stress to be unless it is 0, I will always have a 100 % error here.
On the other hand if I somehow can get the outer radial stress to be 0, I will have no error at all, 0 %.
By all accounts I am in a catch 22, but not necessarily.

As I have shown before there is great value in probing for results and in this case it really rings true. Figure 42 shows the probe results for the top middle of the face going thru the wall thickness from the inside to the outside of the cylinder. If we look at the bottom probe box in figure 42 we see the node number 71458 and to the right of that are two other numbers, these are Y and Z coordinates with X coming out of the page. The Y coordinate is measured from the Z axis, it is our inner radius with a value of 2.0625 but the box is only so big so it truncates the number to 2.06. The Z value is .5 or half of my model. Below these numbers you see = -159.5 psi and this is our compressive radial stress, compressive because the value of the number is negative, if it were positive we would have tension.

So, why is this so important?
Between figures 41 and 43 notice how there is quite a bit of difference between the inner and outer values for the stress. In figure 41 the automatic measurements occur at the outer edge of the part where as when I probed for results in figure 42 & 43, I took them from the middle of the part.

I did this because of Saint Venant's Principal which states; uniform stress predicted by classic equations exist only in regions reasonably well removed from points of load application, support locations, or locations of geometric discontinuity.
External forces or internal reactions at these locations typically introduce significant localized effects. Therefore, in these regions, shortcomings of classical stress equations are at odds with the enhanced predictive capability of finite element results.

The results probed in figure 42 are really not that bad and the values of what we expect to see for the inside and outside radial stress are for all intensive purposes acceptable. However, knowing what we encountered before with our probed results for the hoop stress on the two opposite faces, I thought I just better check the other face to see what we get and wouldn't you know I got lucky and found my 0 value at the outside in figure 43.

From *equation 5* for the percent error results we get:

For the inner radial stress
[(159.6– 159.9)/ 159.6] * 100 =.18796 %

% error inner radial stress = .188% Acceptable

For the outer radial stress
[(0-0) / 0)] * 100 = 0% Yes 0/0 is undefined or indeterminate, but in our case our goal number was 0.

% error outer radial stress = 0% Acceptable

Once again I am quite impressed with those error numbers, especially the 0% we got for the outside radial stress.It looks to me like SolidWorks incorporates Saint Venant's Principal into its equations somehow, a good thing to know and a good thing to be aware of when one compares classic equations to SolidWorks analysis results.

I am not sure how well we will fair with a mid-surface shell analysis here; as I showed before with the hoop stress analysis our values will be at the mid-plane of the wall thickness, which worked great for the hoop stress. A mid-surface shell analysis will tell us nothing about the radial stress that we can verify. Since we are at the mid-plane we cannot verify the inner radial stress and for that matter we cannot verify the outer radial stress either. However, since we have encountered bending stresses in the other two analyses i.e. the hoop and the axial, it might just behoove us to check and see what's there.

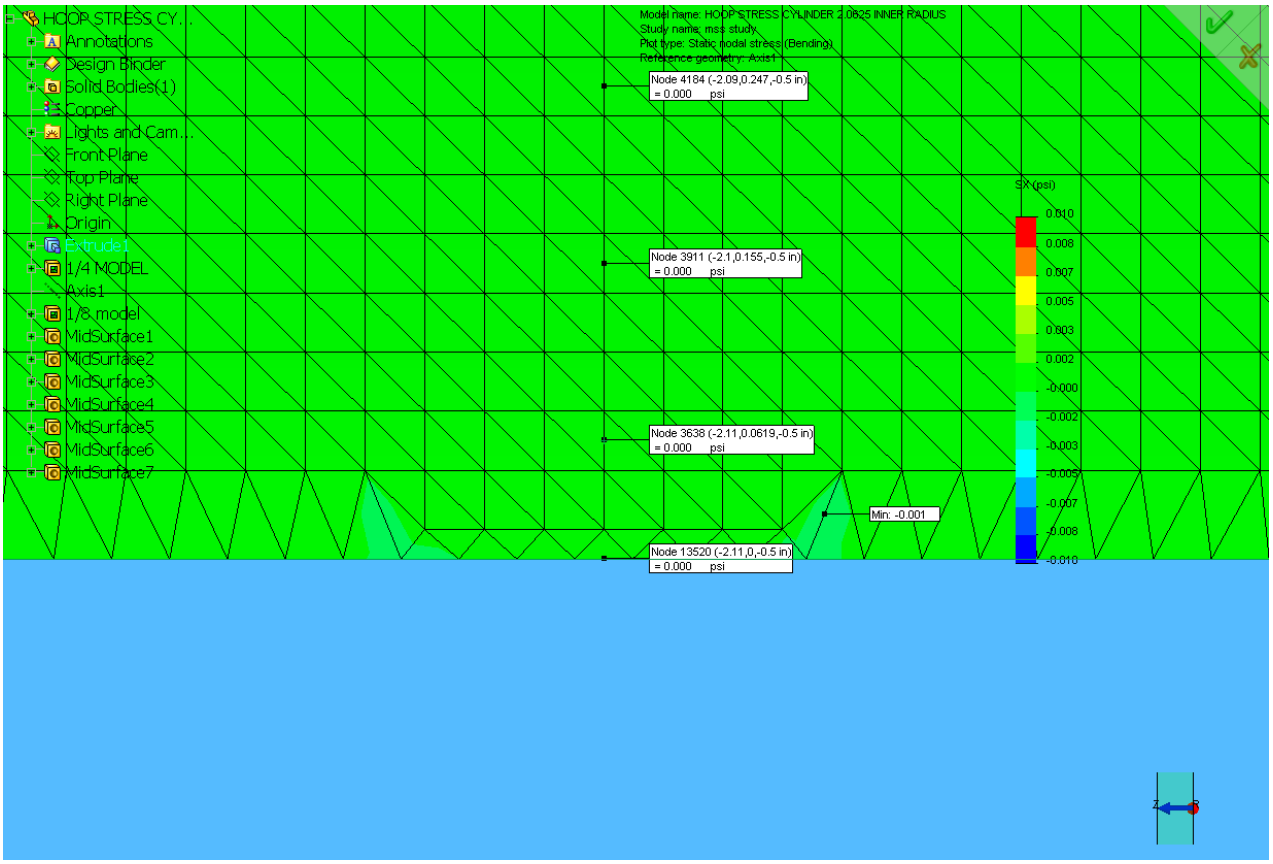


Figure 44: Probe results for bending stress in SX or radial direction, cylindrical coords.

From figure 44 we see that in our same spot where we have probed two other times we get a 0 value for the bending stress, the average listed value for the entire flat face was .000168 psi, so essentially 0.

Displacements, Poisson's Ratio and Hooke's Law, the beginning of the end

Let's see what we can learn from the displacements, as I mentioned before for an open cylinder with no ends we should have an axial stress of 0, but when we ran the analysis in SW with Saint Venant's Principal in hand, we got a value of 1284 psi for the solid elements and 1342.3 psi for the shell elements. I speculated then that the reason for this might be in the fact that I had to attach a 0 translation restraint in the Z direction (axial) on the back face of the cylinder to prevent rigid body motion, without this restraint the analysis fails.

Upon further investigation, I found a relationship between the X and Y stresses and the Z strain and then calculated an axial stress based on the axial strain, which gave us good numbers for the axial stress that SolidWorks produced.

So now let's see what x, y, z and radial displacements SolidWorks gives us.

Z displacement rectangular coordinates

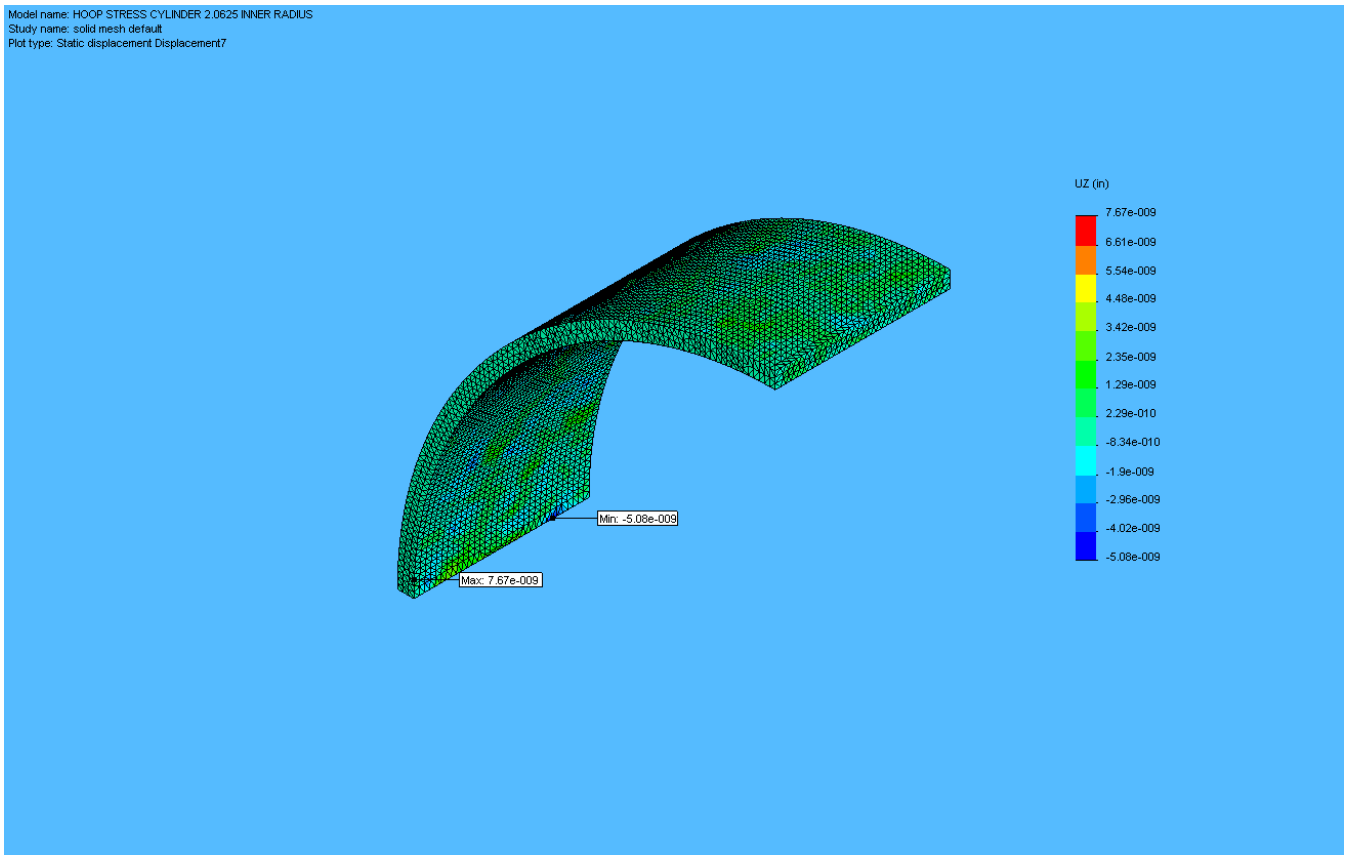


Figure 45: Z displacement rectangular coords with default chart settings.

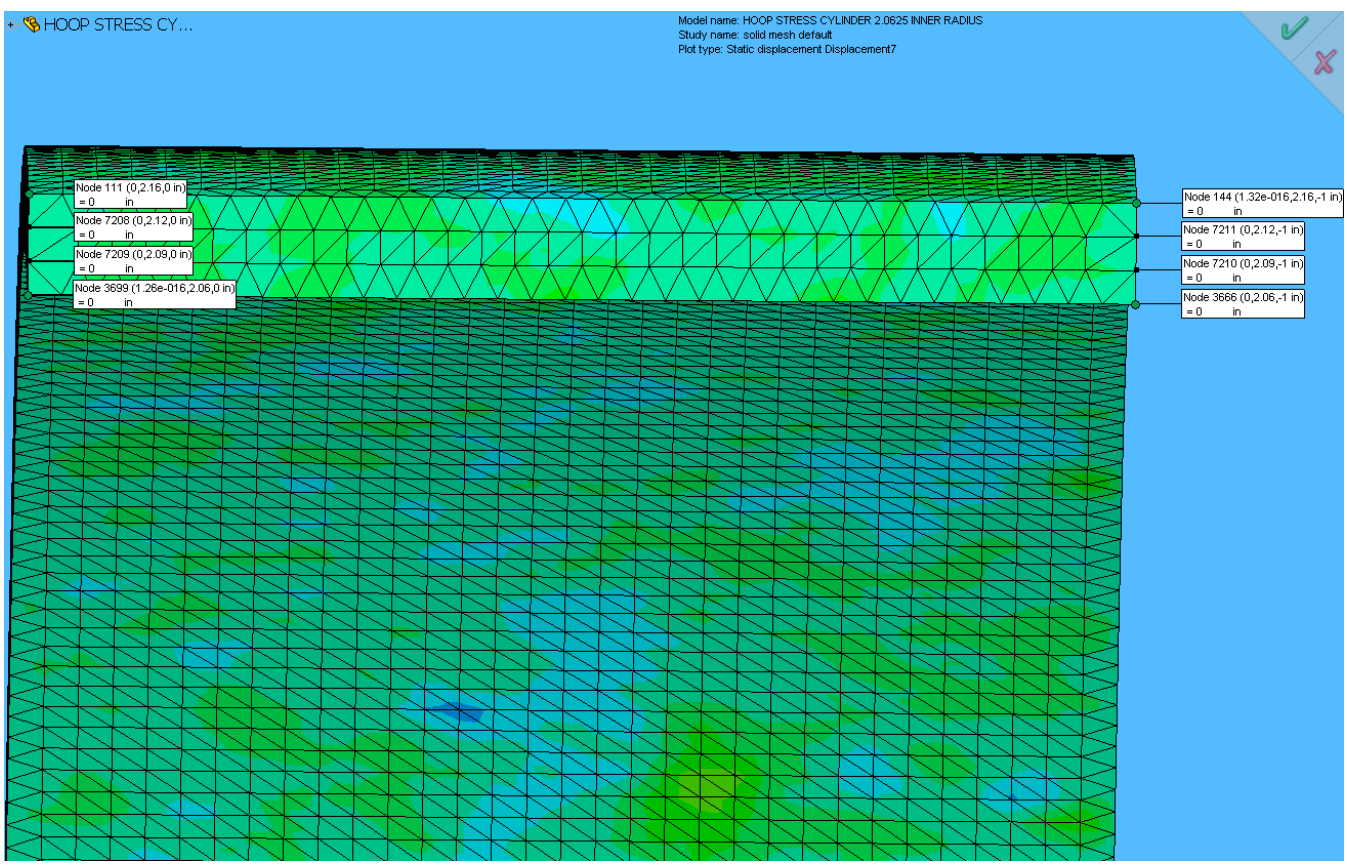


Figure 46: Probe results on the ends of the cylinder Z or axial displacement, rectangular coords.

