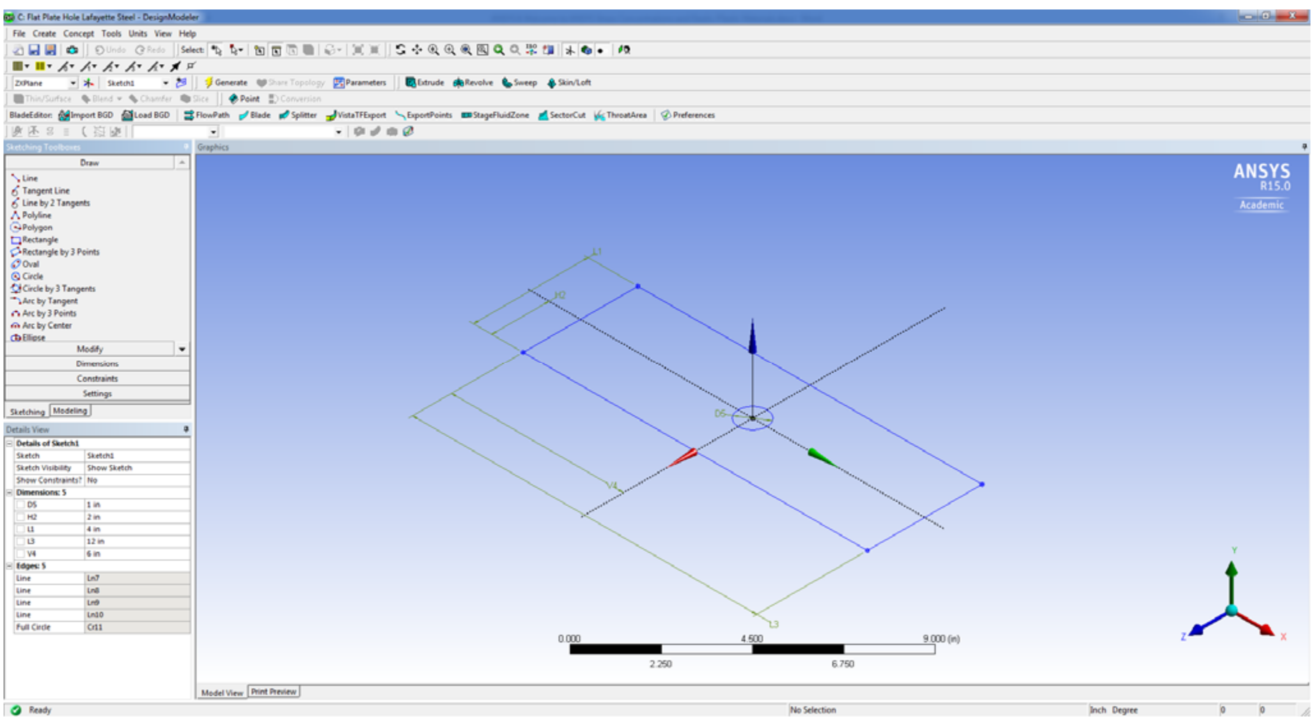
<u>TUTORIAL 7:</u> Stress Concentrations and Elastic-Plastic (Yielding) Material Behavior

In this tutorial you will learn how to recognize and deal with a common modeling issues involving stress concentrations in ANSYS. In this tutorial, you start by looking at the stress concentrations that occur in a flat plate with a circular hole. With this simple model you will be able to compare the stress concentration results obtained for a plate made out of a non-yielding material, which is modeled by our 'Lafayette Steel', and a plate made out of a yielding material, which will be modeled by a new material 'Elastic-Plastic Lafayette Steel'.

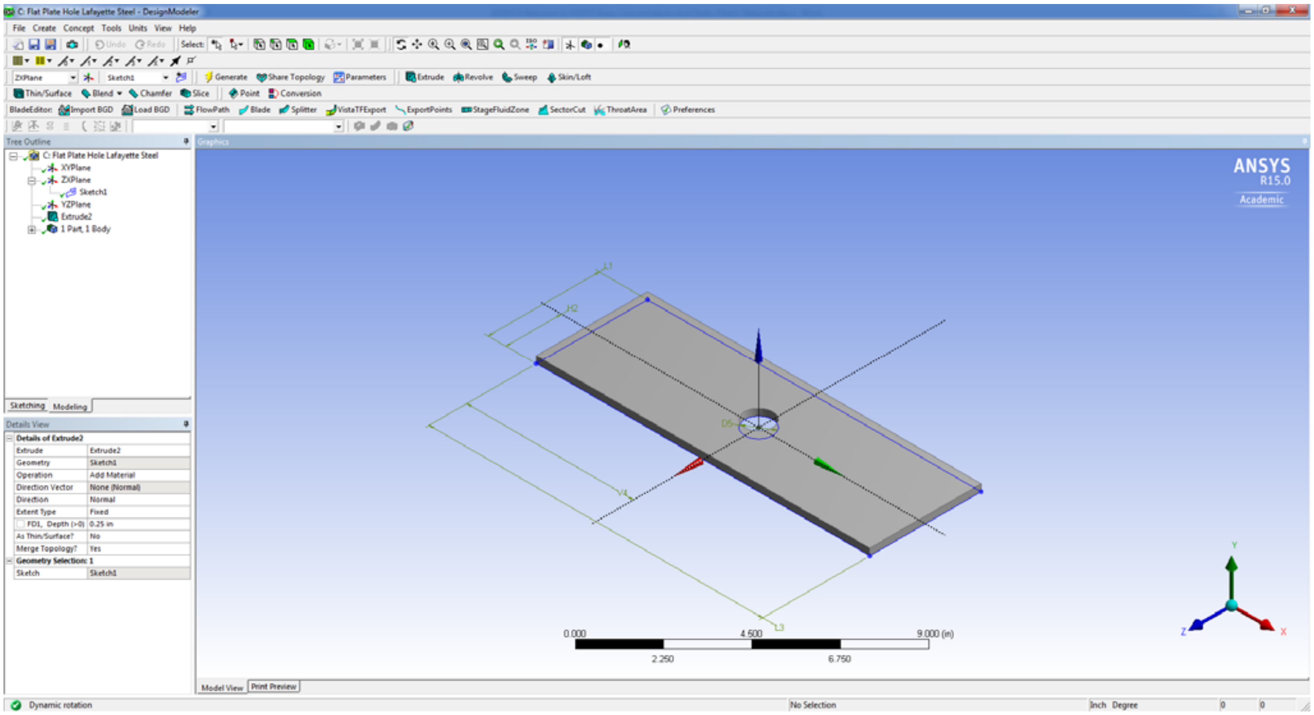
The flat plate itself is 12 inches long, 4 inches wide and .25 inches thick. The hole will be centered in width of the plate and will have a 1 inch diameter. One end of the flat plate will be fixed and an axial load of 10,000 lb. will be applied to free end of the flat plate in the x-direction. The material to start with is 'Lafayette Steel'.

Initial Project Space Setup

Set up a new **Static Structural** project in a new **ANSYS** environment. Change the material by *Importing* in 'Lafayette Steel' and set it as the *Default Material* for the material, set the *Units* to US Customary and start up the *Sketching* environment with the *Units* in <u>Inches</u> and with the *Auto Constraints* turned <u>On</u>. Sketch out a rectangle on the **ZX Plane**. Make sure the *Units* are in <u>Inches</u>. Add two *Dimensions* to set the length and the width of the rectangle to match the 12" length and 4" width of the plate being modelled. Then add two *Dimensions* to help center the rectangle shape on the ZX axes. Then *Draw* a circle centered at the axis origin in the view. *Dimension* the circle with the 1 inch diameter.

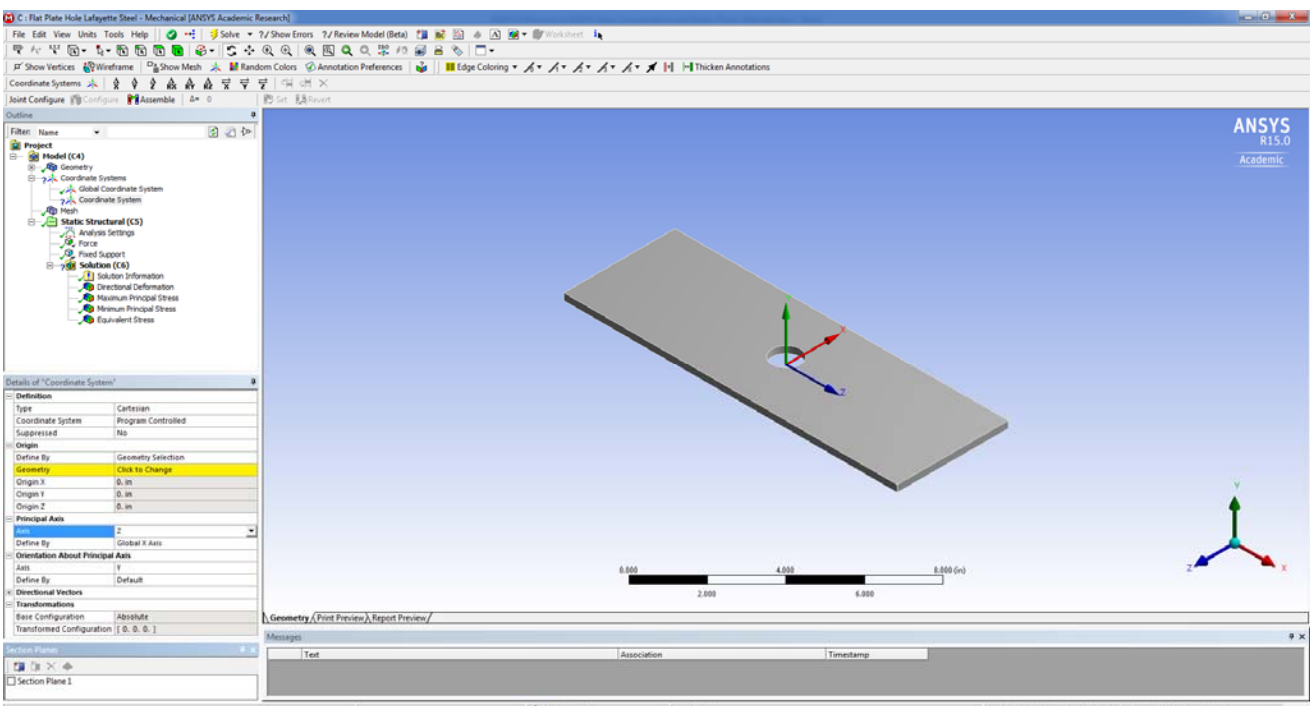


Then select the *Sketch1* you just created in the ZX plane in the Modelling view and use **Extrude** and then **Generate** to create the 3D flat plate that has the 0.25 inch thickness desired.

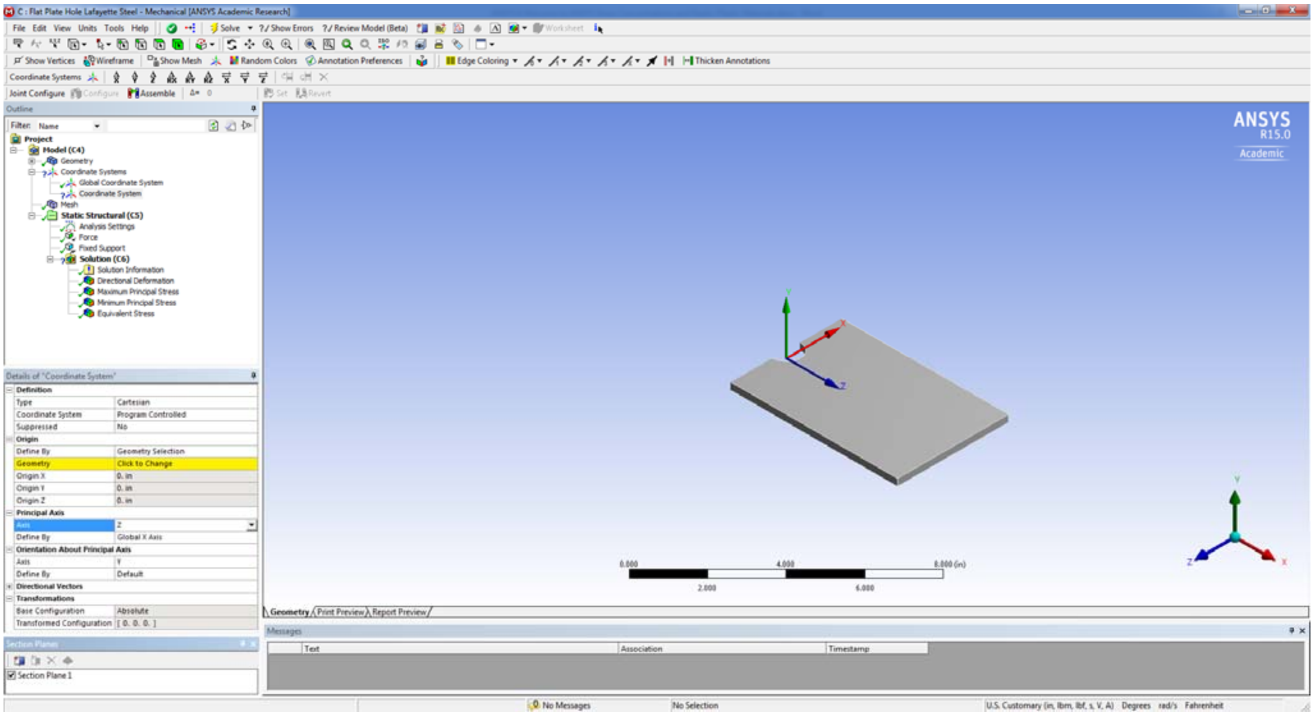


Now open the **Mechanical** window. Change the **Mesh** to have an *Element Size* of 0.1 in and the right-click on **Mesh** to *Generate Mesh*. Add a load of 10,000 lbs in the +x-axis direction to the near end of the flat plate and add a fixed support to the far end of the flat plate. At this point you should have a 3D flat plate model ready to analyze. Add the *Maximum Principal Stress, Minimum Principal Stress, and Equivalent Stress Viewers* as well as a *Deformation Directional Viewer* that tracks deformation in the x-axis direction. At this point also create a new Coordinate System at the original origin (0 inches) inches from the support with the Z axis specified as done in previous tutorials. Turn this Coordinate System into a Section Plane cut at this location.

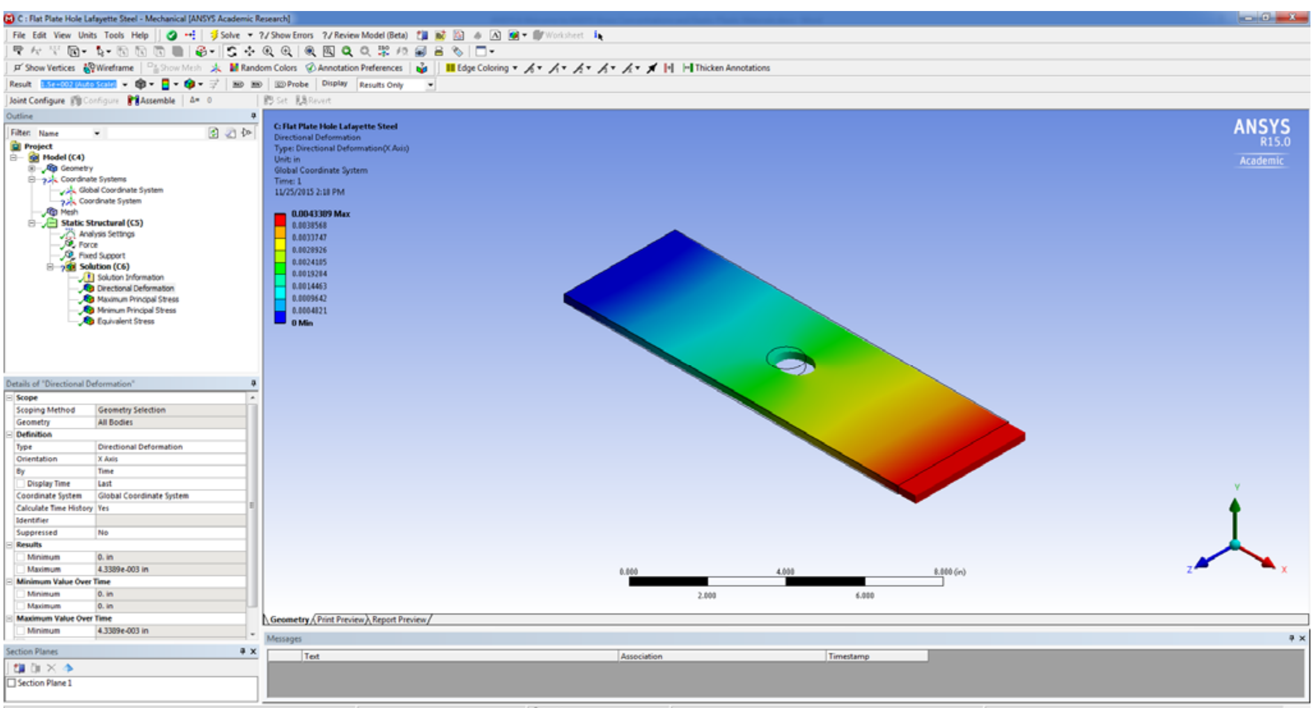
Entire Flat Plate Model with Viewers – Section Plane checked off



Section Cut Taken Thru Hole in the Flat Plate Model – Section Plane clicked ON



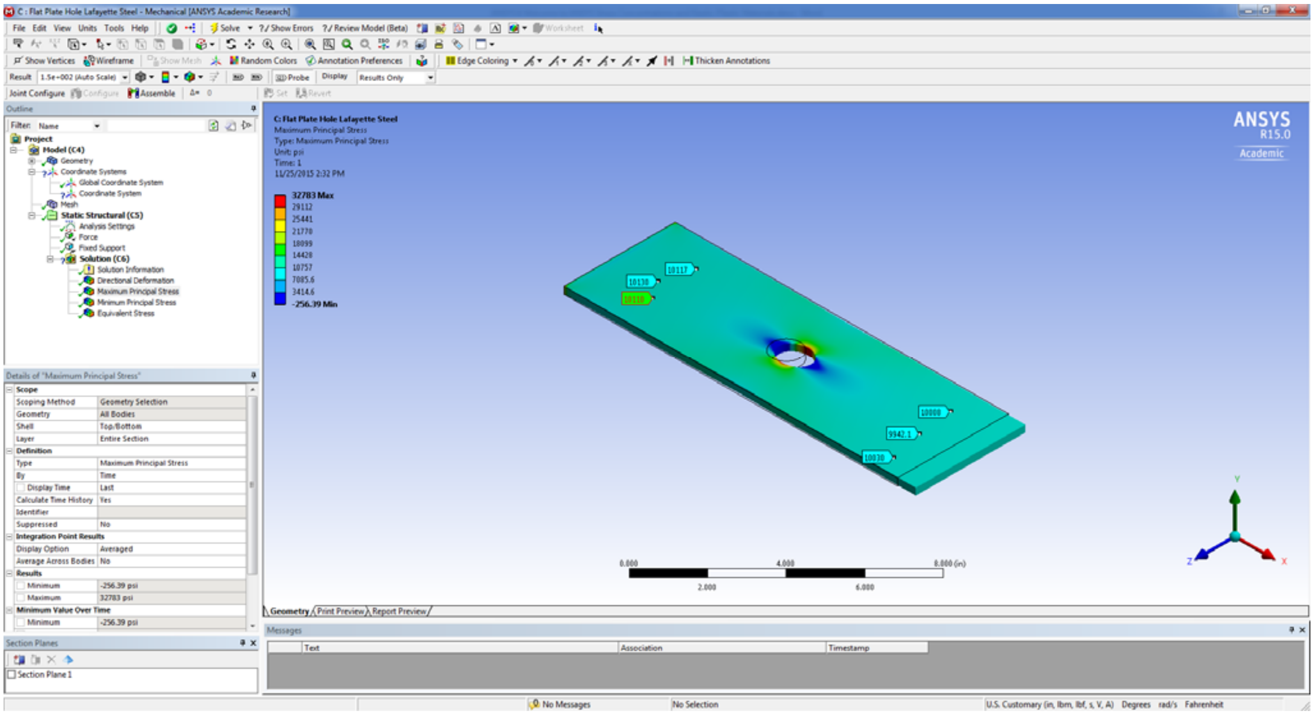
Solve the model and view the X-axis Directional Deformation results for the whole model. If you get an *Assignment error* at this point, you need to assign 'Lafayette Steel' as the material for the model either as the default in Property Data or to the Bar element in the **Mechanical** window. You can do a quick check of the deformation reported by ANSYS by running the PL/AE equation for the flat plate. You should expect to get a larger deformation reported by ANSYS due to the presence of the hole in the plate, but the numbers should be fairly close.



Viewing the Maximum Principal Stress Results

Change to the **Wireframe** option on the top Toolbar to view the mesh points and to help select Probe points.

View the **Maximum Principal Stress** results for the whole flat plate model. Use *Probe* to first determine the maximum principal stress, σ_1, for a point out in the plate away for the hole. Check the ANSYS value obtained for the maximum principal stress away from the hole location against a hand calculation of the maximum principal stress, which can be calculated using $\sigma_1 = \sigma = P/A$ since there is only axial stress acting on these points away from the hole location. The two stress values obtained by hand and from ANSYS should be pretty close as shown below.



Now view the **Maximum Principal Stress** results for the whole flat plate model in the region near the hole in the plate. Still keep the whole flat plate view for this comparison. Use *Probe* to determine the maximum principal stress, σ_1, for several points near the hole. If you would like to compare the stresses located near the hole to hand calculated values you will need to account for an approximate stress concentration factor. This factor is an adjusted factor that depends on the relative dimensions of the plate and the diameter of the hole that is present. This topic is covered in more detail in Section 4.7 in the Hibbeler textbook. The stress concentration factor, K, accounts for the expected ratio between the maximum stress in the plate near the hole and the average stress in the plate considering the net section area at the hole. So $K = \sigma_{max}/\sigma_{avg}$. The stress concentration factor, K, is usually obtained from design tables for specific applications. For a circular hole with the property *2*hole radius/plate width* of 0.25 for the plate and hole considered in your model, the K factor is 2.38. Therefore, the expected maximum normal stress in the vicinity of the hole is $2.38 * P/A_{net}$ where the A_{net} = (*width-diameter*)**thickness* = 0.75 in^2. This approximate hand calculated maximum stress in the region of the hole is approximately 31733 psi. The values you are obtaining are slightly higher than this approximate stress concentration stress found using table values.