No displacement on the ends of the cylinder in the axial or Z direction.

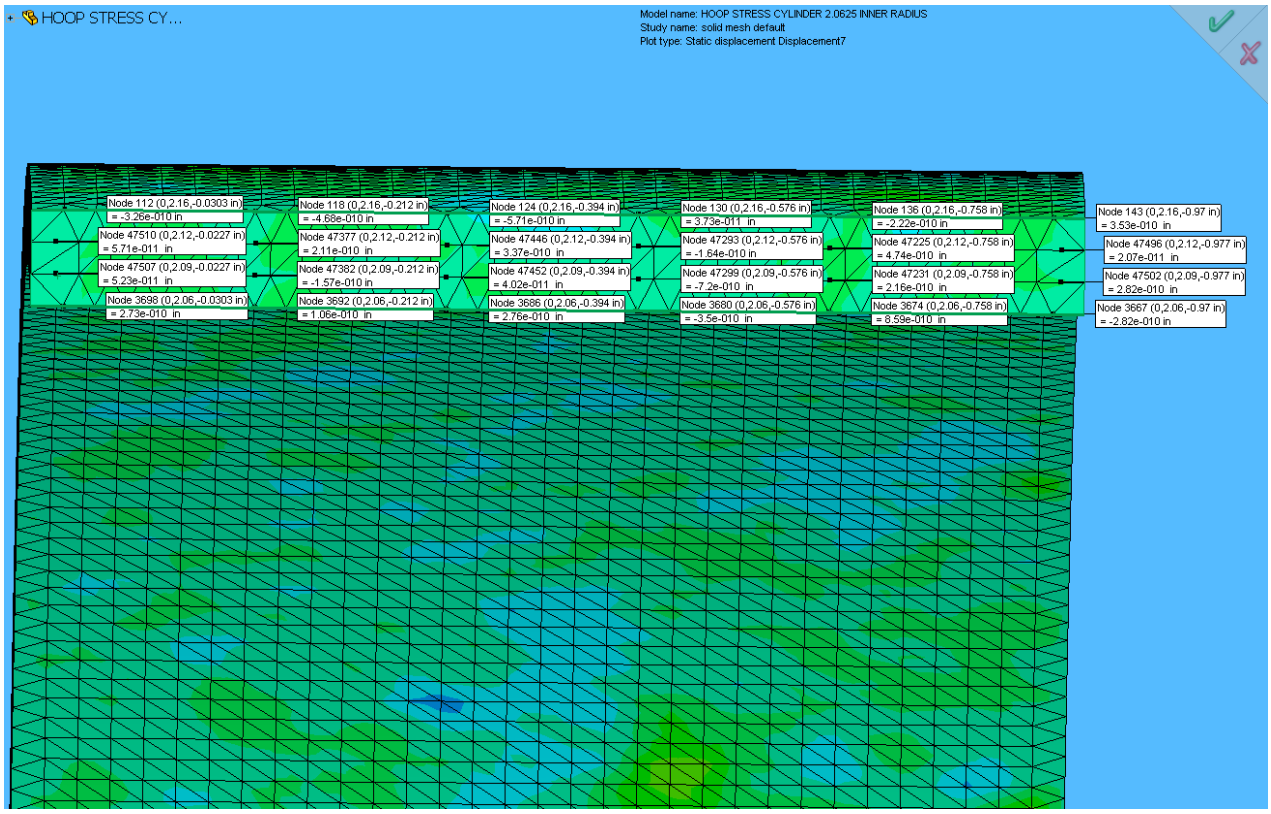


Figure 47: Z direction probe result inside the ends of the cylinder, rectangular coords.

Well there certainly seems to be a lot going on here. From figure 46 we see that there is no axial displacement on the very ends of the cylinder but then in figure 47 we see positive as well as negative displacements and notice how the positives and negatives are stacked from the inner wall to the outer wall. I am not sure what is really going on here, essentially these numbers are zero, most of the numbers you see have nine zeros in front of them some have 10, I think we can safely assume there is no movement in the axial direction. With a little chart manipulation we get figure 48.

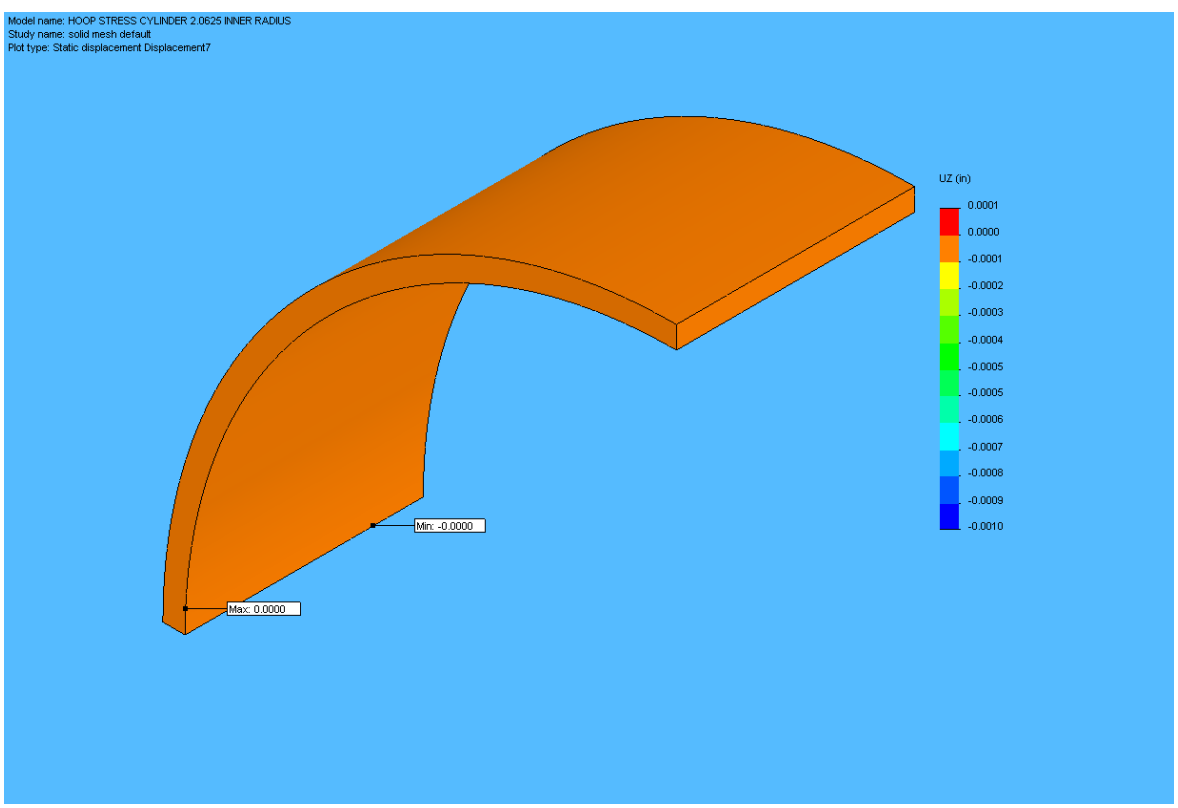


Figure 48: Axial displacement, rectangular coords.

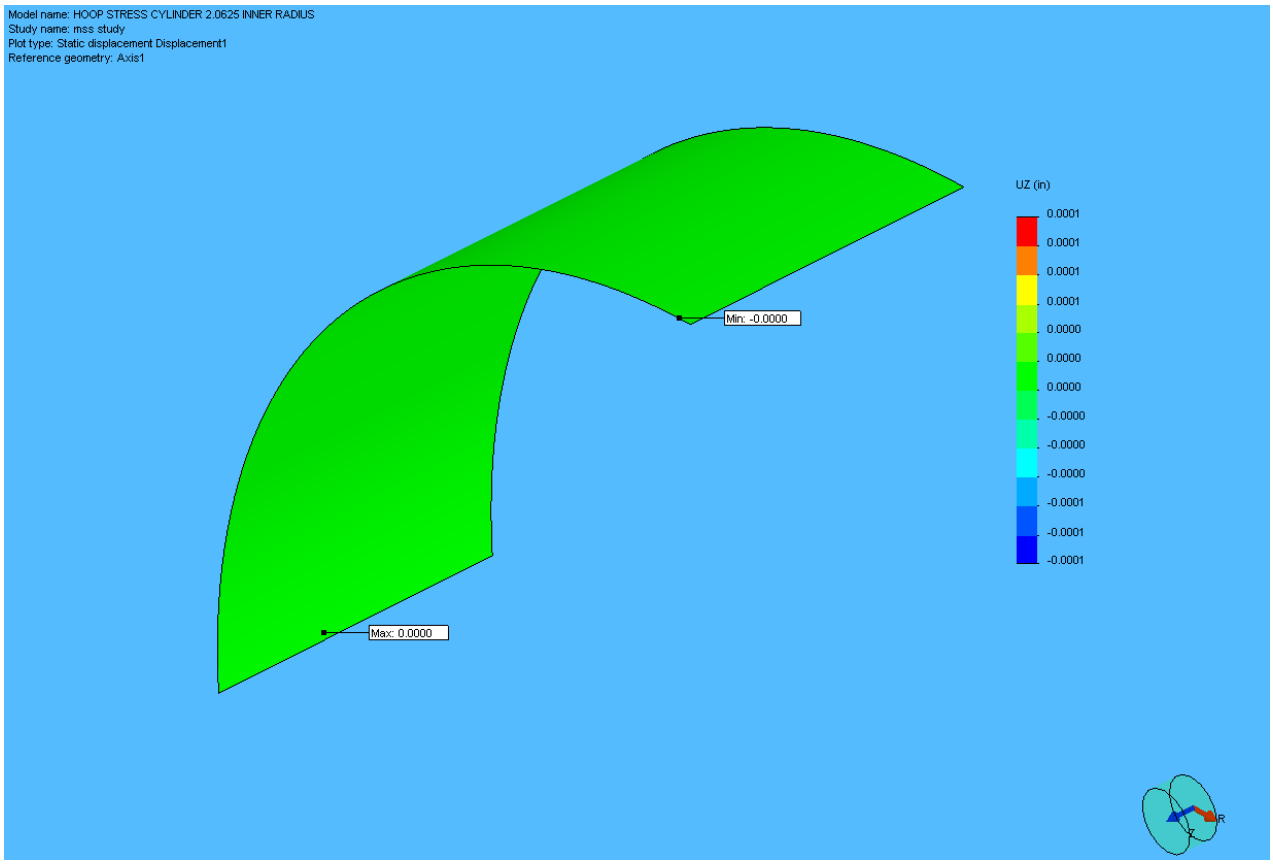


Figure 49: Axial displacement, cylindrical coords with shell element.

The axial displacement for the solid element flat face was 6.9222e-11, the listed average value for the axial shell displacement was 1.3411 X 10^{-9} inches or essentially zero.

Strain and Hooke's Law

Simple definition of strain can be expressed as a change in length divided by its original length.

Equation 9

Strain = (change in length) / (original length)
Or
$\varepsilon = \Delta L / L$

Hooke's Law **the law is named after 17th century British physicist Robert Hooke**.

A law stating that the stress applied to a material is proportional to the strain on that material.

Equation 10

Hooke's Law.
$\varepsilon = \sigma / E$
So

$$\varepsilon = \Delta L / L = \sigma / E$$

$$\sigma = \varepsilon E$$

I want to see if I can hit that axial displacement number with a quick calculation, but I can already tell I will not even come close. For an open cylinder we should theoretically have an axial stress of zero, and

a zero displacement, so if I put my zeros in the equations above I will get a value of zero, no worries there.

If we look at the last equation $\sigma = \varepsilon E$ and plug in the numbers we get; SW value for E = 15.954 e+6

-.00008229(calculated z strain from x-y stresses prior) * (15.954×10^{6}) = 1312.9 psi (our calculated axial stress from before)

Now if we look at $\varepsilon = \Delta L / L = \sigma / E$ and rearrange a little we get;

$$\Delta L_{axial} = \sigma_{axial} L / E$$

$$\Delta L_{axial} = 1312.9\ (2)\ (\text{length of the cylinder}) / (15.954 \times 10^{6}) = 1.646 \times 10^{-4}$$

$\Delta L_{axial} = .0001646$ or 1.646×10^{-4} is not even close to the SW value of 1.3411×10^{-9} for displacement.

Below in figure 50 is a table I created to show the axial values for the displacement, strain and stress values that SolidWorks gave me not only for the solid element study but for the shell study too.

			Table of SolidWorks Values for Axial Measurements				
		Solid Elements				Shell Elements	
	Displacement inches	Z-Strain Normal	Stress psi		Displacement inches	Strain	Stress psi
Curved face-Average Listed Value	0	1.36E-09	1284		1.34E-09	6.69E-11	1342.3
Front face Average Listed Value	6.92E-11	-1.17E-10	1284		N/A	N/A	N/A
Front face Mid-Point Probe	0	3.35E-09	1284		1.80E-09	1.99E-11	1342.5

	2nd Principal - E2	Principal Strains	2nd Principal - E2
Curved face-Average Listed Value	1.36E-09		3.71e-9
Front face Average Listed Value	-1.17E-10		N/A
Front face Mid-Point Probe	3.35E-09		-1.01E-11

Figure 50: Table of SolidWorks values for axial displacement, strain and stress for both solid and shell elements.

So now let's plug in a few numbers from the table and see what we get.
If I rearrange the equation again and try to get a number for the strain based on the SW displacement and stress values I get. For the solid elements I have to look at the front face average listed value;

$$\varepsilon = \Delta L / L = 6.92 \text{ X } 10^{-11} / 2(\text{length of the cylinder}) = 3.46 \text{ X } 10^{-11} \text{ (SW value = -1.17E-10)}$$

$$\text{But } \sigma / E = 1284 / (15.954 \text{ X } 10^{6}) = 8.048 \text{ X } 10^{-5}$$

Now let's look at the mid surface shell element values and see what happens.

$$\varepsilon = \Delta L / L = 1.34 \text{ X } 10^{-9} / 2(\text{length of the cylinder}) = 6.7 \text{ X } 10^{-10} \text{ (SW value = 6.69E-11)}$$

$$\text{But } \sigma / E = 1342.3 / (15.954 \text{ X } 10^{6}) = 8.413 \text{ X } 10^{-5}$$

And

$$\varepsilon = \Delta L / L = 1.80 \text{ X } 10^{-9} / 2(\text{length of the cylinder}) = 9.0 \text{ X } 10^{-10} \text{ (SW value = 1.99E-11)}$$

$$\text{But } \sigma / E = 1342.5 / (15.954 \text{ X } 10^{6}) = 8.415 \text{ X } 10^{-5}$$

And looking for a stress we get for the solid element;

$$\sigma = \varepsilon E = 1.36\text{e-}9 \text{ X } 15.954\text{e}6 = .0217 \text{ psi}$$

I won't even bother to do the other calculations as those would be the around the same value as this or less and one would be negative.