In many cases you can calculate more accurate stress concentration factors, K, using equations, such as thru (http://www.amesweb.info/StressConcentrationFactor/CentralCircularHoleInFiniteWidthPlate.aspx). Using the equation setup, the maximum stress at the hole is 32,271 psi. This value matches well with ANSYS.

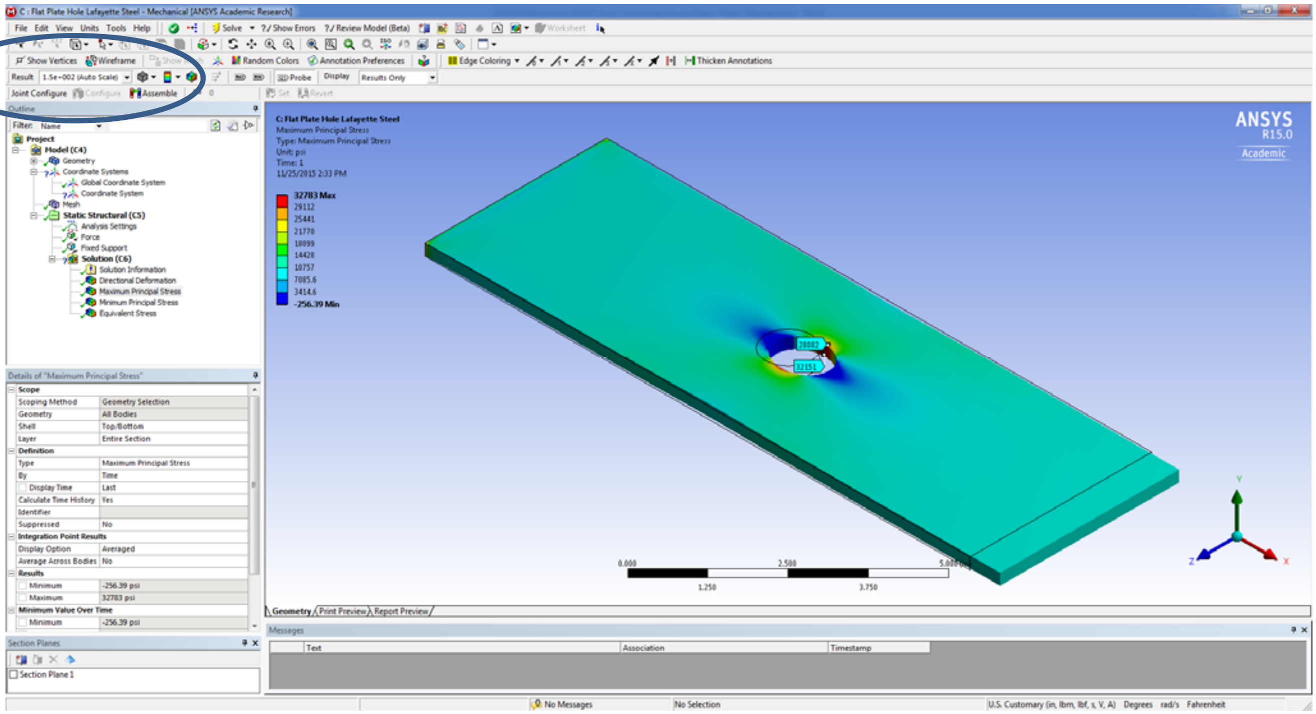
INPUT PARAMETERS

Parameter	Symbol	Value	Unit
Plate width	D	4	inch
Hole diameter	d	1	
Plate thickness	t	.25	
Tension force	P	10000	lbf
In-plane bending moment	M	0	lbf*in
Transverse bending moment	M_1	0	
Calculate			

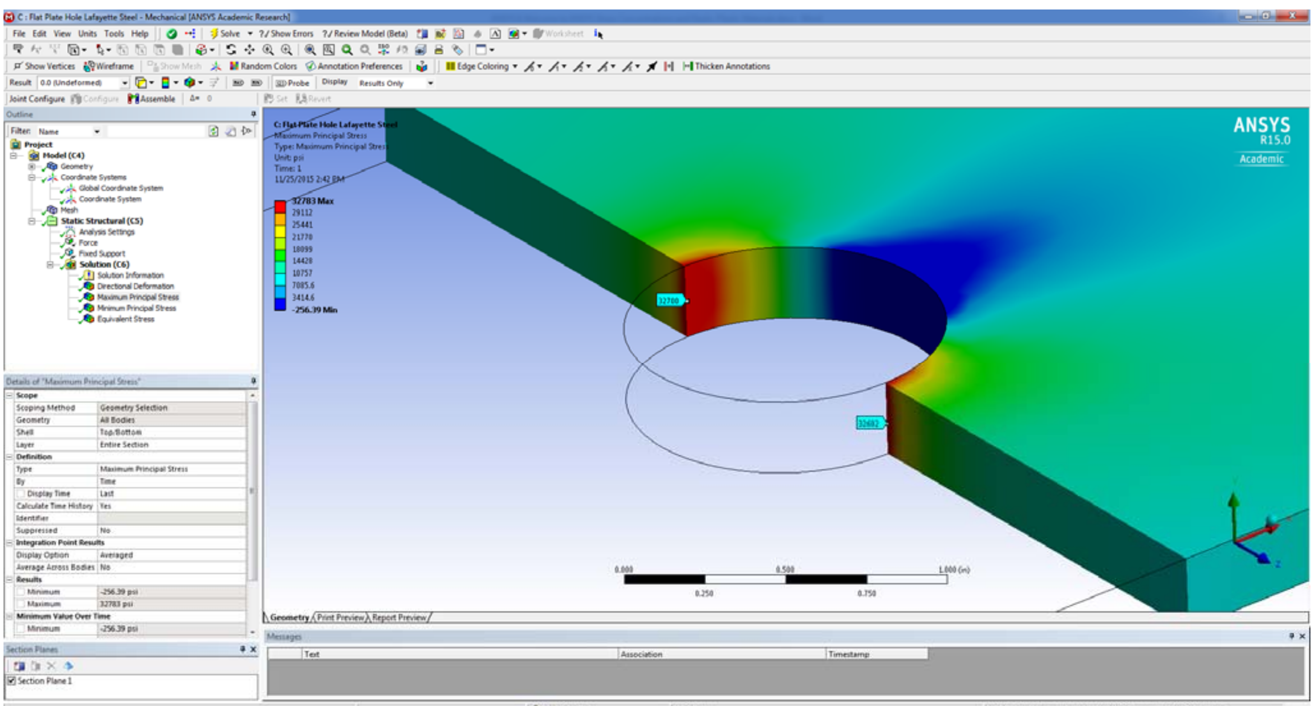
RESULTS

LOADING TYPE - TENSION

Parameter	Symbol	Value	Unit
Stress concentration factor *	K_t	2.42	---
Nominal tension stress ◊	σ_{nom}	13333.33	psi
Maximum tension stress at the edge of hole	σ_{max}	32271.03	



To look at the results in closer detail you can view the **Maximum Principal Stress** results on the Section Plane cut through the flat plate. To view the correct location change the Result from AutoScale to Undeformed in the toolbar. Use *Probe* to determine the maximum principal stress, σ_1, for several points on the cross-section at this section plane cut.

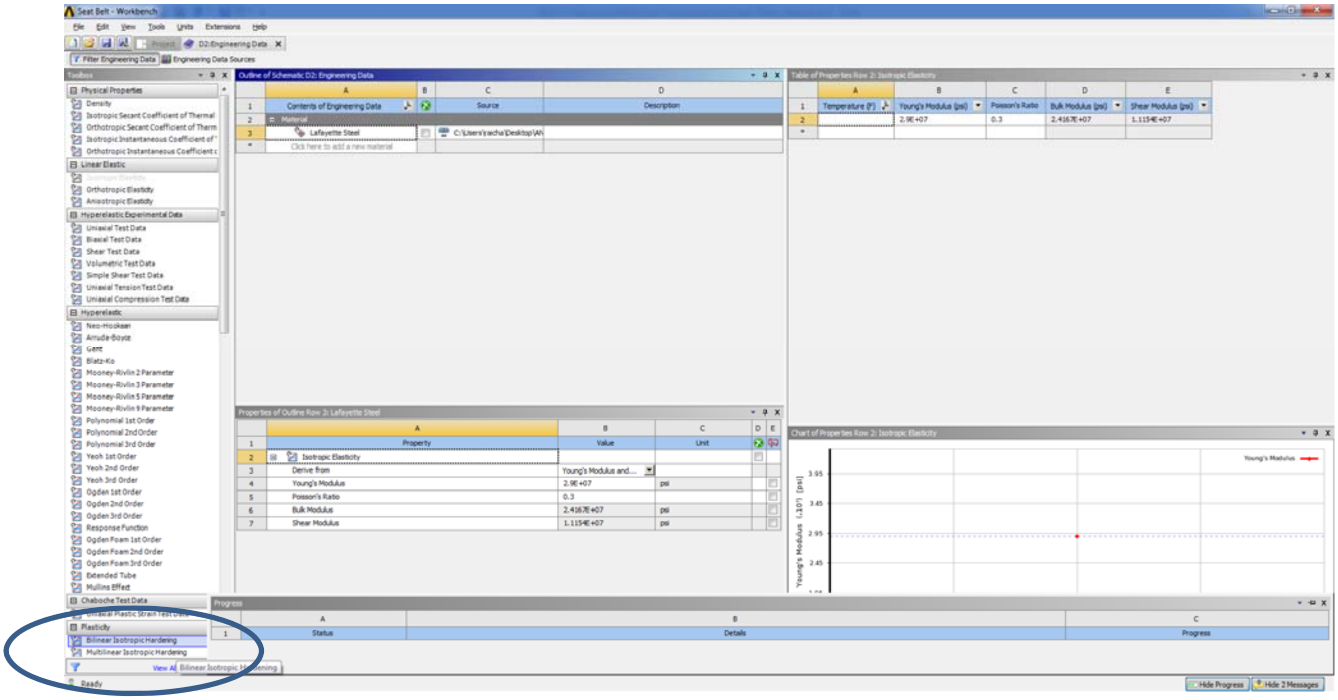


Save your work so far.

Creating an Elastic-Plastic Lafayette Steel Material

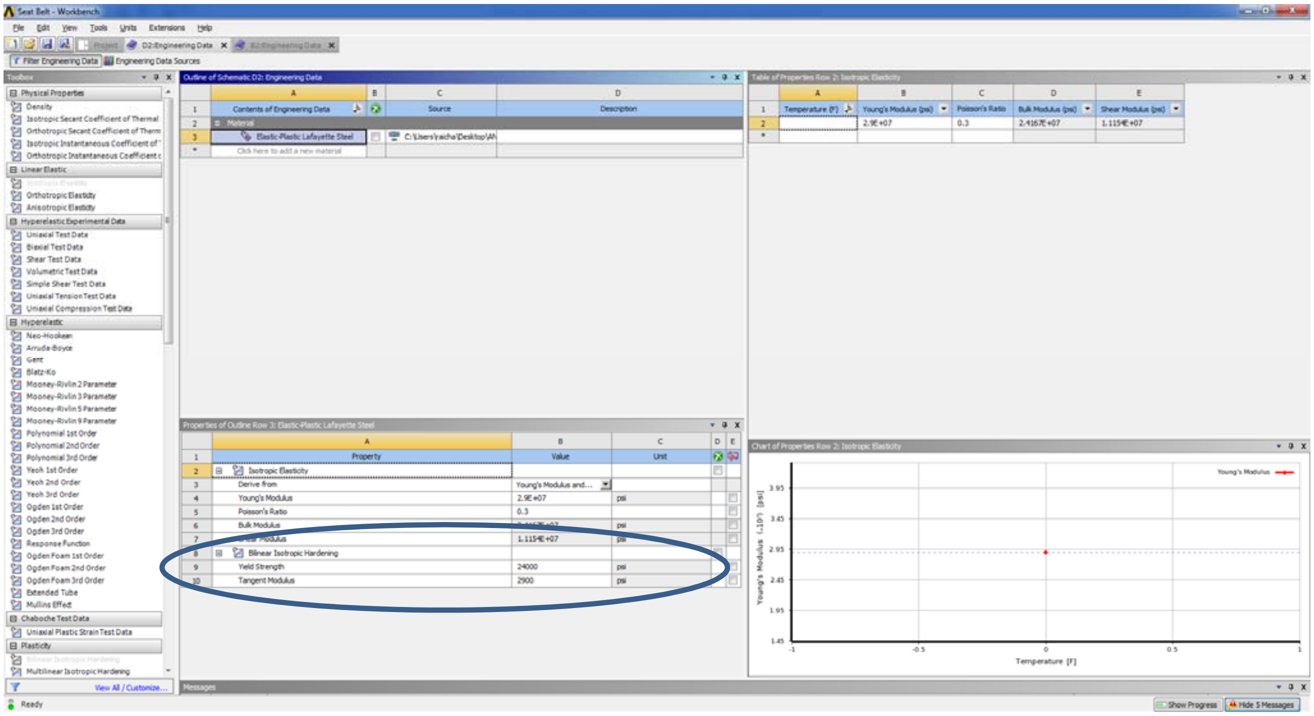
In the ANSYS **Project** space create a duplicate of your current Flat Plate Model. Then open up the **Engineering Data** window for this new model. You will change the properties of the 'Lafayette Steel' material to include a yielding behavior. This requires setting a yield stress and then a tangent modulus for the material to follow once yielding occurs.

In the **Engineering Data** window, *double click* on the Bilinear Isotropic Hardening option in the LHS set of choices, which is located as shown below towards the bottom of the LHS menu. This will open up a new set of variable options in the window, as shown in the next screenshot view.

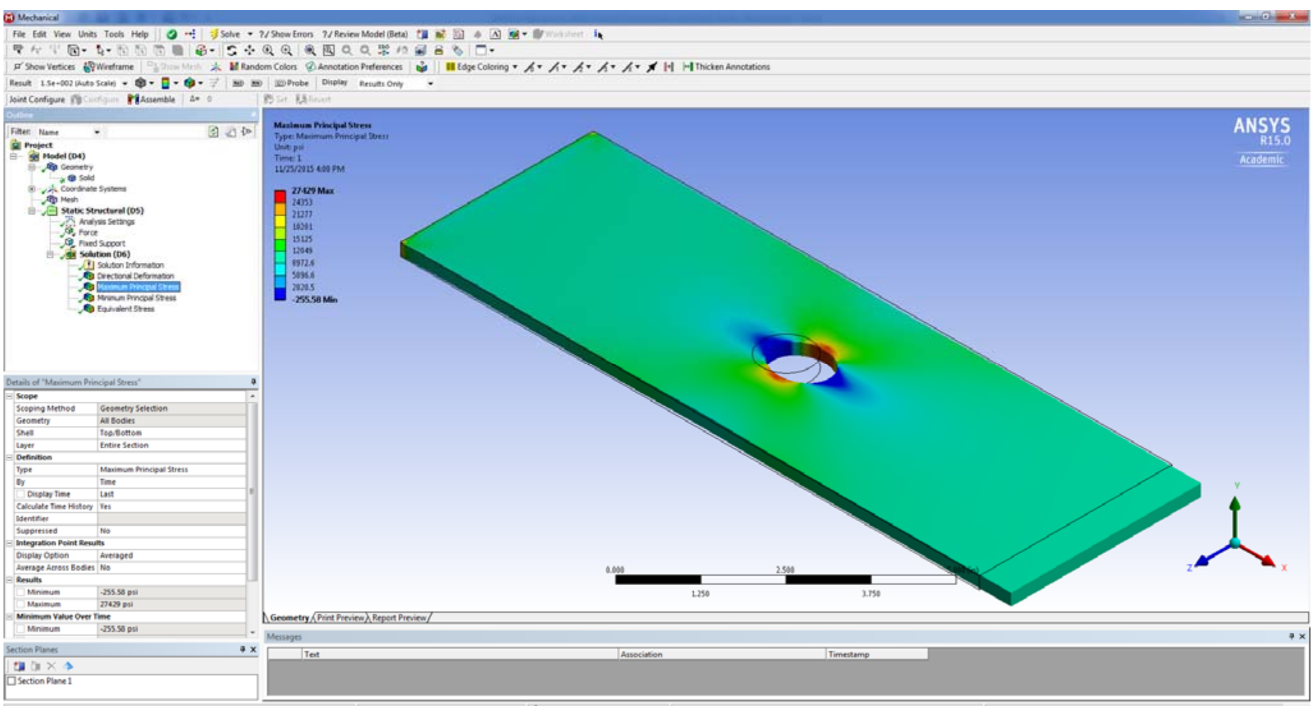


Enter 24,000 psi as the Allowable Yield Strength of the material and also 2900 psi as the Tangent Modulus that will be used to track the post-yielding behavior of the material. If you do not see the stress-strain diagram, which is shown in the bottom right figure, try to click on the Yield Strength box in the data window. It is not important to see the diagram, although it is reassuring typically to see what yielding behavior you have modeled. This specific type of material behavior is called elastic-plastic behavior. The material behaves in a linear, elastic manner until the yield stress is reached. When the yield stress is reached at a particular location the material will yield at the location and will not be able to carry additional levels of stress without having substantial strains (deformations) occurring in this location. Modeling an elastic-plastic material behavior helps to reduce errors in calculating stresses in areas that have stress concentrations in ANSYS. In a real system, the material will yield in locations in the vicinity of high stress concentrations and this behavior will result in higher stresses in these regions, but not the artificially high levels of stress that ANSYS would suggest if a linear material is used in the model. To avoid these artificially high levels of stress (which would not actually occur at inflated high stress with yielding) you should use an elastic-plastic material model or some other nonlinear, yielding material model as you work on more advanced modeling problems.

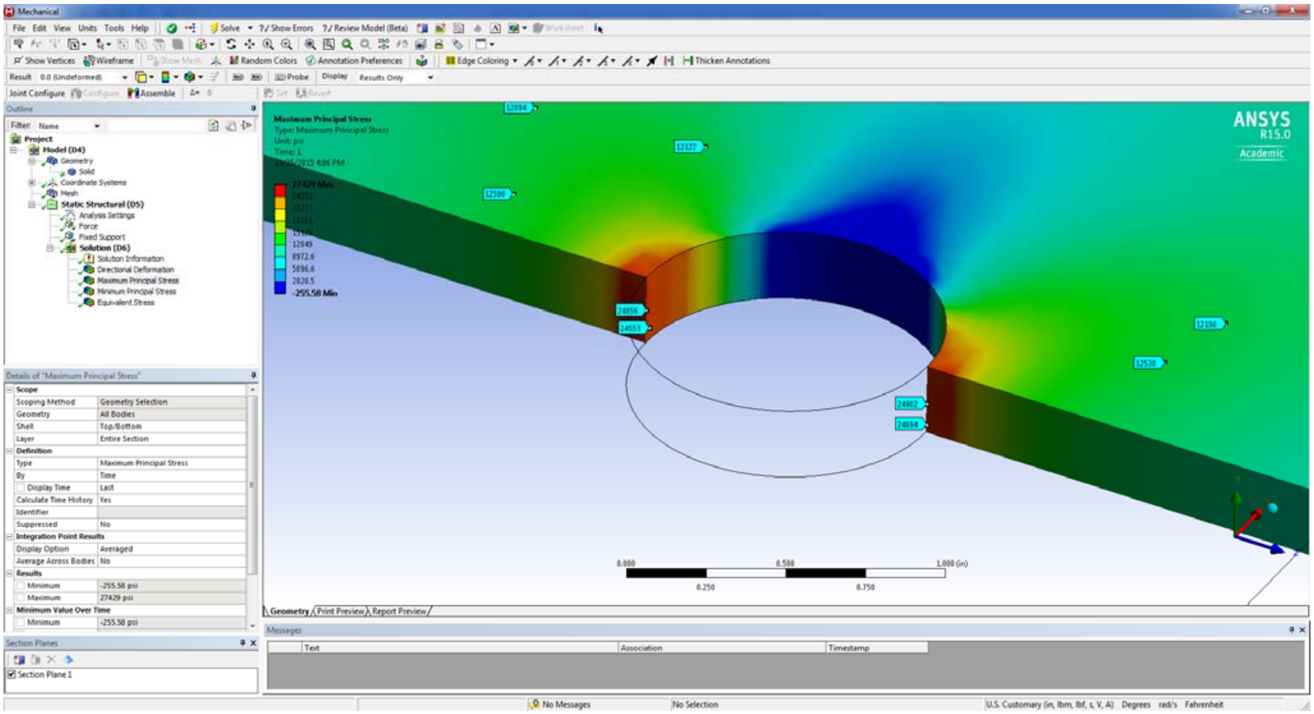
You can **Export** this new Elastic-Plastic material model and **Save** it under a new name such as 'Lafayette Elastic-Plastic Steel. The fillets are run at



Go back to the *Project* Window and right-click on **Model** to **Update** the system in the *Mechanical* window to reflect that there were changes made to the material model. Then double-click on **Model** to open the *Mechanical* window for this new model. **Solve** the new model of the flat plate that has the 'Elastic-Plastic Lafayette Steel' material.



Note that the maximum stress is lower in this model. This is due to the yielding of the material in these regions of high-stress due to the hole in the plate. However, even though the material has yielded in these local regions, the stress is transferred to other regions near the yielded region. These regions may also yield in the region near the hole, but overall the stress in the rest of the plate remains below the yield stress. Therefore, the overall plate deformations are not considerable even though local regions have yielded. This results is a more accurate results of the material behavior given a yield strength of the material and elastic-plastic behavior. Note that in the region of the hole the maximum stress is right around the yield strength of 24,000 psi. The maximum stress is still in the model at the support location. The use of the elastic-plastic material only partially addressing the support localized stresses. As you will learn in future courses, this high stress is also artificially inflated due to the overconstrained fixed support condition provided in this ANSYS model using the simplified fixed face support condition.