Looking at the principal strain, which is the 3rd principal strain or axial strain the numbers are quite different especially for the solid element values, in fact they are all the same and they are negative which indicates a contraction. Plugging in those numbers and calculating a stress we get;

$$\sigma = \varepsilon E = -1.17\text{e-}5 \text{ X } 15.954\text{e}6 = -186.7 \text{ psi}$$

The other values for the shell elements would be far less again, so I won't bother with those.

OK what's going on here, why don't the numbers correlate? The only thing I can think of here is what I have stated before when I calculated the z strain; a state of plane stress does not necessarily mean a state of plane strain. The axial strain/stress is induced by the hoop stress, that's where it comes from. SolidWorks has given me essentially zeros for the axial displacements and the strains, which is what we expect to see in an open cylinder, and as I stated before we should have an axial stress of zero. Zero displacement = zero strain = zero stress, so the stress should = 0.
The axial stress that SolidWorks has shown that is present is caused I think by the hoop stress; therefore I believe the strains are residual effects of this showing up in the z direction. Also there is one side of the cylinder that is restrained, this too could be the source or a contributor to the stress.

I am not positive about this and if anyone reading this knows better as to what is going on please enlighten us all.

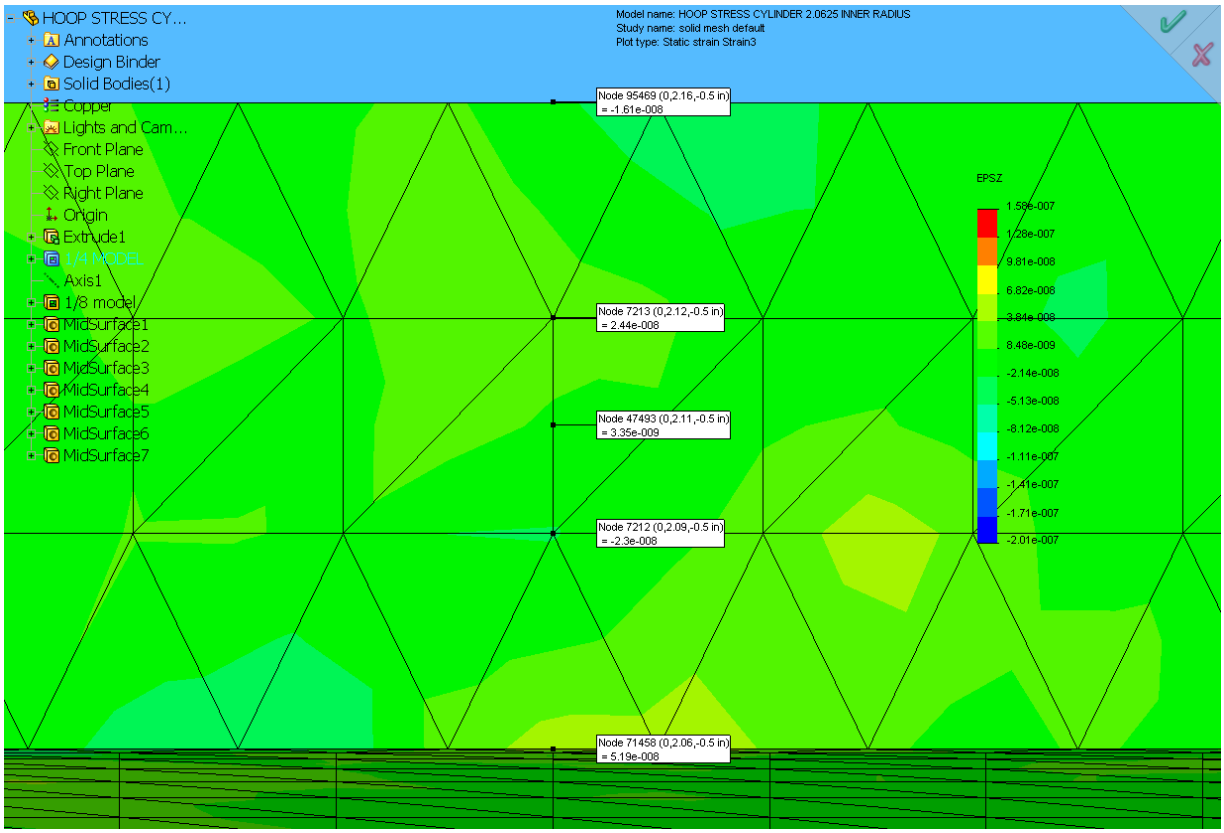


Figure 51: Probed strain value axial direction solid element mid-point, rect. Coords.

X and Y displacements in rectangular coordinates

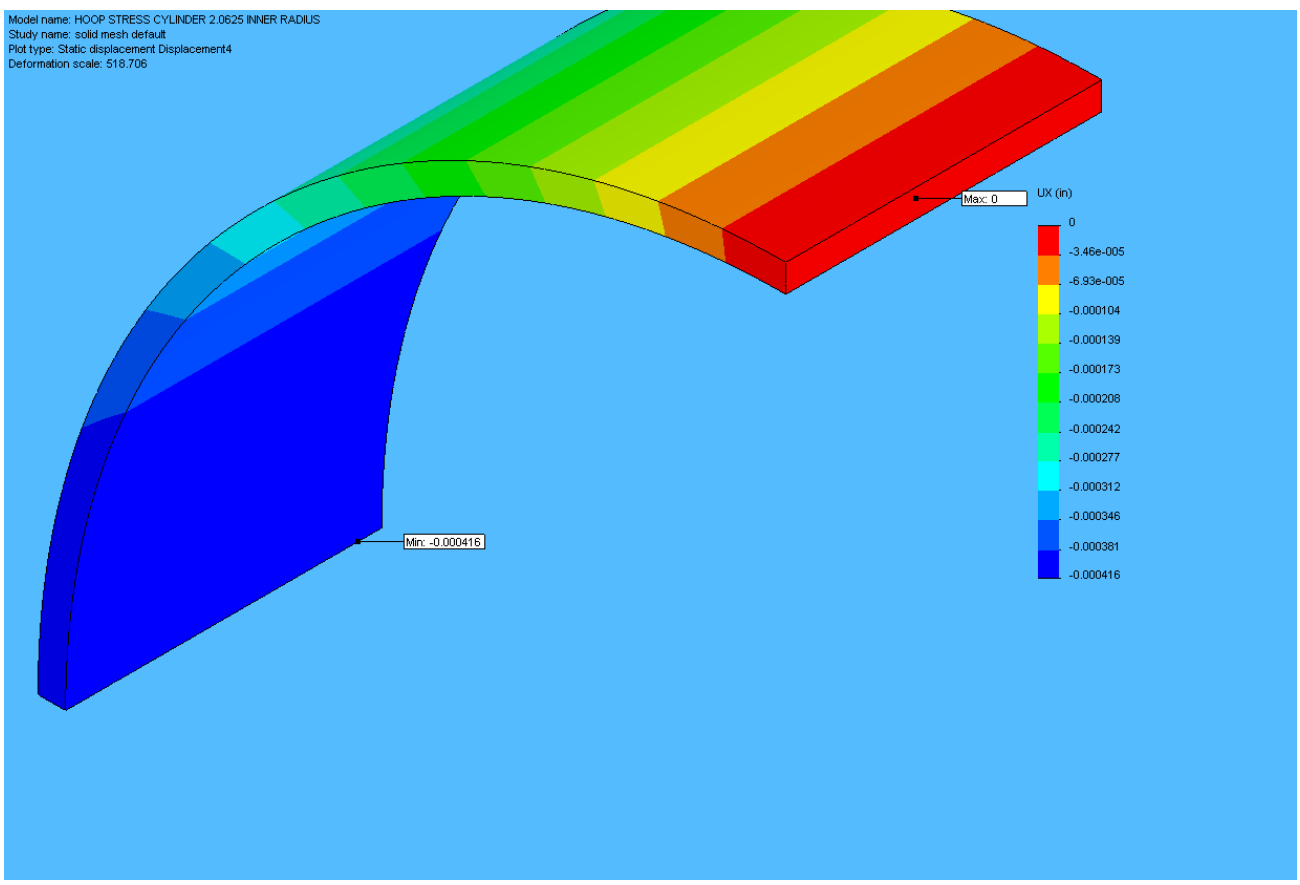


Figure 52: Displacement in X direction (UX) rectangular coords.

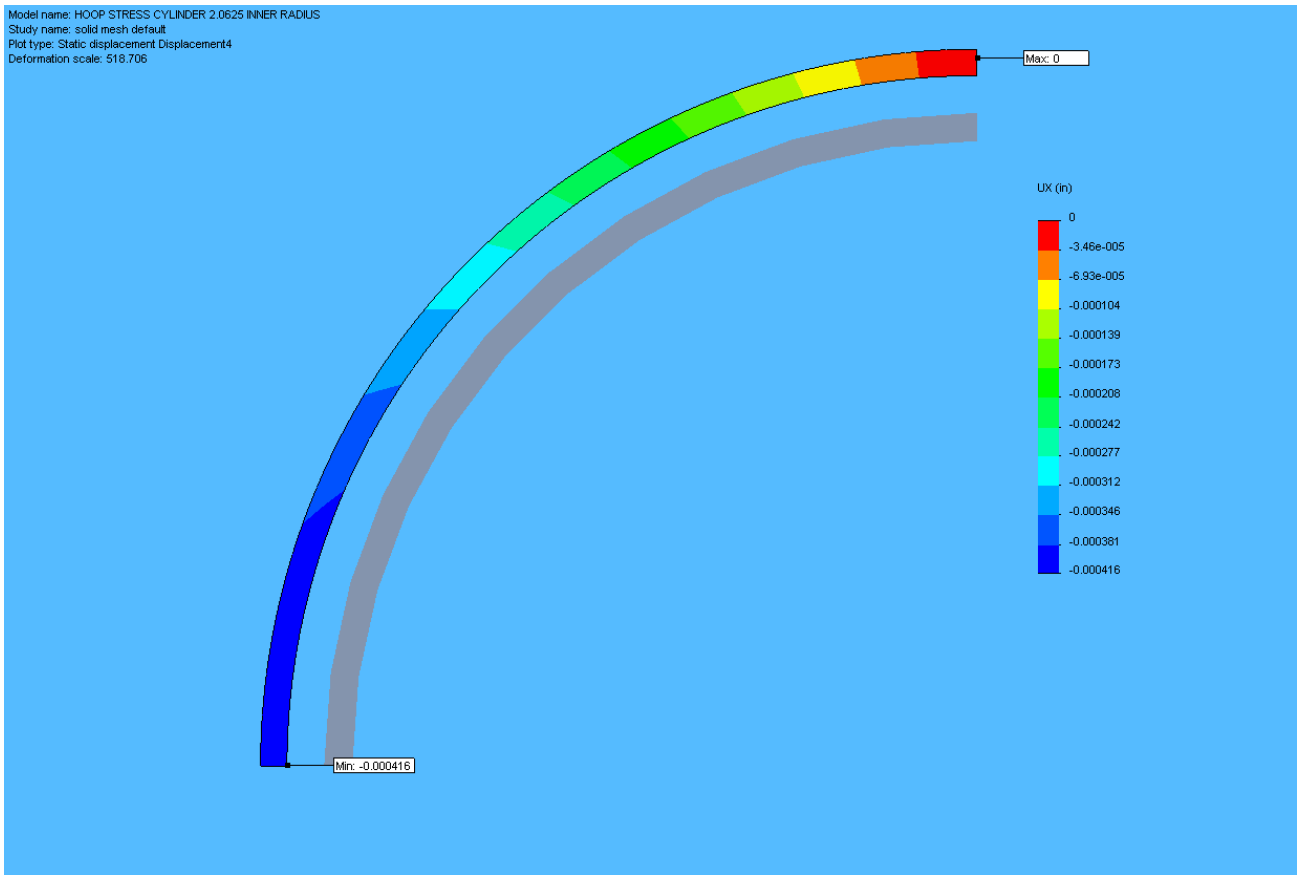


Figure 53: Front view of X direction displacement in rectangular coords with superimposed model on the deformed shape.

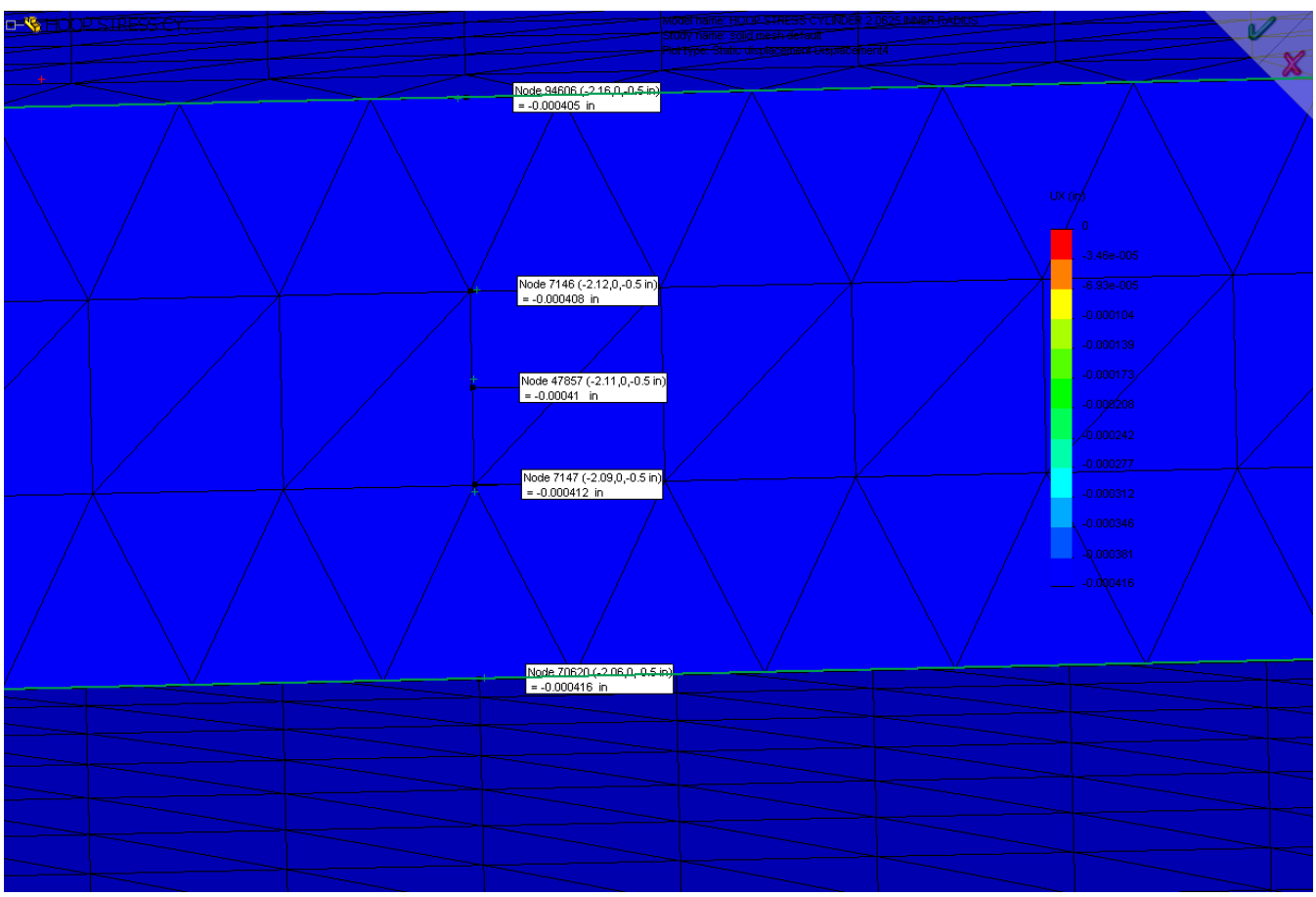


Figure 54: Probe results for the X displacement.