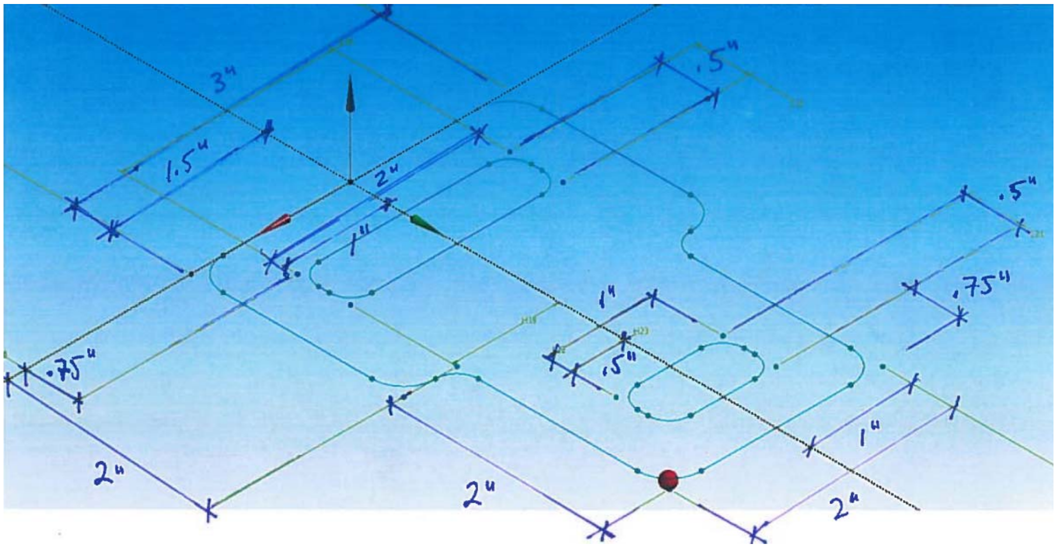
Next Model – Modeling of a Seat-Belt Tab and Investigating Stress Concentrations

Create the following model of the tab part of a seat belt. Set up a new **Static Structural** project in a new **ANSYS** environment. Change the material by *Importing* in 'Elastic-Plastic Lafayette Steel' with the 24,000 psi yield strength limit and Tangent Modulus that was defined in the first part of this Tutorial. Set this new material as the *Default Material* for the material, set the *Units* to US Customary and start up the *Sketching* environment with the *Units* in Inches and with the *Auto Constraints* turned On. To help draw the shape sketch out the larger two rectangle to start **ZX Plane**. Make sure the *Units* are in Inches. Add enough dimension lines to define the width and height of each rectangle and also to help center each rectangle on the axis as shown. Use the Modify Cut option to delete the extra line that exists between your two main rectangles. If too much is cut use the Draw Line tool to get back what is missing before moving on.

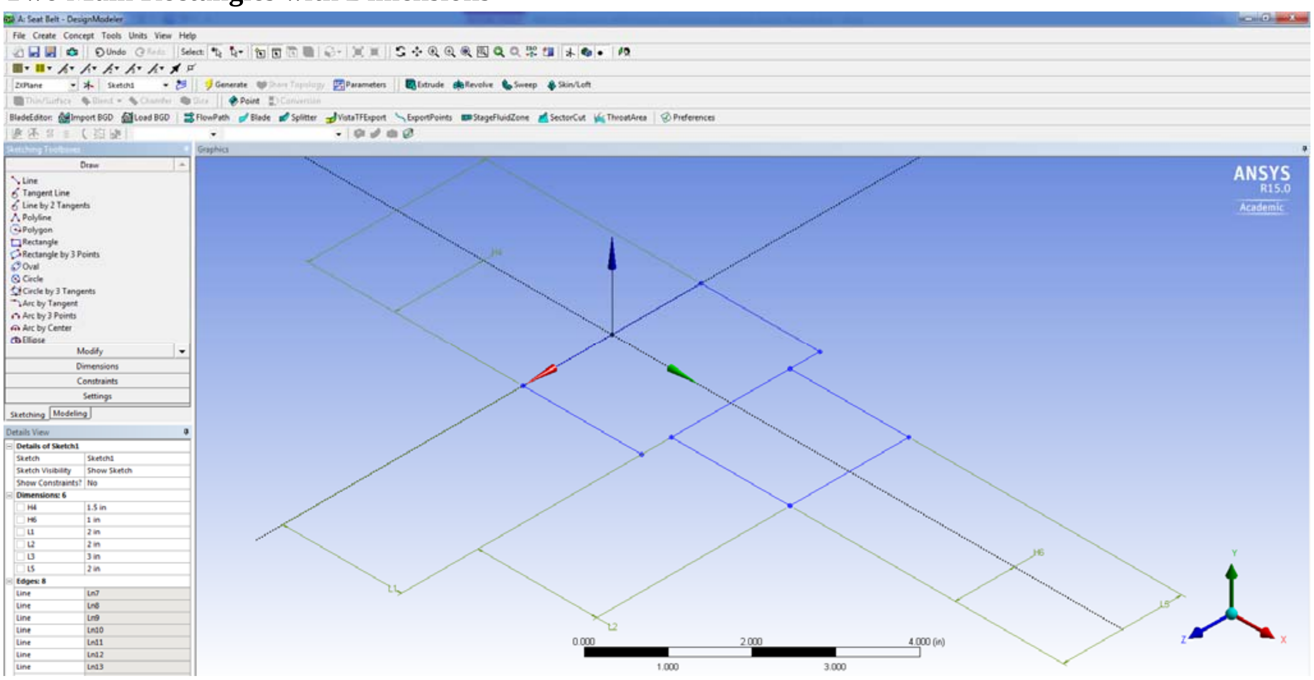
Once these two main rectangles are drawn to scale you can draw two new rectangles to represent the smaller holes. Dimension the hole rectangles for their respective width and length and also add dimension lines that located them from the sketch edges and to center the hole rectangles. The fillets can be added using the Fillet tool in the Sketch Modify options. The fillets are set at 0.3 for most of the outer corners and at 0.2 for all of the inner hole corners. See the view instructions to add the last two fillets to the section in the last view.

See the following figures to help get the geometry correct for the seat belt tab. If you can't get the fillets to work out on the outside of the shape that is okay. Your results for the stresses will still be in the same range for the maximum stresses in the region of the holes. However, you need to get the fillets on the inside hole corners or else you will see much higher stresses in the region of the holes than shown in this tutorial.