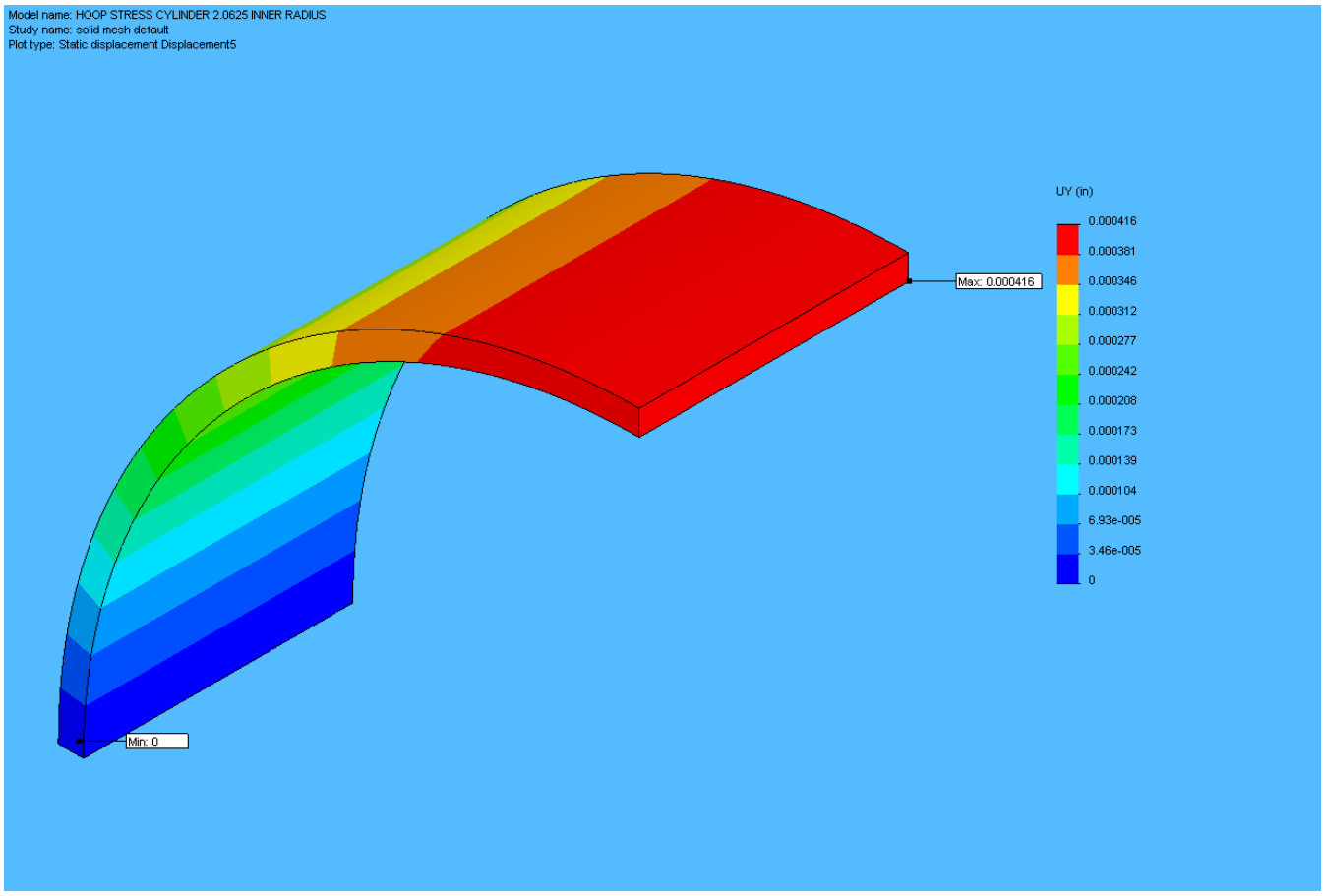


Figure 55: displacement in the Y direction, rectangular coords.

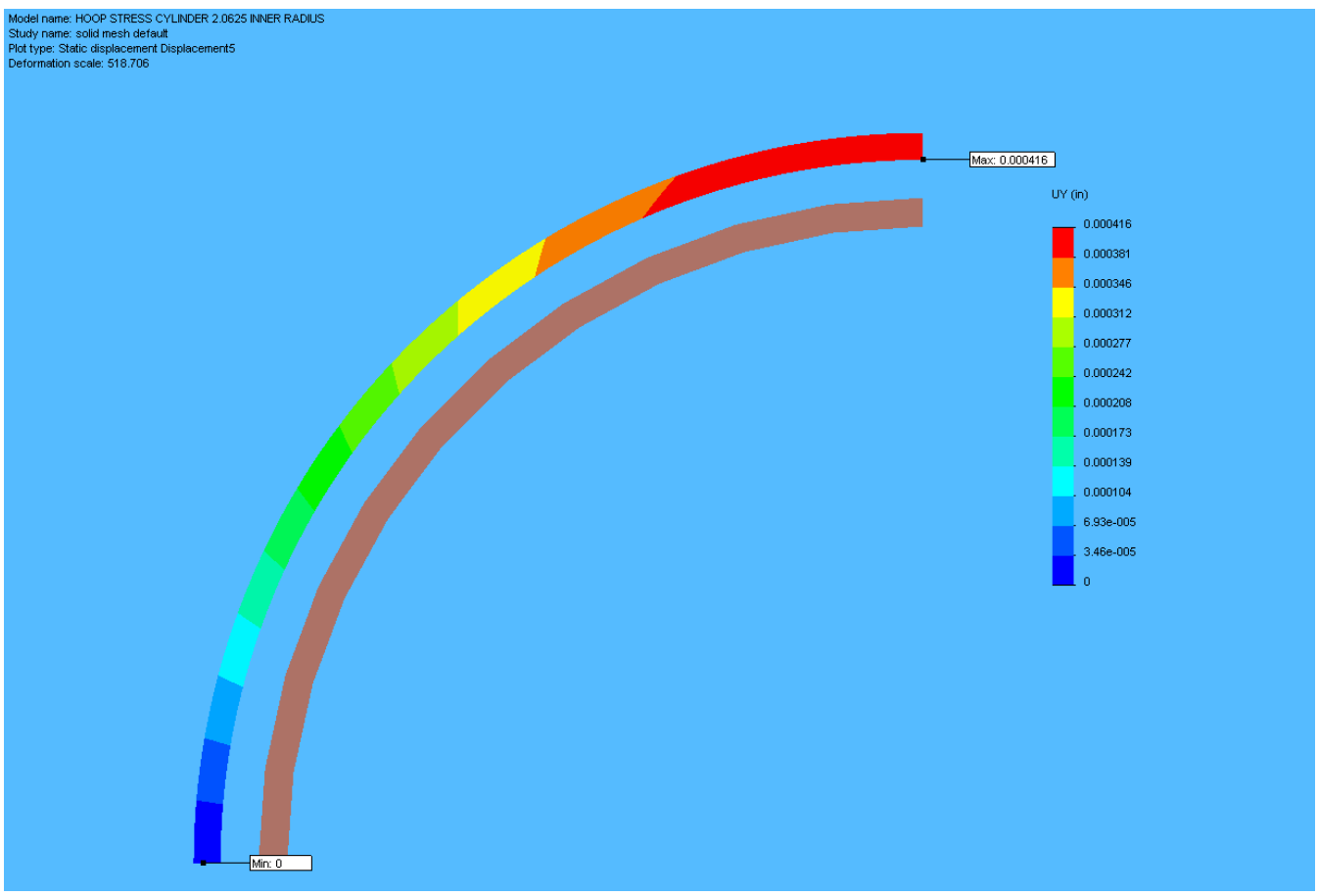


Figure 56: Y displacement with superimposed model on deformed shape.

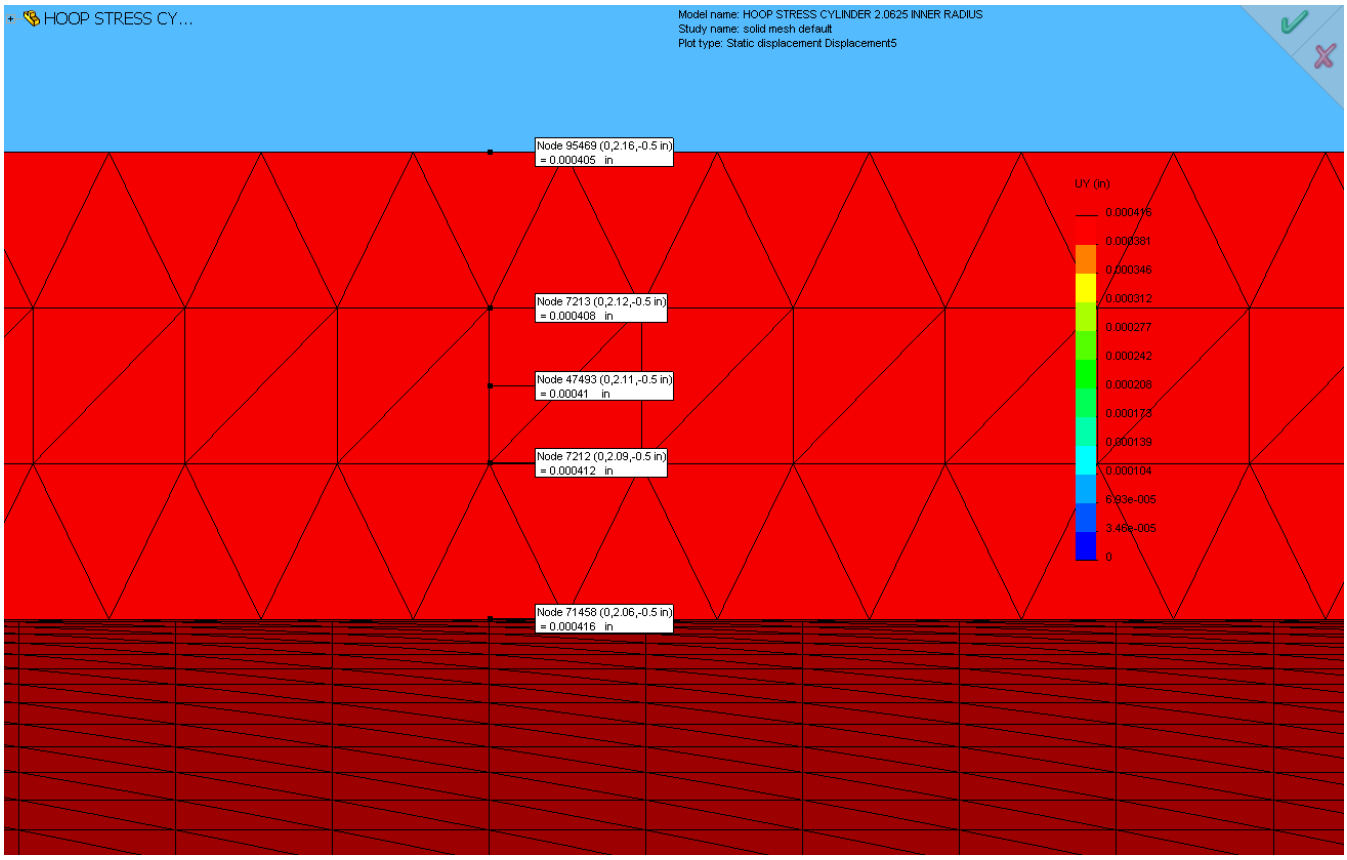


Figure 57: Probe results for Y displacement in rectangular coords.

Probe results between the X and Y displacements are exactly the same which is exactly what you would expect to see with asymmetric loading. The reason the X displacement is negative is due to the fact that the measurements for the displacement are on the left side of the axis thus inducing a negative number, the absolute values are exactly the same.

Another observation can be made here also and that is, since the displacements between the top and the bottom of the wall thickness are not the same number, (i.e. .000416 inches on the inner wall and .000405 on the outer wall) a compression has occurred in the wall thickness. Subtracting the two displacement values we get .000011 inches. If no compression had presented itself our top and bottom values for the displacements at the inner and outer wall would have been the same.

Figures 52 thru 57 represent the x and y displacements in the x-y plane and as you will notice they are exactly the same .000416 inches as is the contraction, which is what one would expect in a cylinder pressurized on the inside, a nice uniform expansion of the cylinder.

Strain, Poisson and Hooke

If we look at these displacements in the x-y plane we can note that classical plane strain is equal to the change in length divided by the original length or;

Equation 9

Simple definition of strain can be expressed as;

Strain = (change in length) / (original length)
Or
$\varepsilon = \Delta L / L$

This means that both strains are equal in the x-y plane since both our displacements are equal as well.