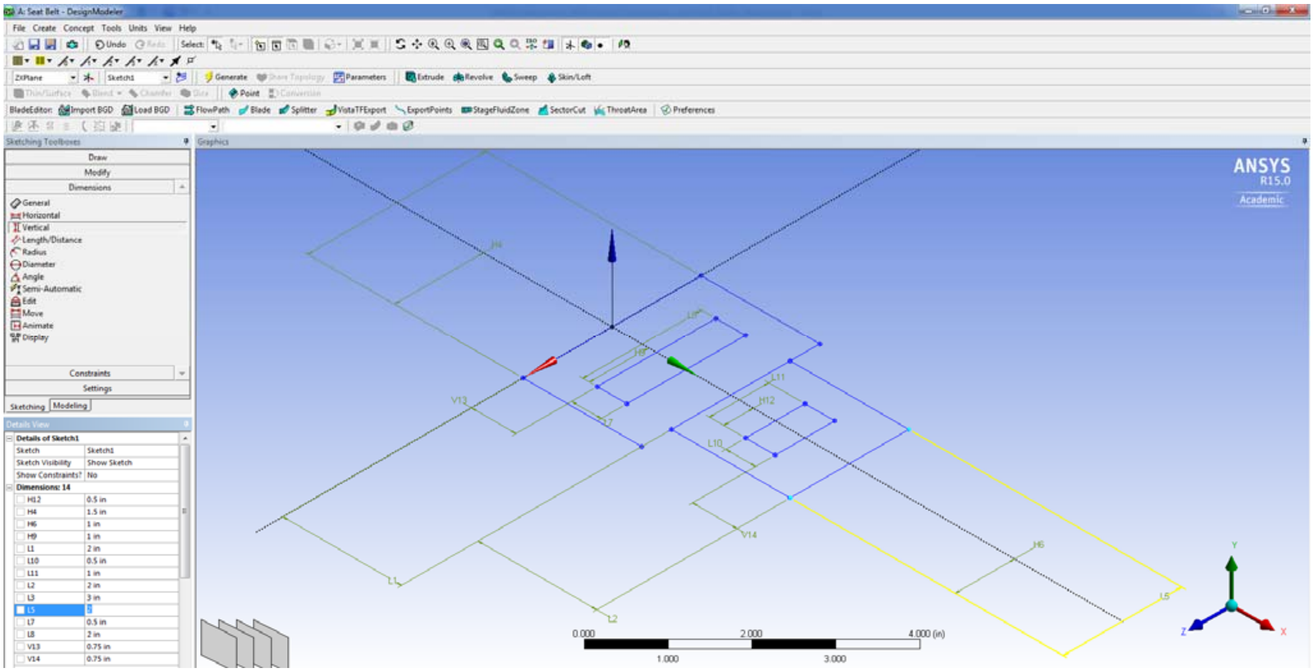
Main Dimensions Run



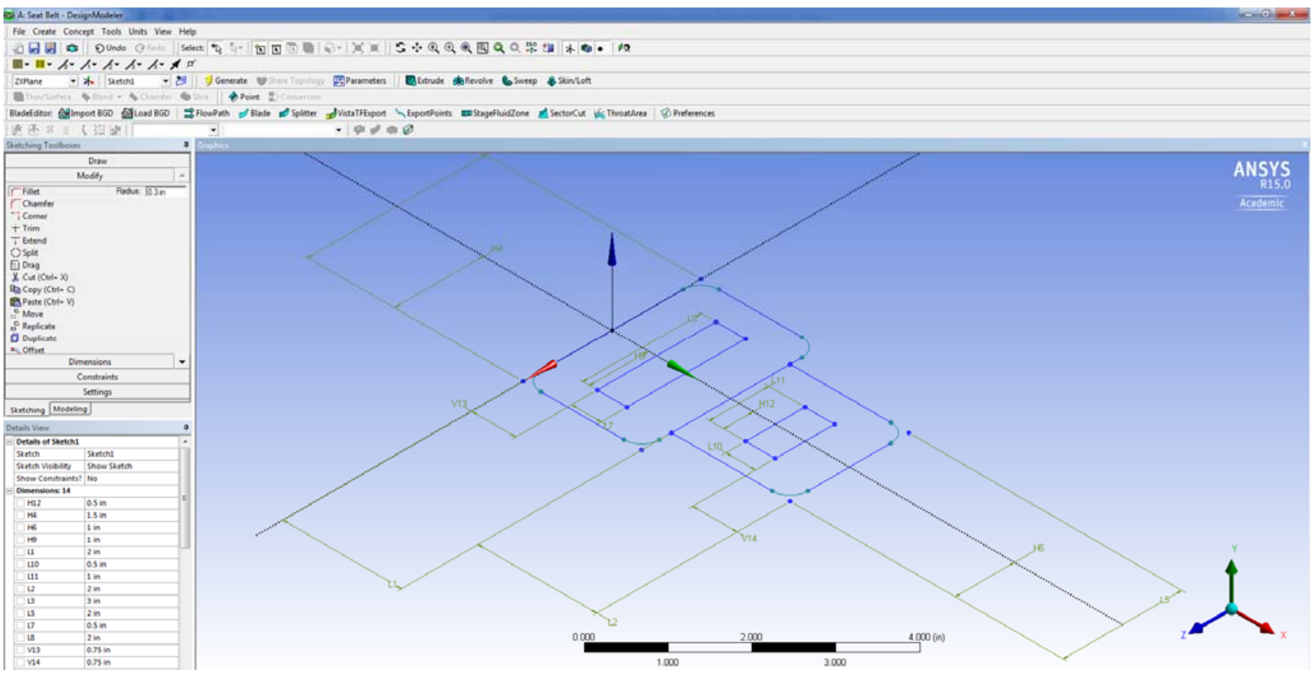
Two Main Rectangles with Dimensions



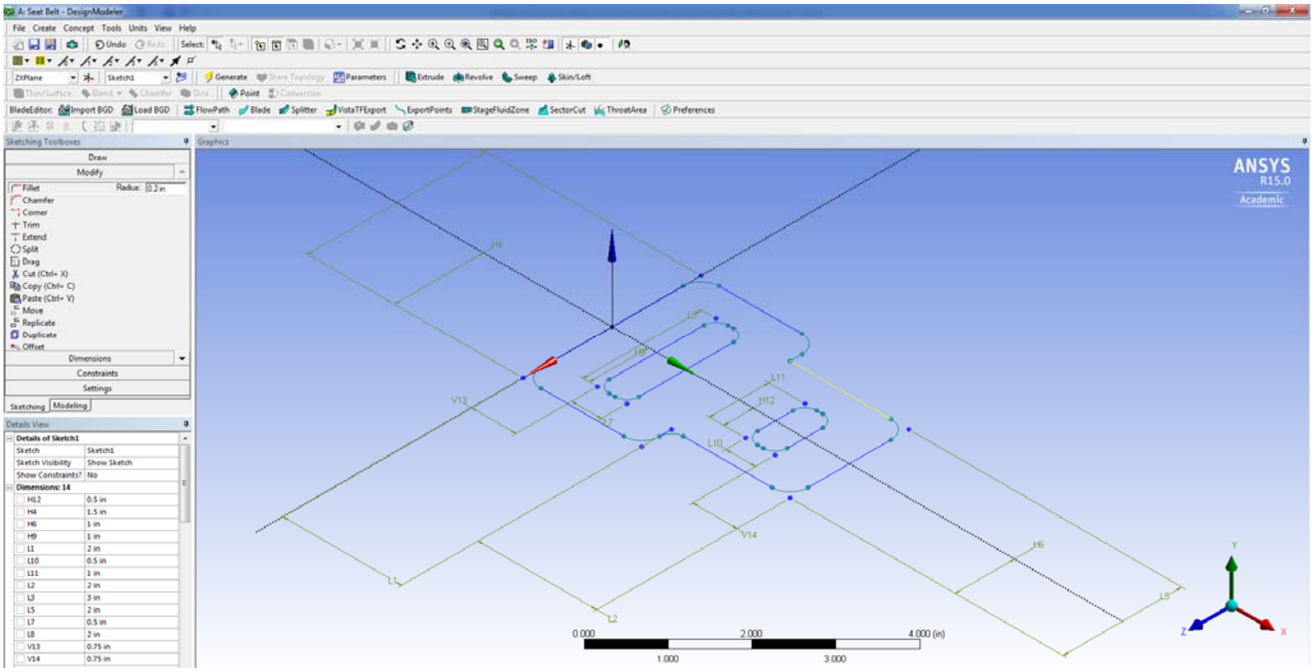
Two Hole Rectangles with Dimensions (use Vertical dimensioning to get the distance from the interior rectangle to the outer rectangle defined)



Add 0.3 inch radius Fillets

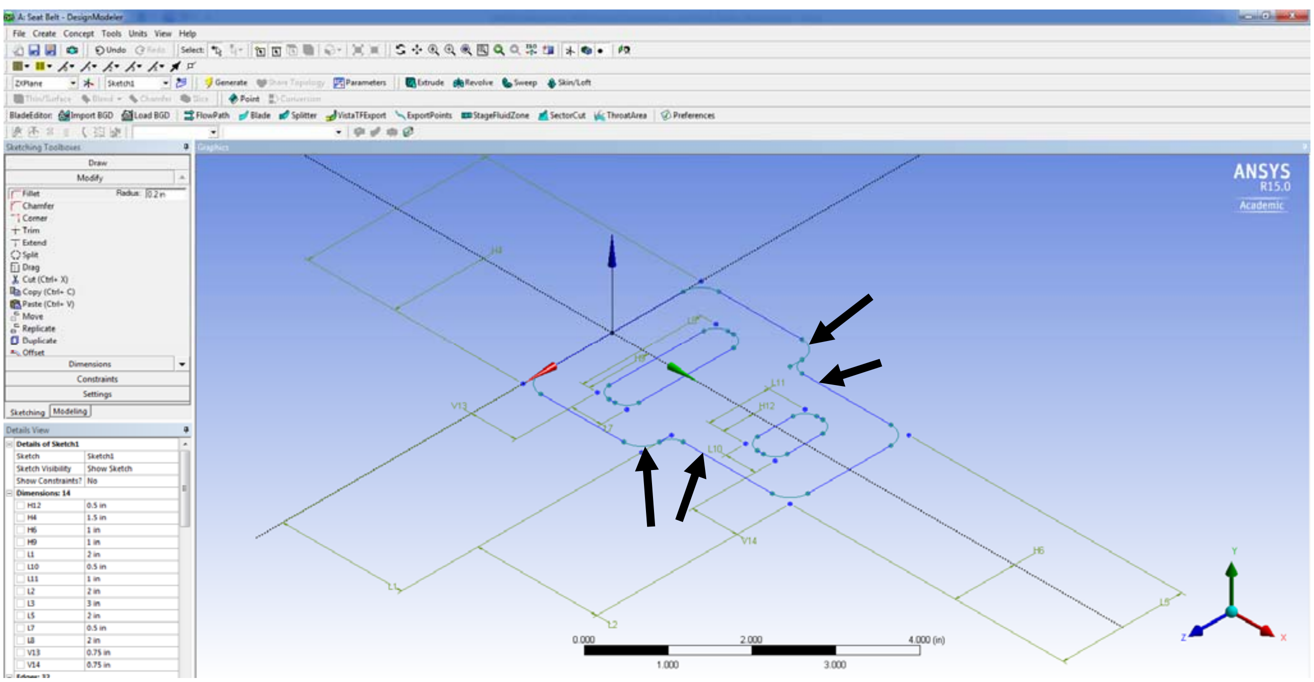


Add 0.2 inch radius Fillets to all the interior hole corners

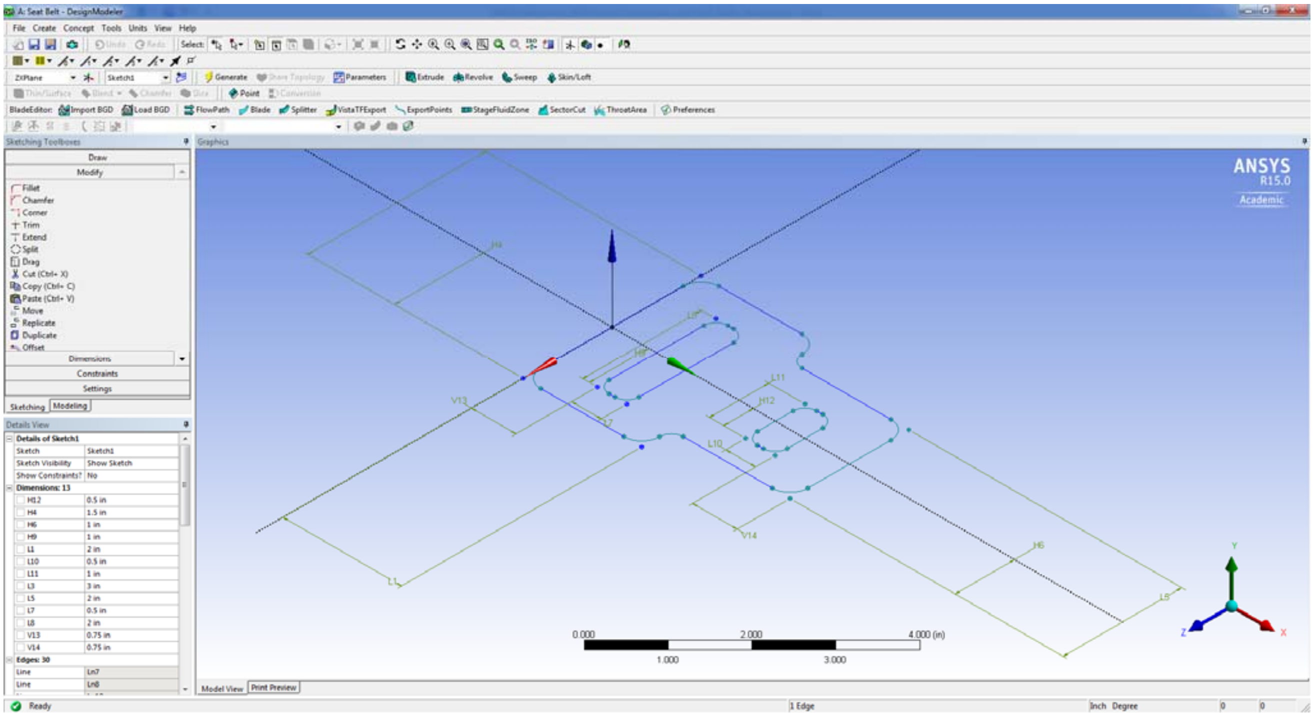


Last Two Fillets – need to select adjacent lines to complete curve in these two places

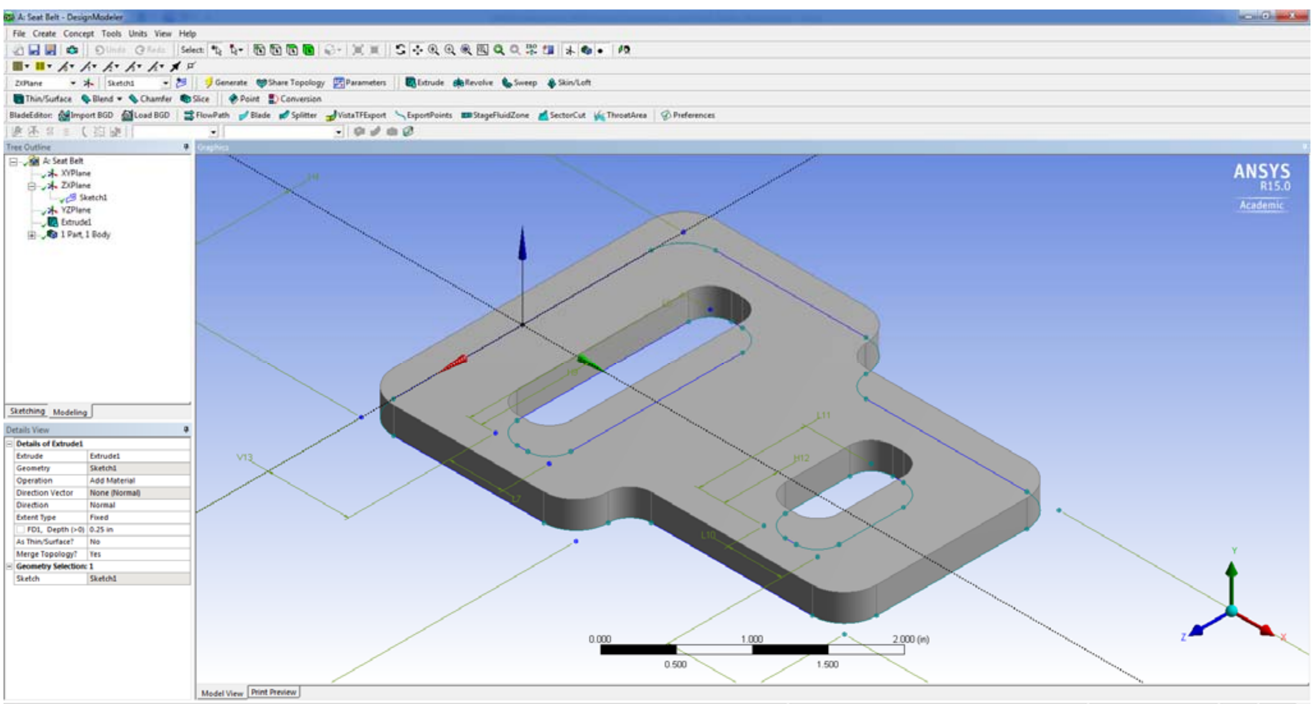
To get the final two curves in select the side line and the previous fillet instead of the little extension side line for each side of the plate. Use the Modify Cut tool to get rid of the little extension side lines on each side.



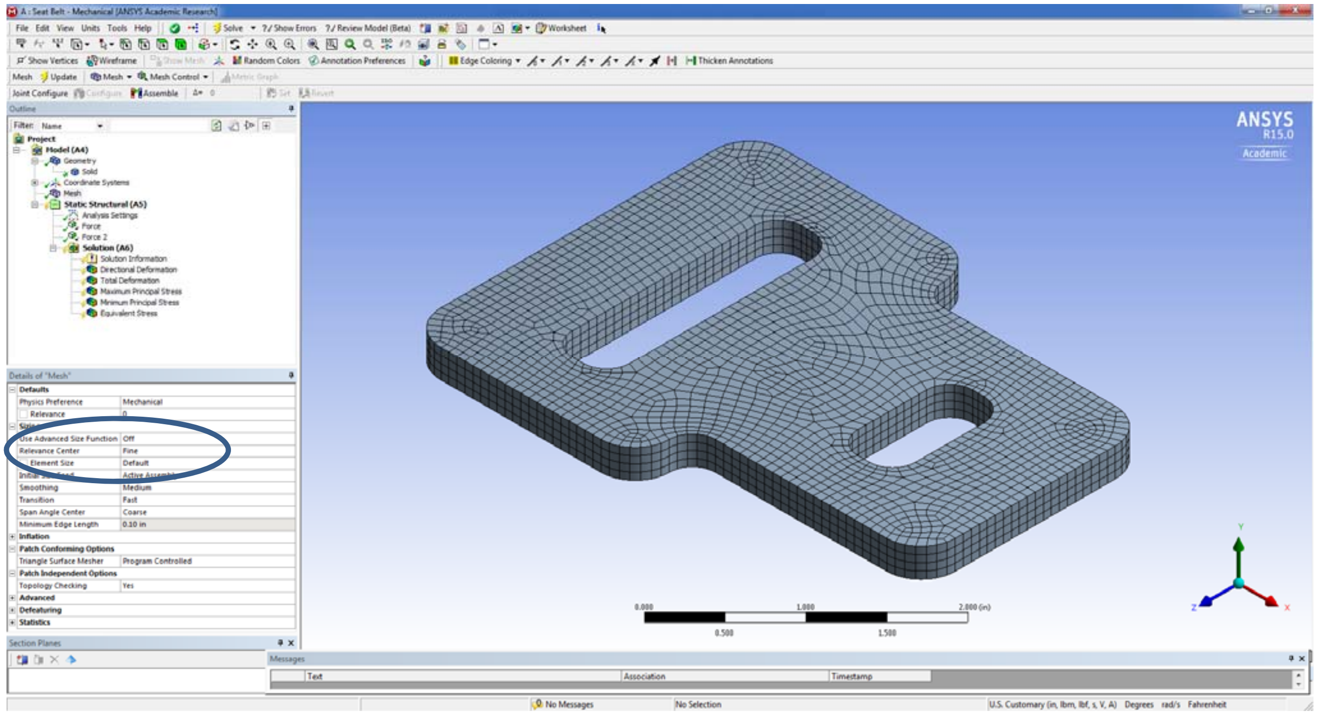
Final Sketch View



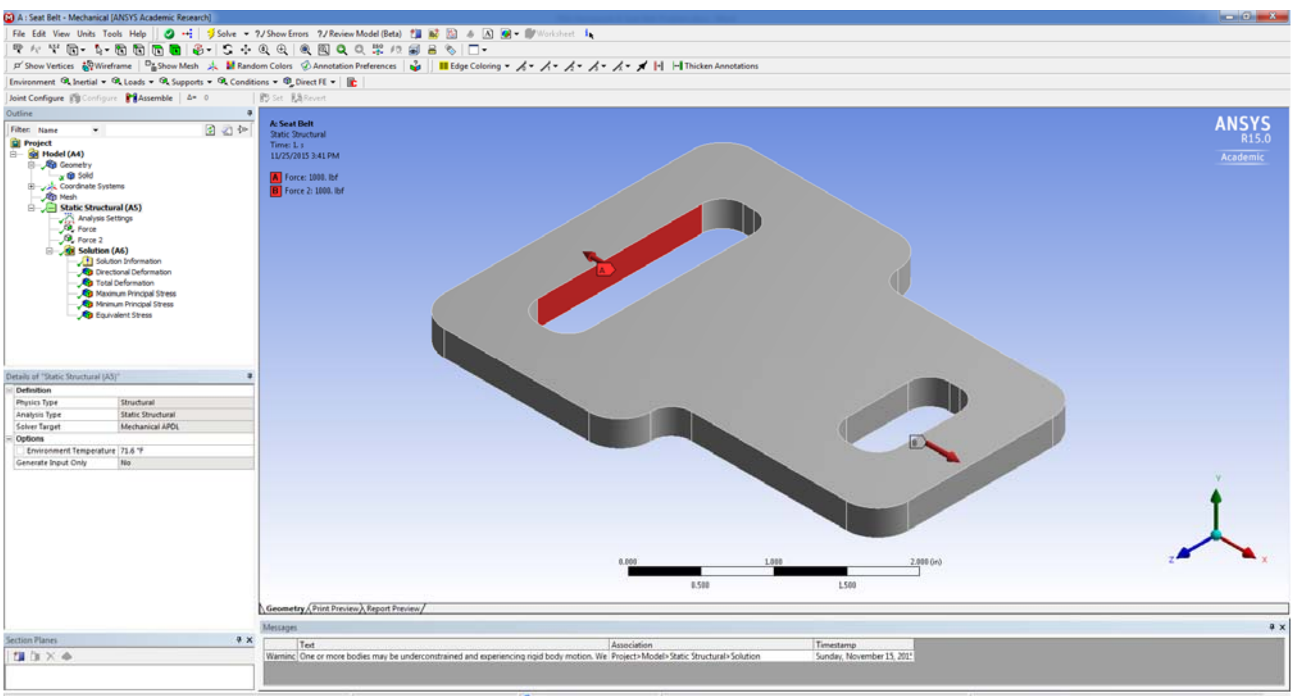
Once your Sketch is complete, **Extrude** the Sketch to a 0.25 inch thickness and then **Generate** the Model Part.



The *Geometry* of the 3D bar is now complete, so open the **Mechanical** Model window to add the mesh, loads, and supports to the bar. **Generate** a *Fine Mesh* for the bar using the **Mesh** options. A refined, finer mesh will give more accurate ANSYS stress results.



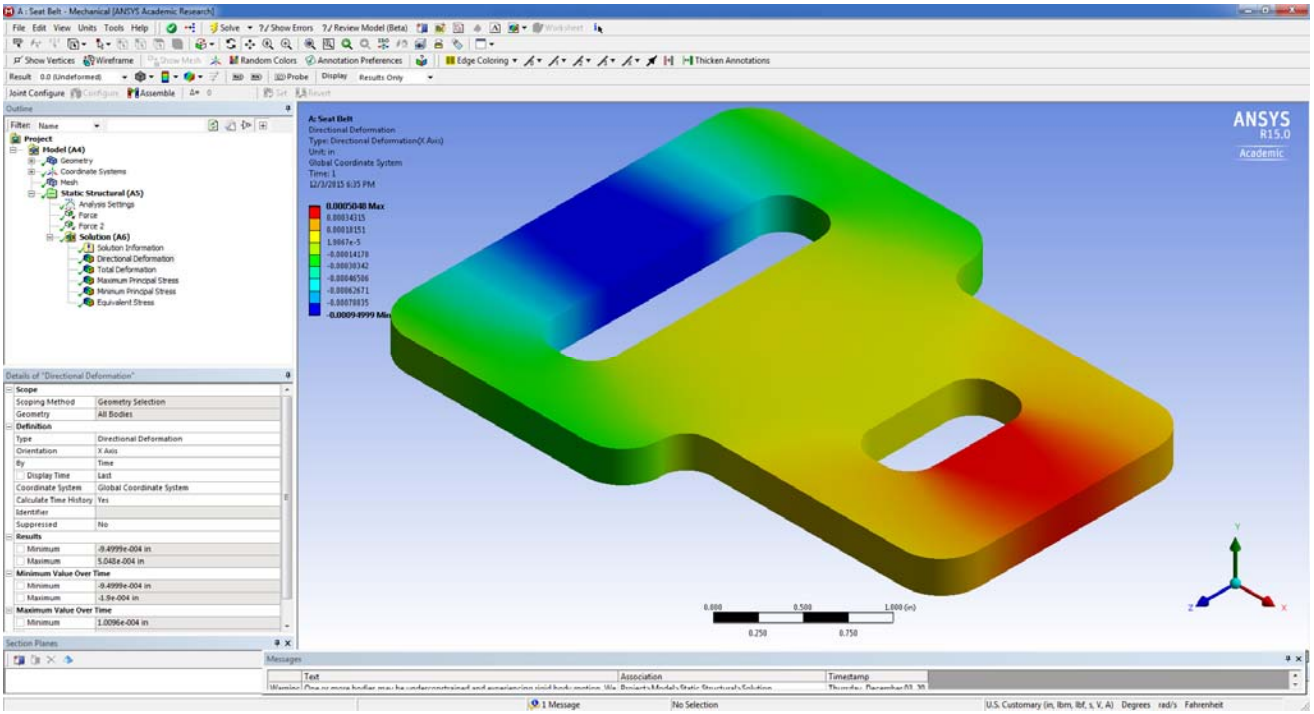
Now add a force of 1000 lb to each of the two interior faces of the cutouts in the seat belt tab in the positive and negative X-axis directions. You should have the following model now.