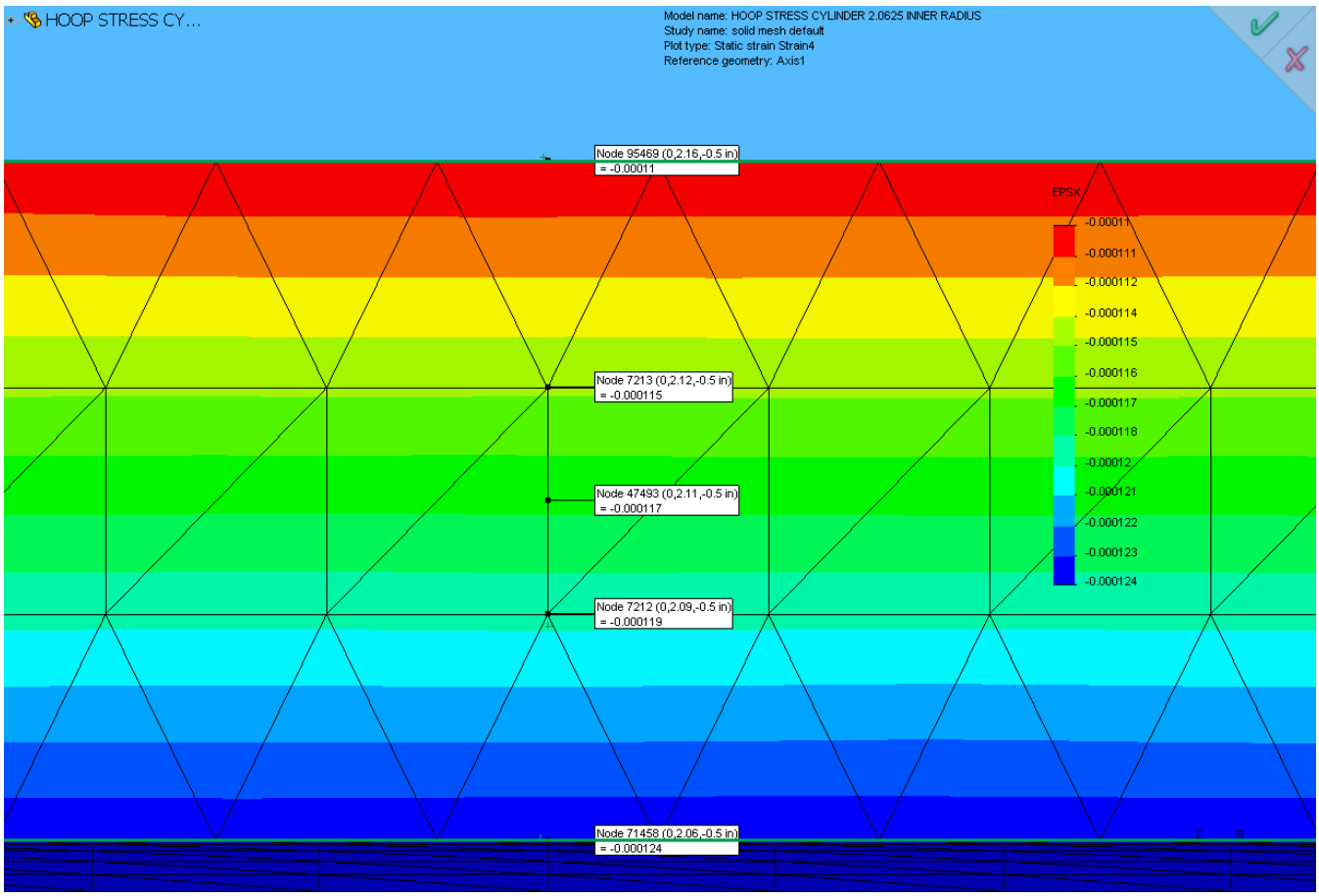


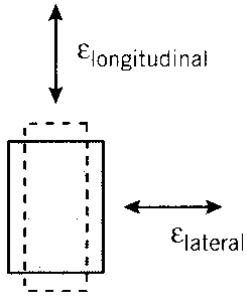
Figure 58; Strain in the x direction, rectangular coords.

Recall Poisson's ratio, which states;

Poisson's ratio is the ratio of transverse contraction strain to longitudinal extension strain in the direction of stretching force. Tensile deformation is considered positive and compressive deformation is considered negative. The definition of Poisson's ratio contains a minus sign so that normal materials have a positive ratio. Poisson's ratio, also called the Poisson coefficient, or coefficient de Poisson, is usually represented as a lower case Greek nu, ν.

The Poisson effect and is defined by;
Named after the French mathematician Simeon Denis Poisson, (1781–1840).

Equation 10



$\nu = - \varepsilon_{lateral} / \varepsilon_{longitudinal}$

Figure 59: The Poisson effect.

Poisson's ratio for Copper is approximately .37, that is $\nu_{copper} = .37$

With a calculated value for the plain strains based on our plane displacements, which were equal, we will never get to a correct value of .37 for the Poisson ratio in the x-y plane. Our value will always be 1. If we look at trying to get a good stress value by Hooke's Law, we will only get one value. If we go thru the motions and calculate a displacement based on our calculated stress we get:

Equation 11

Hooke's Law. **The law is named after 17th century British physicist Robert Hooke**.

$\varepsilon = \sigma / E$

And noting that $\varepsilon = \Delta L / L$ we can write;

$\Delta L / L = \sigma / E$ = or $\Delta L = \sigma L / E$ with L = to our radius of 2.0625

Where;

E equals the Modulus of Elasticity for the giving material. For Copper SW uses $E = 15.954 \text{ X } 10^6$ psi.

$\Delta L = (3548.4 * 2.0625) / 15.954e6 = .0004587$

Our SW value for the x displacement at the inner part of the cylinder is .000416.

The error here for the x or y displacement from equation 5 and the probe result from figure 54 or 57 is:

% error = ((.000416 - .0004587)/ .000416) * 100 = 10.264

%error for $\Delta L_{x\text{-}y}$ =10.26% Not Acceptable

If we look at the meridian values for shell elements we get, with r = 2.109 and sigma H = 3628.4 we get:

$\Delta L = (3628 * 2.109) / 15.954e6 = .0004796$

The value for x-y displacement for the shell element is .000414

% error = ((.000414 - .0004796)/ .000414) * 100 = 15.845

%error for $\Delta L_{x\text{-}y}$ =15.85% Not Acceptable

What is going on here? Number one we have radial displacement as well as radial strain. Poisson's ratio is defined by $\nu = - \varepsilon_{lateral} / \varepsilon_{longitudinal}$. We do not have that condition here. If we look in the x-y plane we have an expansion of the material and a contraction of the material along the same axis we do not have a lateral/longitudinal condition, we have a radial condition.

For two-dimensional (2D) objects, the deformation is not limited to lengthening or shortening in one direction. A 2D object can lengthen or shorten along the x or y axis (normal strain) and can also distort (shear strain) by the relative displacement of the upper to the lower border or the right border to the left border. Thus, in two dimensions, strain has four components, two normal strains, and two shear strains.

More complex is the deformation of three-dimensional (3D) objects such as in this case there are three normal strains (along the x, y (in plane) and z-axes (out of plane)) and six shear strains. To completely define the deformation of this 3D object, all nine-strain components must be defined, that means a general 3 dimensional stress calculation with matrices, stress transformation equations, strain transformation equations and everything else involved. Those calculations would be better suited for someone other than me. I had one class in advanced strengths about 12 or 13 years ago and it was the hardest class I have ever had, not to mention I have not done anything like that for many years.

Since most all of our calculations for the stresses so far have validity I shall accept the displacement values SolidWorks has given me. Remember, SolidWorks calculates displacements first, then strains and finally stresses, so based on our very low stress errors I am very inclined to accept what SW has given for displacements.

The relationship that SolidWorks uses is complex and after a little digging around in SolidWorks for formulas, I ran across this:

Isotropic Stress-Strain Relations
The most general form of the isotropic stress-strain relations including thermal effects is shown below:

In order to define the isotropic elastic properties, you must define the elastic modulus E_x. The program assumes a value of 0.0 for Poisson's ratio ν_{xy}, if no specific value is specified. A common value for the Poisson's ratio is 0.3. The shear modulus G_{xy}, is calculated internally by the program even if it is explicitly specified.

$$\boxed{E = 2G(1+\nu)}$$

The stiffness matrix for an isotropic material contains only two independent coefficients. The following sections describe the isotropic stress-strain relations in two- and three-dimensions including the effect of thermal strains.

Isotropic Stress-Strain Relations

The most general form of the isotropic stress-strain relations including thermal effects is shown below:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_o) \begin{Bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \\ 0 \\ 0 \end{Bmatrix} \right)$$

Figure 60: Stress-Strain relationship that SolidWorks uses.

Figure 60 is a 6 X 6 matrix for the relations between stresses – strains; in addition to that, there are stress transformation equations as well as strain transformation equations that would also be used to get to a final answer. Figure 61 shows a simple 2 x 3 by 3 x 1-matrix multiplication so you get an understanding of what is involved with the equation in figure 60.

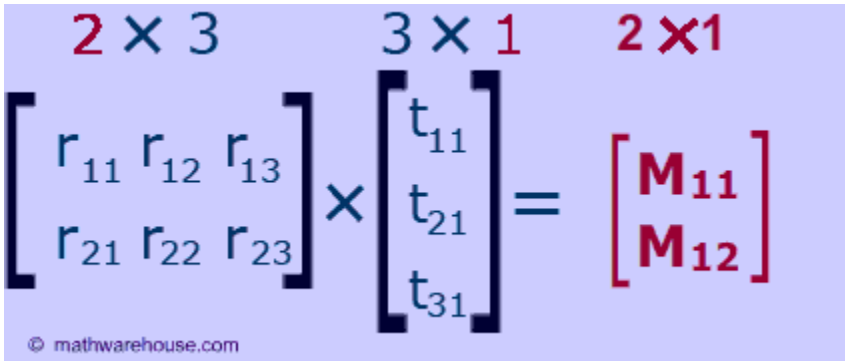


Figure 61: Simple matrix operation

Here is how we get M_{11} and M_{12} in the product.

$M_{11} = r_{11}\times t_{11} + r_{12}\times t_{21} + r_{13}\times t_{31}$
$M_{12} = r_{21}\times t_{11} + r_{22}\times t_{21} + r_{23}\times t_{31}$

One area in which Poisson's effect has a considerable influence is in pressurized pipe flow. When the air or liquid inside a pipe is highly pressurized it exerts a uniform force on the inside of the pipe, resulting in a radial stress within the pipe material. Due to Poisson's effect, this radial stress will cause the pipe to slightly increase in diameter and decrease in length. The decrease in length, in particular, can have a noticeable effect upon the pipe joints, as the effect will accumulate for each section of pipe joined in series. A restrained joint may be pulled apart or otherwise prone to failure.

Obviously, my attempt at doing 2D calculations for displacements and strains, by far missed the mark. There is a lot going on in this part, it is in a state of complex stress, and as such, there should be 3D calculations done to verify the displacements and the strains.

With that said there must be some heavy-duty calculations going on for the displacements and the strains. Recall that SW calculates the displacements first then the strains and finally the stresses. With such low errors on the stresses between SW and my 2D calculations, I have high confidence that the displacements and the strains are correct because of the order of operations that SW uses.

Radial Displacement

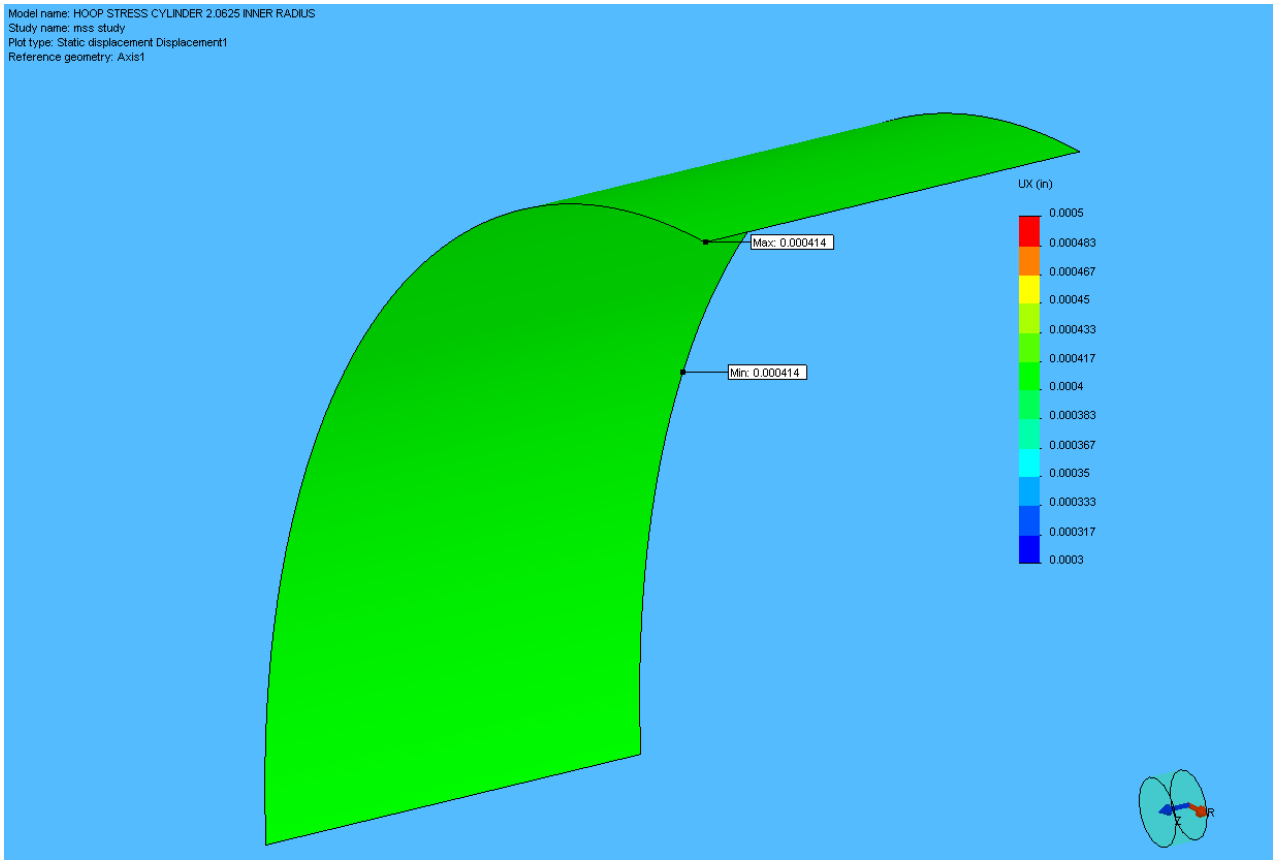


Figure 62: Radial displacement, cylindrical coords with shell element.