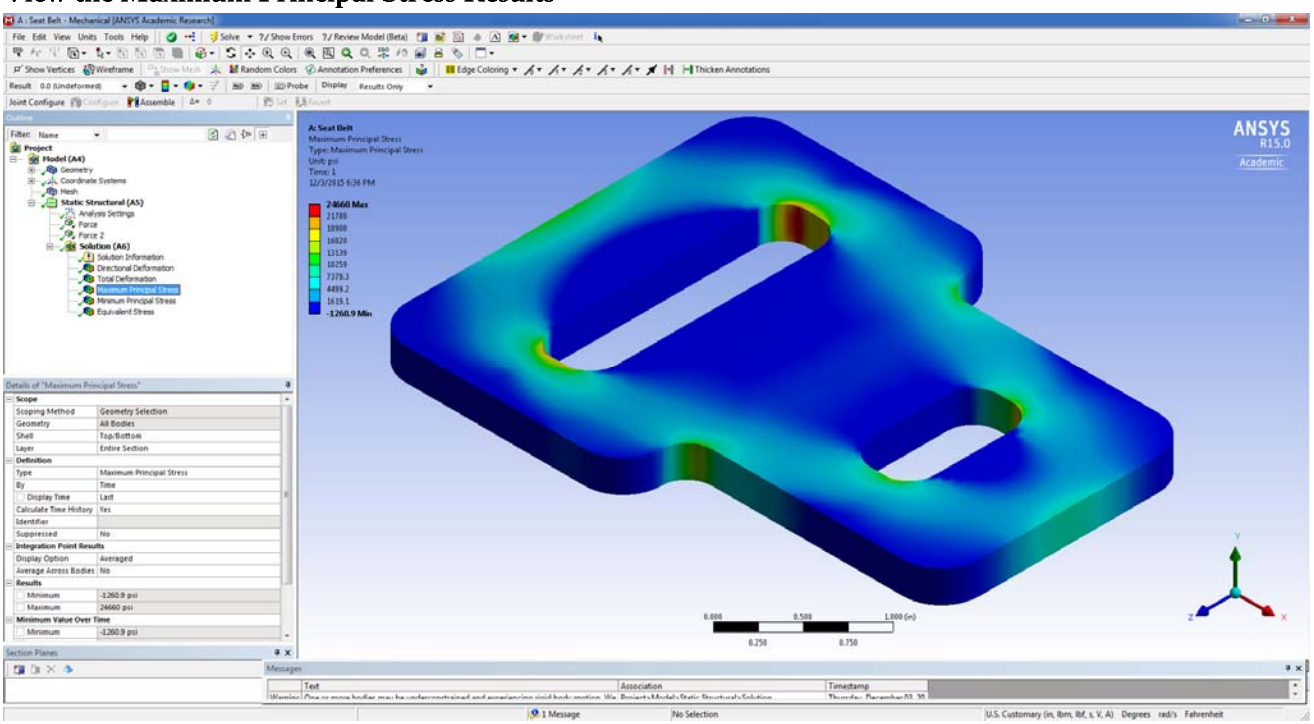
Before hitting **Solve**, add the following *Result Viewers* to the Model: *Maximum Principal Stress, Minimum Principal Stress, and Equivalent Stress Viewers* as well as a *Deformation Directional Viewer* that tracks deformation in the x-axis direction.

Now **Solve** the model and get the deformation, normal stress and shear stress results. It will take a little longer to solve this model since the mesh is finer (i.e. has more elements) then other meshes we have solved.

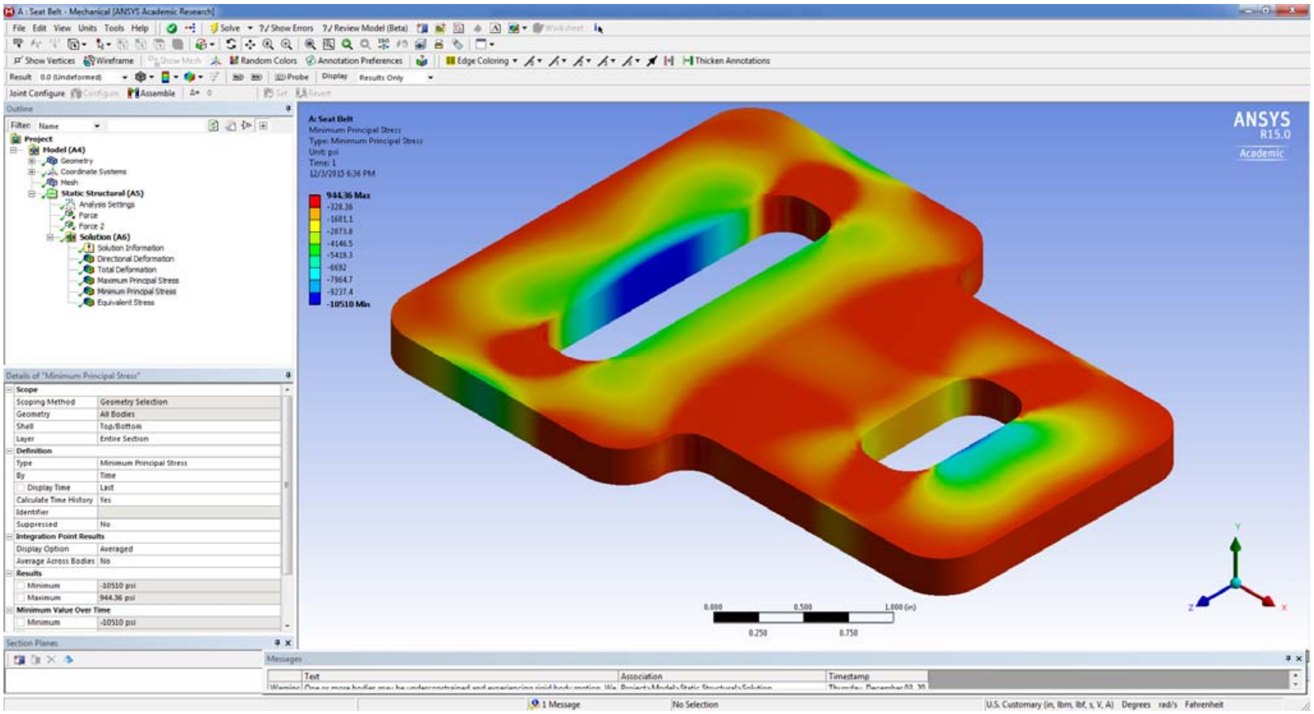
You can't check the deformation results by hand so you will just observe that the deformation behavior looks correct for the seat belt and the loads applied in this model.



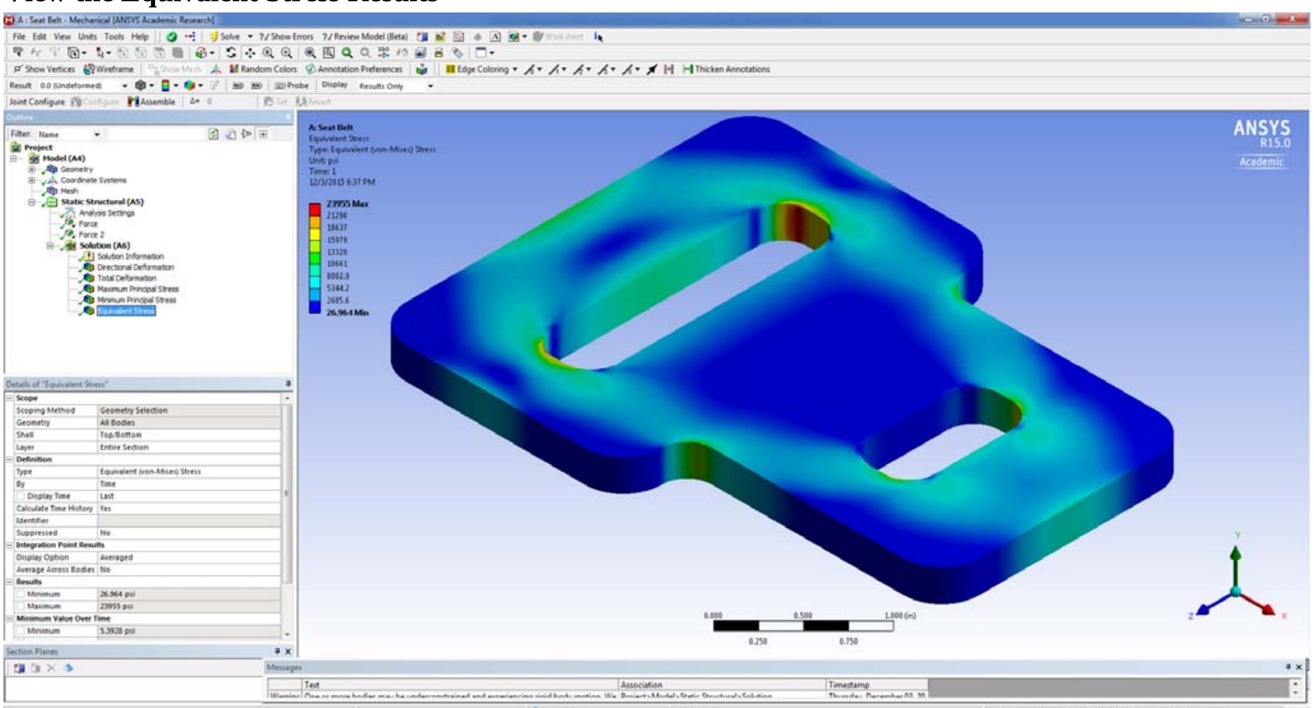
View the Maximum Principal Stress Results



View the Minimum Principal Stress Results



View the Equivalent Stress Results



Note that the **Maximum Equivalent Stress** results provided by ANSYS is right at the set Yield Stress Limit of 24,000 psi, which shows the influence of the elastic-plastic material behavior.

To end the Tutorial process, check to make sure your Equivalent Stresses in your Model match well with those obtained by ANSYS in this Tutorial Example problem as shown in the next view. Matching well means being within a couple of hundred psi.

Important Note: If 24,000 psi was the Ultimate Stress Limit for a Brittle Material that does not yield, then you should would be very concerned with the results obtained. The material would not necessarily fracture immediately when the stress reached the set limit ultimate stress limit since a factor of safety would be involved. However you would want to use a different type of material model if that was the type of material you were working with, so that ANSYS models the correct material behavior and does not allow yielding behavior to dominate when it shouldn't.

Save your Seat Belt Model so that you will be able to modify it for the last ANSYS HW problem.