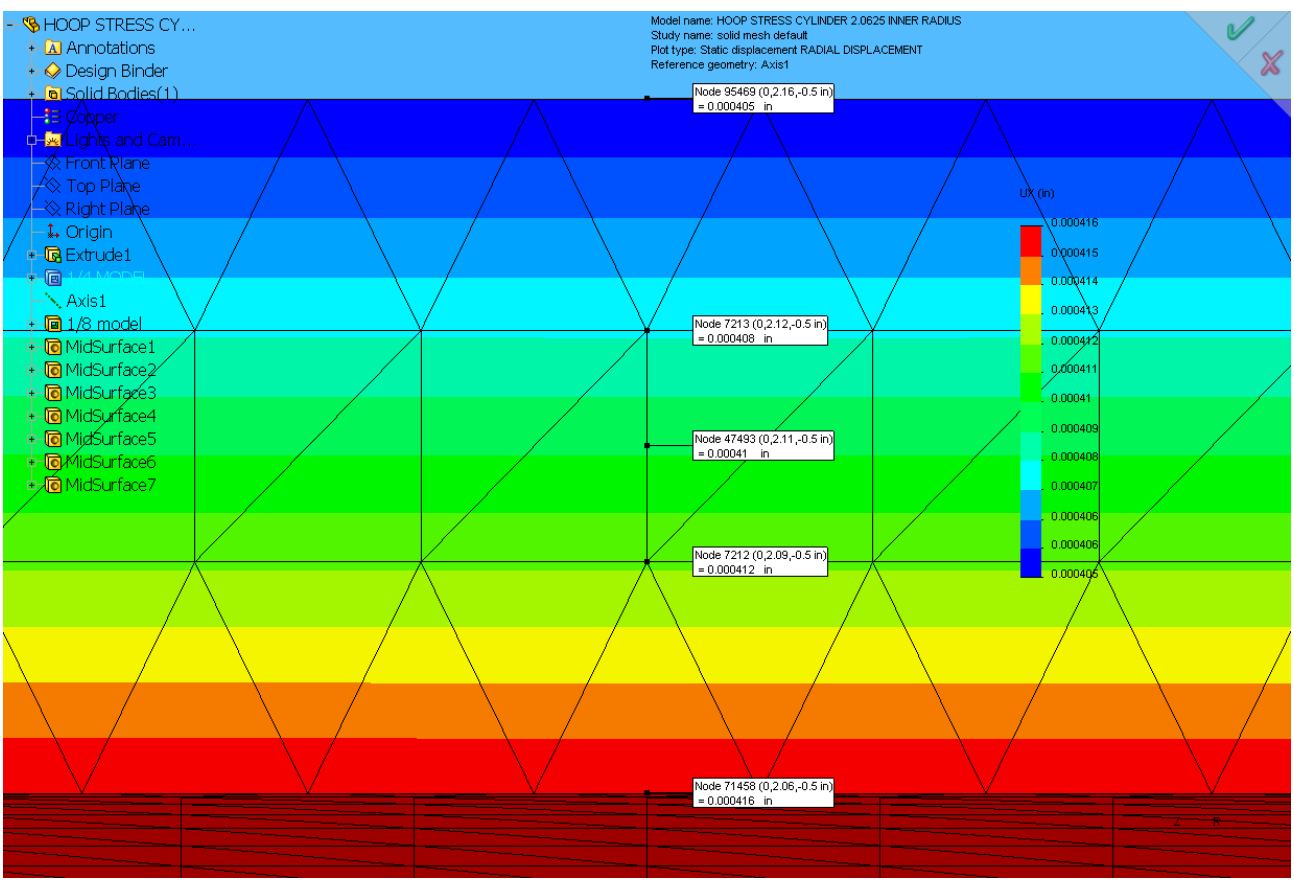


Figure 63: Probe results for radial displacement (cyl. coords) solid element

The radial displacements had differences as well. With the shell elements, we have a radial displacement of .000414 inches at the membrane (figure 62) and the solid element probe at the middle was .000410 inches (figure 63) this may be due again to the different radius value used in the calculations as I mentioned before. All in all though, a .000004 inch variation in radial displacement between the solid and shell elements is not that bad and we have seen other variations between the elements before.

<u>More proof for a 3D stress calculation</u>

When a pressure vessel has open ends, such as with a pipe connecting one chamber with another, there will be no axial stress since there are no end caps for the fluid to push against. Then only the hoop stress $\sigma_H = pr_i/t$ exists, and the corresponding hoop strain is governed by Hooke's Law for linear elastic deformation as:

<u>Equation 11</u> *<u>Hooke's Law</u>*. **The law is named after 17th century British physicist Robert Hooke**.

$$\varepsilon_\Phi = \sigma_\theta / E = pr_i/tE$$

Where;
E equals the Modulus of Elasticity for the giving material. For Copper SW uses $E = 15.954 \text{ X } 10^6$ psi.

Since this strain is the change in circumference δ_C divided by the original circumference $C = 2\pi r_i$ we can write:

$$\delta_C = C\ \varepsilon_\Phi = (2\pi r_i)(\ pr_i/tE)$$

The change in the circumference and the corresponding change in the radius δ_r are related by;

$$\delta_r = \ \delta_C / 2\pi$$

So now the corresponding radial expansion equation is;

$$\boxed{\delta_r = pr^2 / tE}$$

For the solid elements at the inner boundary of the cylinder where $r = r_i = 2.0625$.

At the inner radius $\boxed{\delta_{ri} = (160 * 2.0625^2) / (.093 * (15.954 \text{ X } 10^6))}$

$$\delta_{ri} = .00045873 \text{ inches}$$

The error here for the radial displacement from equation 5 and the probe result from figure 63 is;

$$\% \text{ error} = ((.000416 - .00045873)/ .000416) * 100 = 10.2716\%$$

%error for δ_{ri} =10.27% Not Acceptable

At the meridian boundary of the cylinder where $r = r_M = 2.109$. Looking at the shell elements we get;

At the meridian radius $\boxed{\delta_M = (160 * 2.109^2) / (.093 * (15.954 \text{ X } 10^6))}$

$$\delta_M = .00047965 \text{ inches}$$

The error here for the radial displacement from equation 5 and the average listed value for the shell membrane from figure 54 is;

$$\% \text{ error} = ((.000414 - .0004797)/ .000414) * 100 = 15.8695$$

%error for δ_M =15.87% Not Acceptable

Not very good results in error above, I did however have a thought and that was to see what the values would calculate to be if the ends of the cylinder were capped.

For uniform internal pressure the formula from the Roark's hand book of stress and strain equations for a thin wall cylinder with end caps is;

$$\Delta R = (pR^2 / Et) (1-\nu/2)$$

For the solid element radius value of 2.0625 we get;

$$\Delta R = ((160 * 2.0625^2) / (15.954e6 * .093)) * (1 - .37/2) = .00037386$$

$$\Delta R = .0003739$$

$$\% \text{ error} = ((.000410 - .0003739)/ .000410) * 100 = .08805$$

%error for ΔR =8.81% Not Acceptable

For the shell element radius value of 2.109 we get;

$$\Delta R = ((160 * 2.109^2) / (15.954e6 * .093)) * (1 - .37/2) = .0003909$$

$$\Delta R = .0003909$$

The average listed value for the radial displacement of the shell element face at the membrane is .000414

$$\% \text{ error} = ((.000414 - .000391)/ .000414) * 100 = .05555$$

%error for ΔR =5.56% Kind of Acceptable

Well, I really cannot explain this other than the fact that I did have to restrain the one side of the cylinder to prevent rigid body motion and a complex stress - strain field. Again that calculation above was a 2D not 3D.

Hoop Displacement

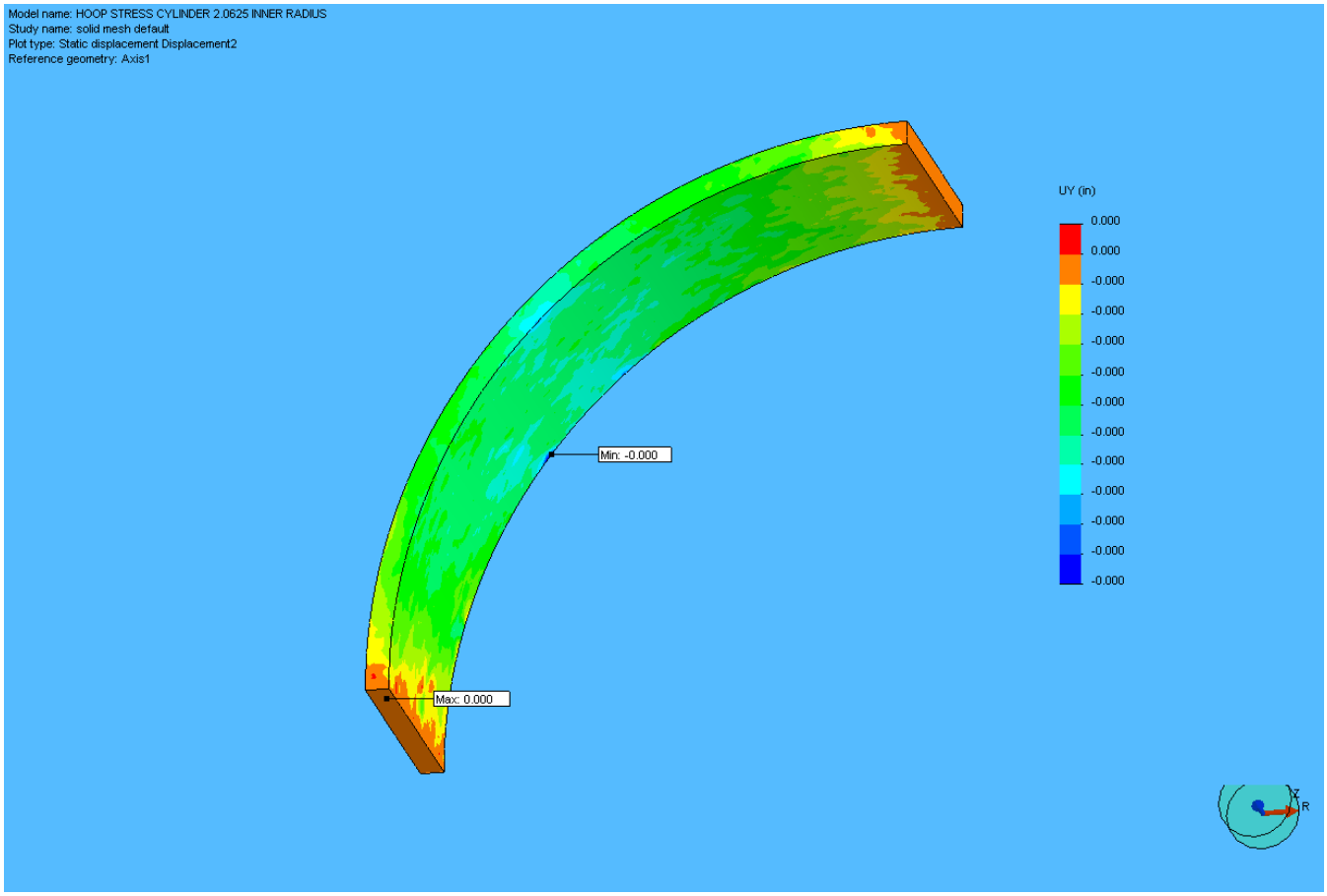


Figure 64: Hoop displacement, UY

The hoop displacements between the two elements were identical, 0 displacement.

I tried my best to find good simple hand calculations to check the strains and the displacements using Hooke's law. Nevertheless, as I showed the strain field seems to be complex and therefore simple 2D calculations to check the strain and the displacements quite literally failed. I showed the complex nature of the calculations that SolidWorks incorporates when looking at the relations between stresses and strains and therefore since it had been years since doing anything like that, I decided to take a pass on doing those 3D calculations.

In conclusion, since SolidWorks has given such highly accurate results for the stresses I think I can be very confident that the displacements and the strains are accurate. Remember SolidWorks calculates the displacements first, and then it calculates the strains and then lastly the stresses. With the stresses being so accurate how can I not trust the displacements and the strains?

If anyone else knows a better way to check for these without doing a 3D hand calculation or perhaps I totally missed something, please let me know and I shall revise this report.

Now let us look at Factors of Safety or F.O.S.

Factor of Safety or FOS

Von Mises Stress Criterion

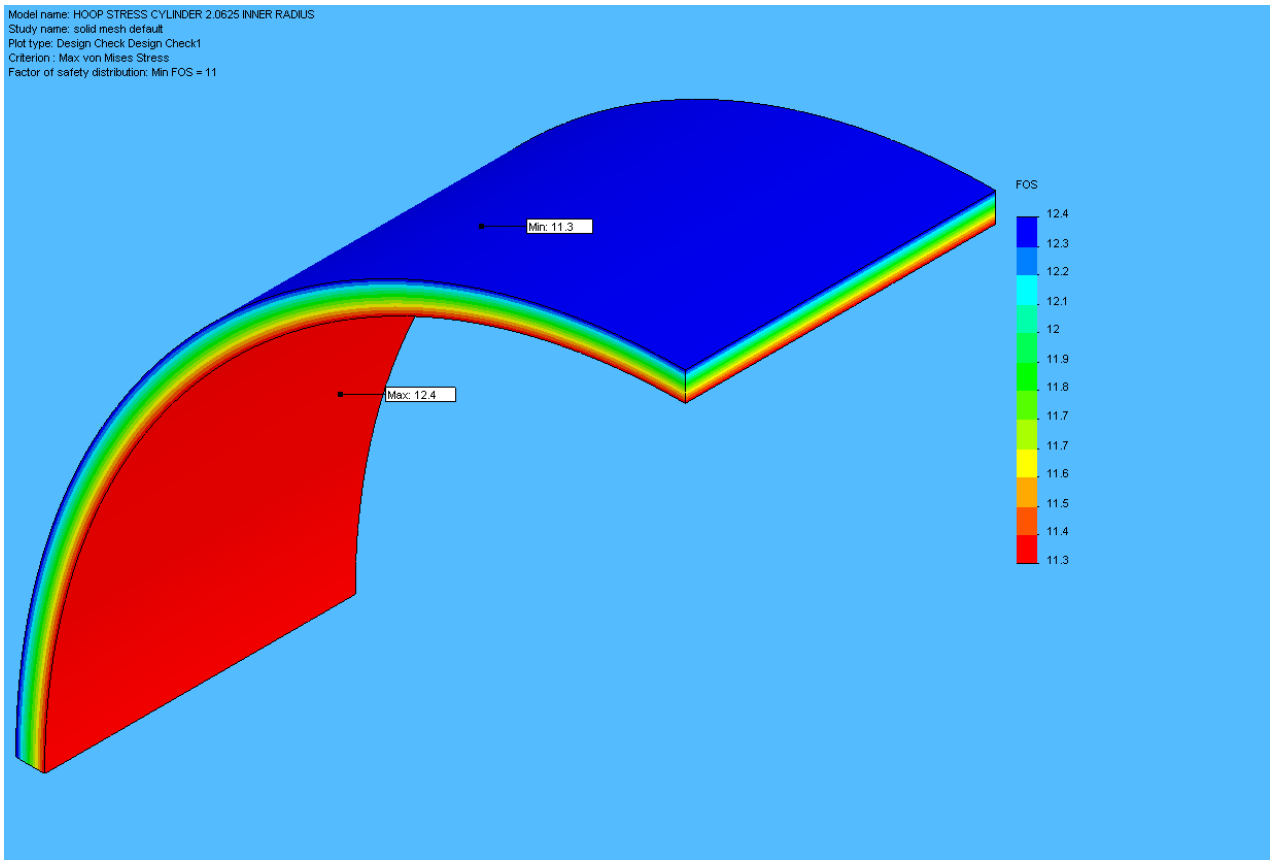


Figure 65: Automatically generated von Mises FOS Distribution plot

SolidWorks can generate what it calls a Design Check (FOS) plot for checking the safety of the part and it has a few different criteria at its disposal. First and foremost it always defaults to the max von Mises stress criterion, but I think you can change this in the COSMOSWorks settings when defining your plots.

The max vM Stress criterion is;

$$\sigma_{vm} / \sigma_{Limit}$$

You can set your stress limit to the yield strength of the material which in our case for copper is 37513.4 psi or you can set it to the ultimate strength which is 57200 psi or you can set your own by typing it in. SolidWorks also gives you the max stress in the model which for figure 65 would be the max von Mises stress which it calculated to be 3316 psi.(Note; this is not the maximum stress in the cylinder, the max is our Hoop Stress which 3548.4 psi with solid elements and 3628.4 with the shells.)

Many people use this criterion because it allows the most complicated stress situation to be represented by one quantity. Consider the most complex state of stress that you can imagine, like a three dimensional stress element subject to a combination of shear and normal stresses on every face. The number represents a “Magnitude” it is a scalar number which can be compared against the yield strength of the material. The equation for the von Mises stress can be based on the principal stresses Sigma 1, Sigma 2, and Sigma 3.

Equation 12

$$\sigma_{vm} = \sqrt{.5[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

Using our theoretically calculated values for the principals gives; sigma 1 = 3548.4, sigma 2 (adjusted per Z strain calc.) = 1372.1, sigma 3 = -160 we get;

$$\sigma_{vm} = 3227.68 \text{ psi} \quad \text{(Theoretical)}$$

From figure 66, the probed mid-point value for the von Mises stress was 3171.62 psi

% error = [(3171.62 – 3227.68) / 3171.62] * 100 = 1.7675%

% error = 1.77% Acceptable

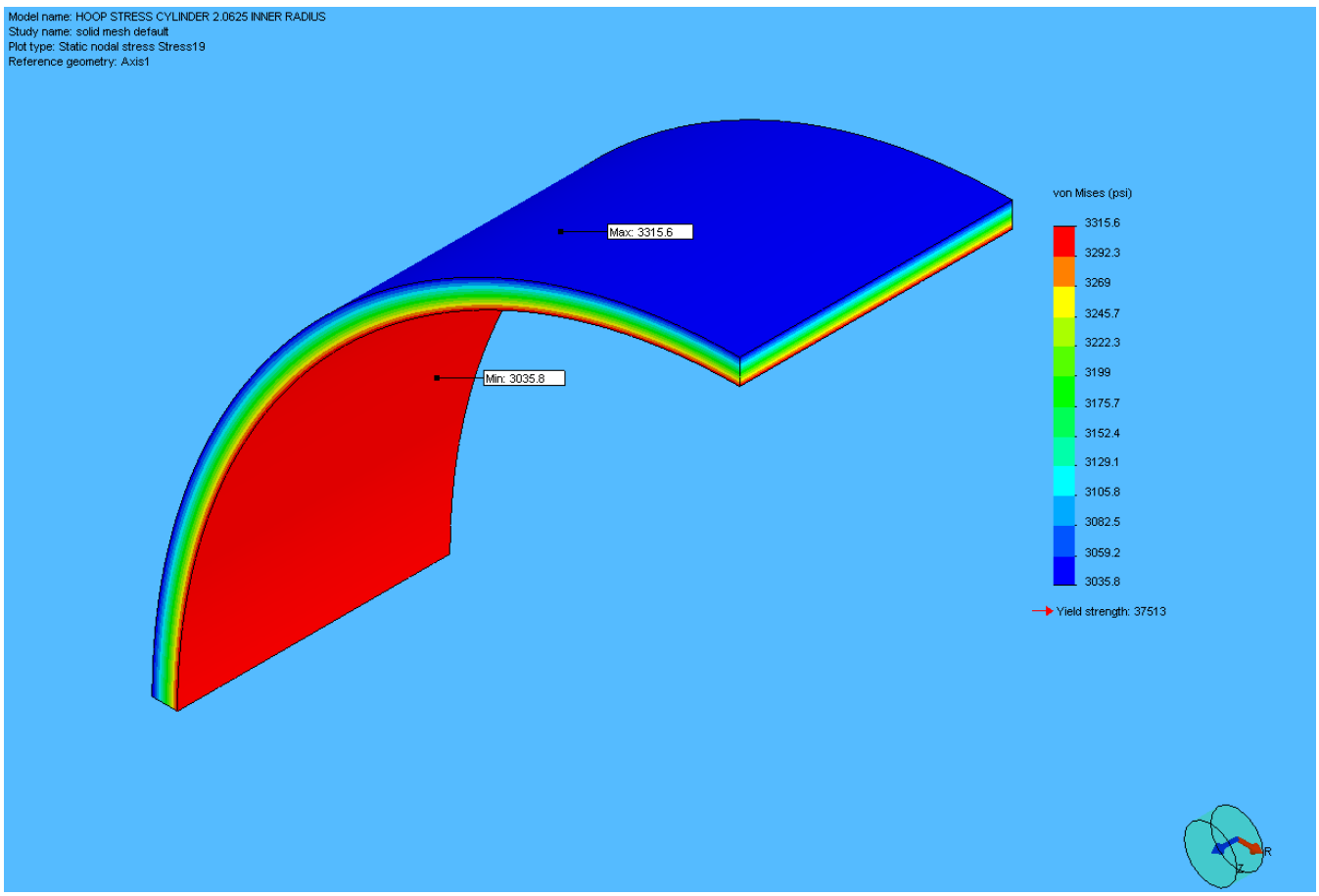


Figure 66: von Mises stress plot in cylindrical coords.

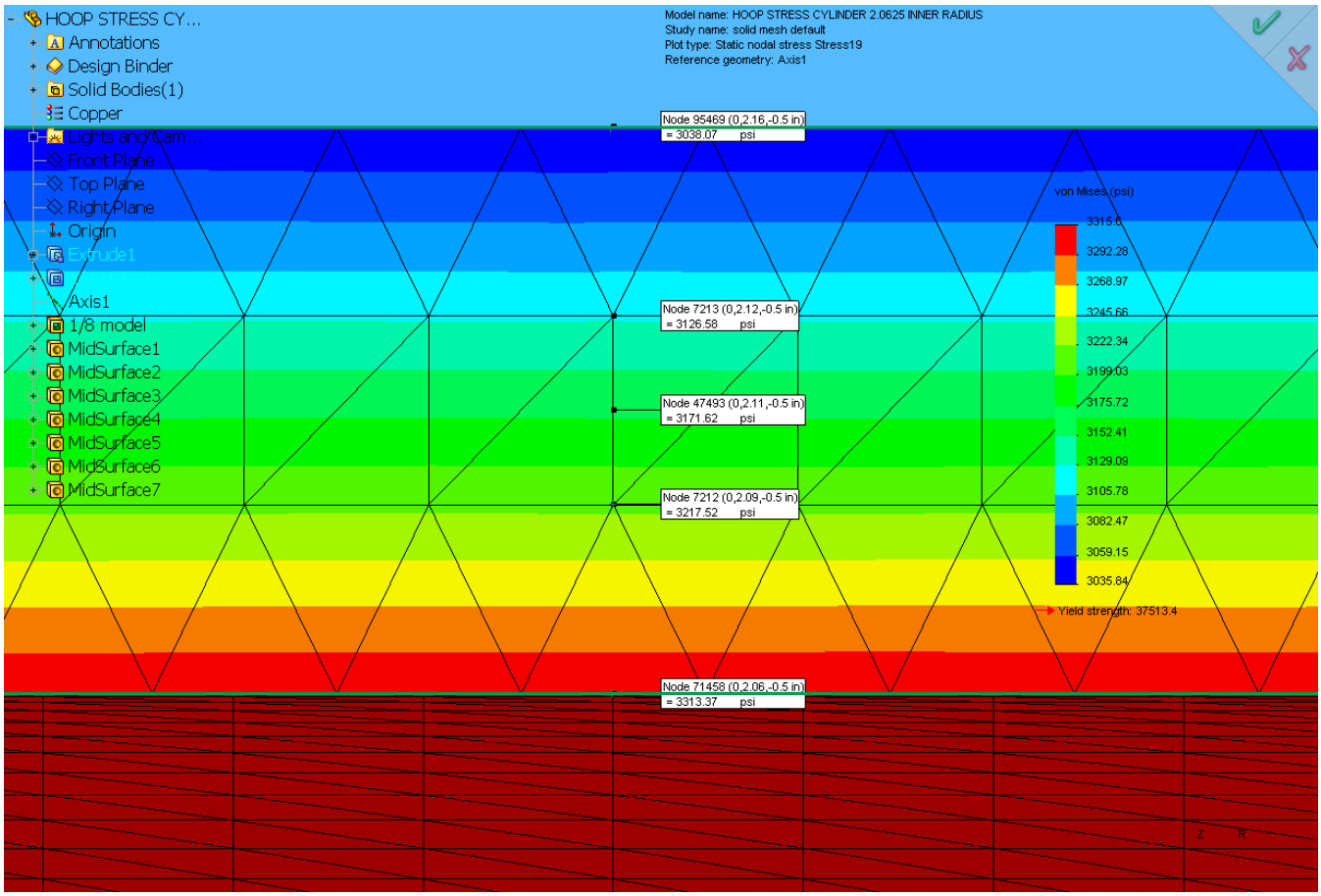


Figure 67: Probed values for the von Mises stress in cyl coords

As I mentioned before when we were looking at the FOS plot of figure 65 our maximum stress in the cylinder is more than the von Mises stress (figure 66) so we cannot really take the von Mises FOS as a true interpretation of the safety for the cylinder. We can however come up with our own FOS based on the Hoop stress, which is the largest stress in the part and the most likely stress that will cause a failure.

<u>Equation 13</u>

For the solid element with a radius value of 2.0625 we get;

FOS = Allowable Stress (strength) / Calculated Stress = σ_{yield} / σ_H, where σ_{yield} = 37513.4 psi.

Theoretical Hoop Safety FOS = 37513.4 / 3548.4 = 10.57 Acceptable

For the shell element with a meridian radius value of 2.109 we get;

FOS = Allowable Stress (strength) / Calculated Stress = σ_{yield} / σ_H, where σ_{yield} = 37513.4 psi.

Theoretical Hoop Safety FOS = 37513.4 / 3628.4 = 10.34 Acceptable

Using the probed result from figure 67 for von Mises stress in the middle of the wall thickness we get a value of 3171.62.

SW von Mises Safety FOS = 37513.4 / 3171.62 = 11.82 Not Acceptable

Moreover, our percent error between the two FOS just for the solid element would be;

% error = [(11.82 – 10.57) / 11.82] *100 = 10.575 %

% error = 10.58 % Not Acceptable

The reason I am deeming this error not acceptable is not because of the high error value. It is because the von Mises stress is ***NOT*** the stress that will cause a failure in the cylinder; the Hoop Stress is what will cause a failure in the cylinder it is also our highest value for stress in the cylinder. Also notice that the vM FOS is actually a higher value than the theoretical indicating the cylinder is safer than it actually is. In addition, notice in figure 65 there is a range of values for the FOS from between 11.3 to 12.4. The values go from the inside of the cylinder where the FOS is a minimum to the outside of the cylinder where the FOS is a maximum, if I didn't know better I would swear we were looking at the hoop stress.

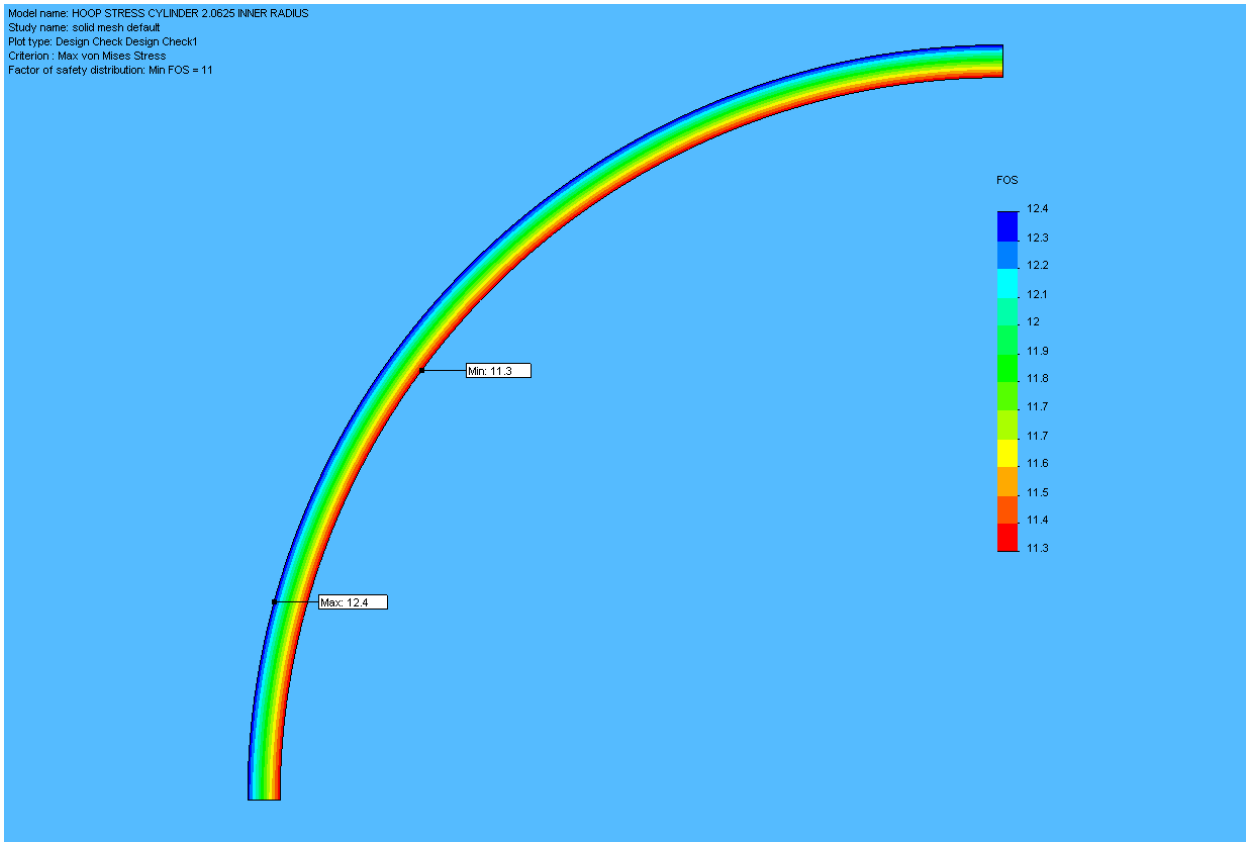


Figure 68: von Mises FOS showing the range of values, solid elements.

This is a distribution of the FOS thru the entire part and it makes sense because we have higher stresses on the inside that decrease to the outside. Our Hoop stress decreased this way and especially the radial stress decreased this way in fact it went to zero on the outside. Therefore, because of those two trends we see our safety factor follow suite with higher numbers on the outside.

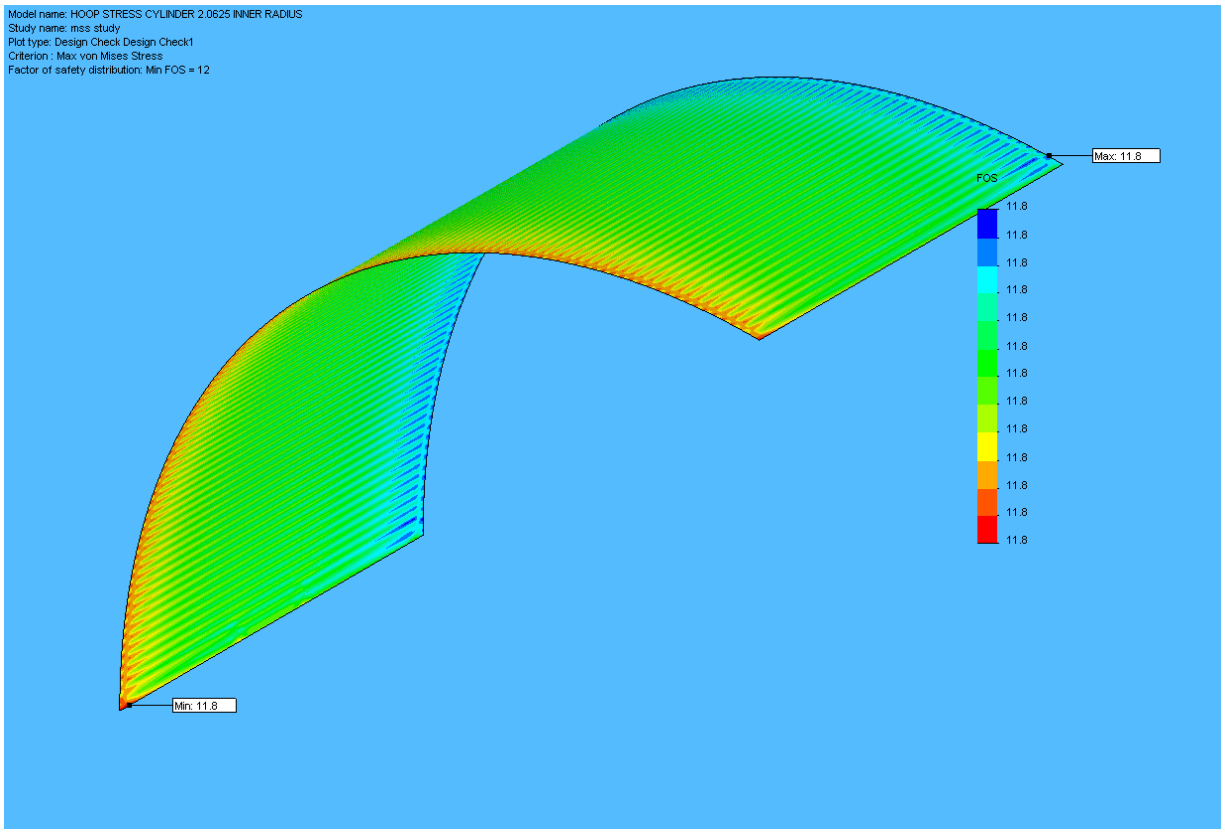


Figure 69: Auto generated von Mises FOS with shell elements at the membrane.

The shell value for the von Mises FOS you see in figure 69 matches up pretty well with the probe values we had for solid element. The listed average value for the entire face of the shell element membrane was 11.807 and our calculated value for the probed solid was 11.82.

Max Shear Stress or Tresca yield criterion

Another criterion recommended for ductile materials like copper is the Tresca (Maximum-Shear Stress) theory. Since the maximum shear stress in an element loaded in pure tension is one half the maximum tensile stress, the shear yield strength is 1/2 Sigma Y. Thus Tresca yield is;

$$\tau_{max} = (\sigma_1 - \sigma_3) / 2 = .5\ S_Y$$

All yielding is said to occur when $2\ \tau_{Tresca} = S_Y$
Using our theoretically calculated values for the principals; sigma 1 = 3548.4, sigma 2 (adjusted per Z strain calc.) = 1372.1, sigma 3 = -160 we get;

$$\tau_{Tresca} = [3548.4 - (-160)] / 2 = 1854.2$$

SolidWorks FOS value given for max shear stress in the model was 1896 psi.

% error for shear stress = [(1896 – 1854.2) / 1896] * 100 = 2.2046%

% error for shear stress = 2.2% Acceptable

SolidWorks uses FOS = $\sigma_{Limit} / (2\ \tau_{max})$ and our sigma limit is our yield strength of the material which is 37513.4 psi.

FOS = 37513.4 / (2*1854.2) = 10.116

Probed value from figure 71 at the mid point is 10.348 for the solid element.

% error for Tresca FOS = [(10.348 – 10.116) / 10.348] * 100 = 2.242 %

% error for Tresca FOS = 2.2 % Acceptable

Listed average value for the shell FOS at the membrane was 10.34.

Although our values for the Tresca FOS that SolidWorks gives are more in line with our theoretical it is still not the maximum stress component in the cylinder. The Tresca takes into account sigma's 1 & 2 and divides that number in half and then compares that to the yield which is why we have a slightly lower FOS with the Tresca. 10.35 as opposed to our calculated Hoop stress FOS 10.57 and in this case the Hoop stress FOS should be used because it is the governing stress.

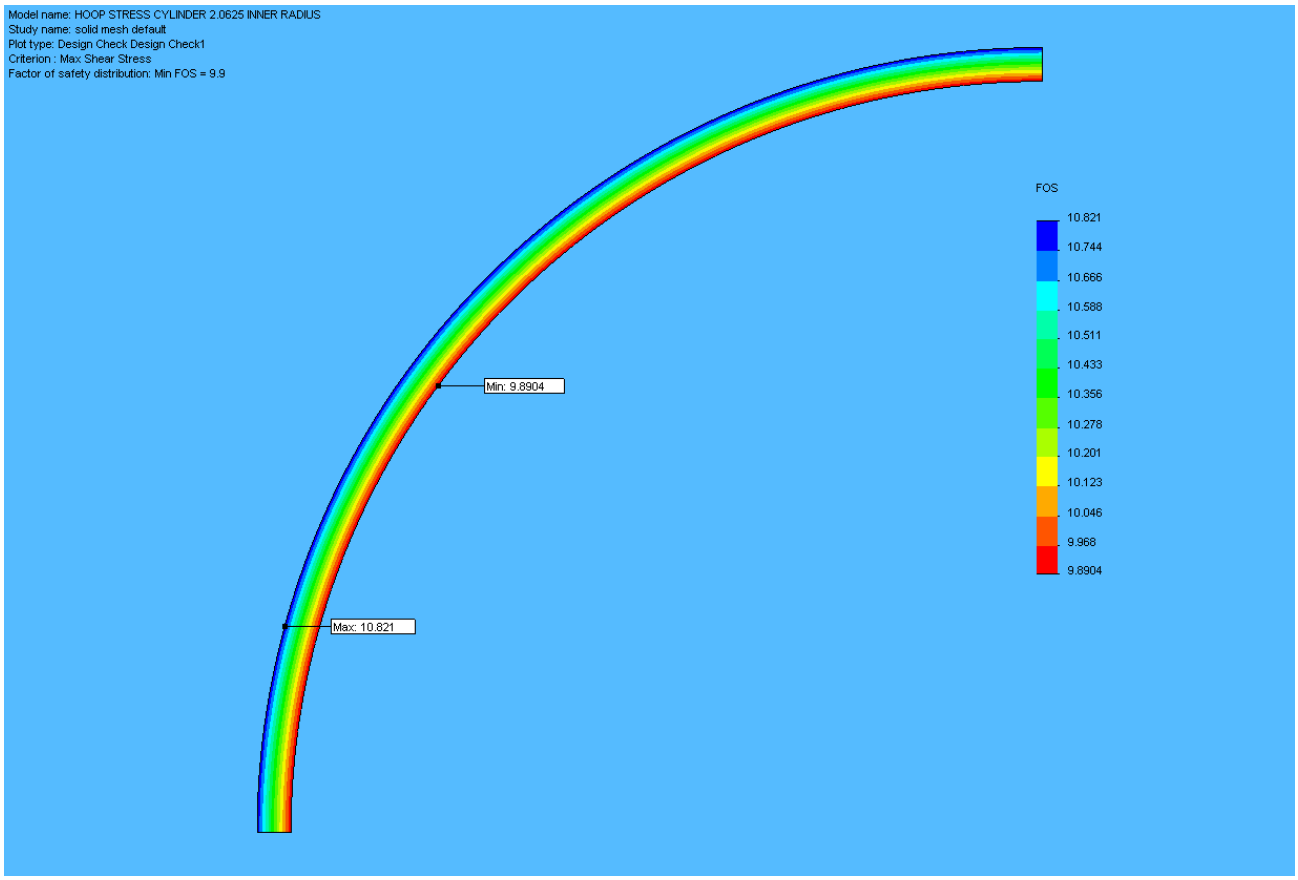


Figure 70: Auto generated Tresca FOS plot, right side view.

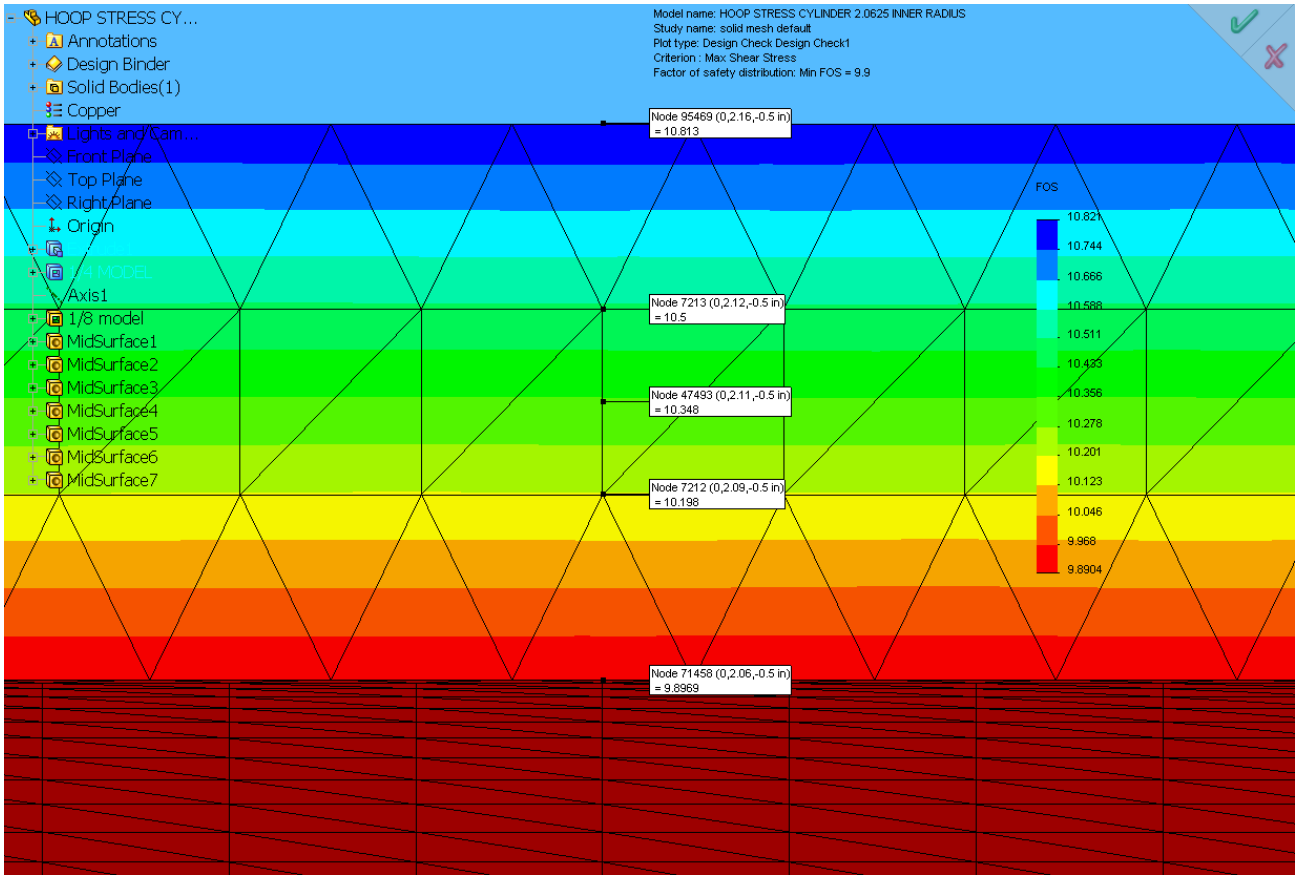


Figure 71: Probe results for Tresca FOS mid point.

Conclusion

A mountain of information has as I know been presented here and its intent was to show the differences between classic stress analysis equations for a cylinder and SolidWorks element analysis in both solid and mid-surface shells along with the variations between them all for the same cylinder. Future endeavors for the boiler analysis will not go this far. Though it would have been nice to be able to do a 3D classic stress analysis for the displacements for verification, my knowledge in this area has since diminished and is better left to a more polished mind in that area.

Considering the low errors, we were getting for the stresses and the somewhat rigorous approach undertook overall, I believe the displacements as well as the strains in SolidWorks are reasonably valid numbers.

Overall, I would say that we got very reasonable numbers from SolidWorks for the hoop, axial and radial stresses. We saw the error between the classic hand equations and the values that SW gave us were low for the stresses, but not the displacements and the strains. It was my thinking that in order to get good error numbers for the latter two, a 3D stress calculation is would be needed, and in place of doing this I decided to accept the values for the strains and the displacements because the errors for the stresses were so low.

The reason I succumbed to this conclusion was do to the order of operations that SW uses when doing its calculations. Displacements are calculated first then strains and then finally stresses, so logically it stands to reason that if there is a very low error percentage in the stresses the calculations that were done prior to those must also be worthy of a low error percentage and therefore be correct.

We discovered a bending stress in the shell element analysis, something we did not know existed in the solid element analysis.

For the most part it just comes down to what you are looking for and what you want to look at, but at the same time you have to know what to look for and where to look for it. We saw, there is a lot to look at in SolidWorks and one does need a good baseline from which to start.

The most critical stress in a thin wall cylinder is the hoop stress. I showed how that stress should be used to calculate the factor of safety as well as why not to use the vonMises stress FOS.
I also showed why we should look at both solid elements and shell elements, one showed us a bending stress while the other did not and one showed us a compression of the material while the other did not.

As a general rule, always use your highest stress value for your safety calculations.
I think a FOS of 10 for the cylinder at the 160psi test pressure is more than adequate and when we look at the running pressure of 80 psi we will get an even better FOS, it probably won't be 2 twice as much but I'm guessing should fall in around 15 or so.

Bibliography

1.) Machine Design, an Integrated Approach, 2nd Edition, Robert L. Norton
2.) Engineering Analysis with SolidWorks Simulation 2009, Paul M. Kurowski, PhD, PE
3.) Analysis of Machine Elements using COSMOSWorks Professional, John R. Steffen, PhD, PE
4.) Advanced Strength and Applied Stress Analysis, 2nd Edition, Richard G. Budynas
5.) Pressure vessels, David Roylance, MIT, August 23, 2001
6.) SolidWorks 2007
7.) Roark's formulas for stress and strain. WARREN C. YOUNG, RICHARD G. BUDYNAS, 2002
8.) http://en.wikipedia.org/wiki/Pressure_vessels