ME 24-688 – Week 10

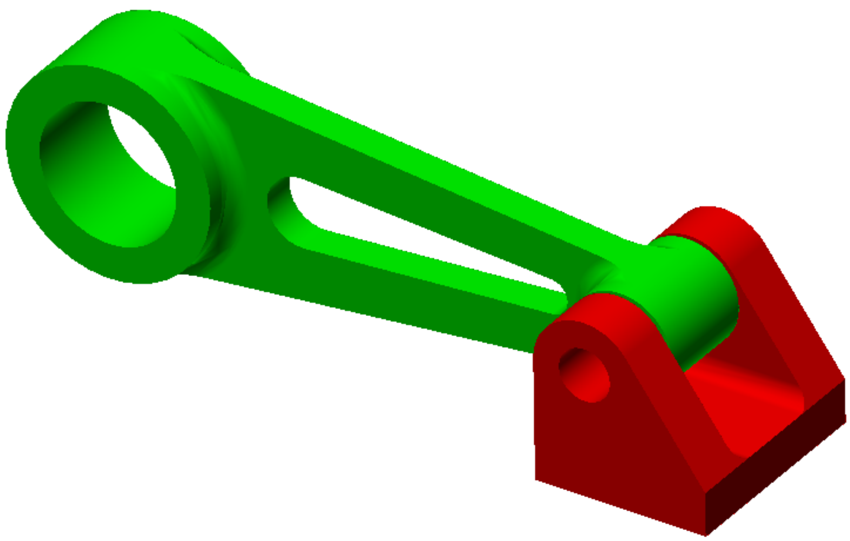
Linear Static Stress

Linear Static Stress

During this project you will learn the general information about performing a linear static stress analysis in Autodesk Simulation Multiphysics. The goal of these projects is to provide an introduction to the Autodesk Simulation Multiphysics application and user interface before advancing to nonlinear simulations.

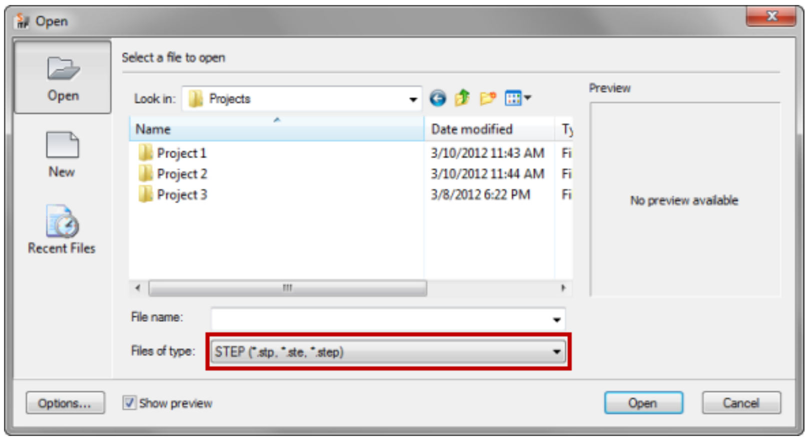
1.1 Project 1 – Yoke and Clevis Assembly

During this project we will conduct a static stress with linear materials analysis on the yoke and clevis assembly. The constraint methods will be fixing the location of the clevis and adding a bolted connection to hold the yoke part to the clevis part. The yoke will then have an 800 lbf load added to the inside of the large diameter hole to complete the setup. Once the analysis is run we will review the results of this linear analysis to complete an introduction to Autodesk Simulation Multiphysics.



1. Click on the **Open** command from the Launch panel.

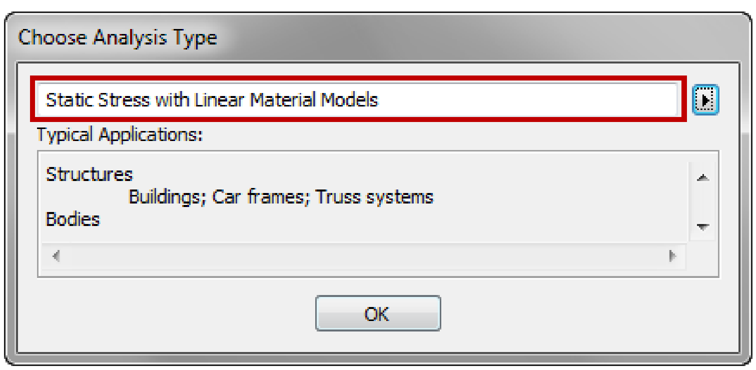
2. Set the **Files of Type** setting to *STEP* to only display STEP files. STEP files are neutral 3D CAD formats exported from most modern CAD systems.



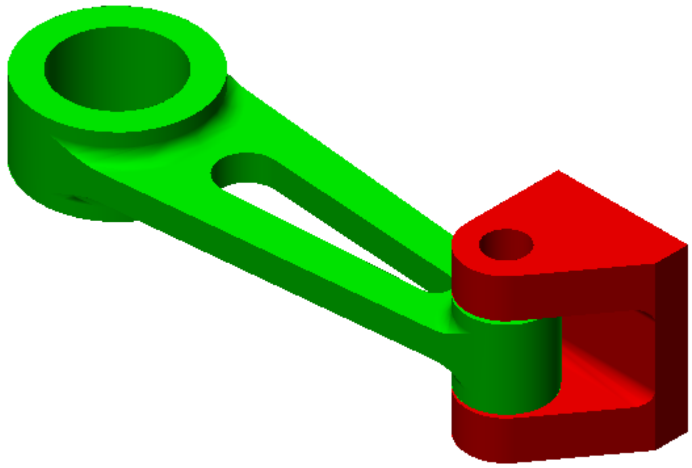
3. Select the "*Yoke and Clevis Assembly.step"* file and click **Open**.

4. Chose the option to "Use STEP file units" to set the length unit value to match the value in the STEP file.

5. Select "*Static Stress with Linear Material Models*" as the **Analysis Type** and click **Ok**.

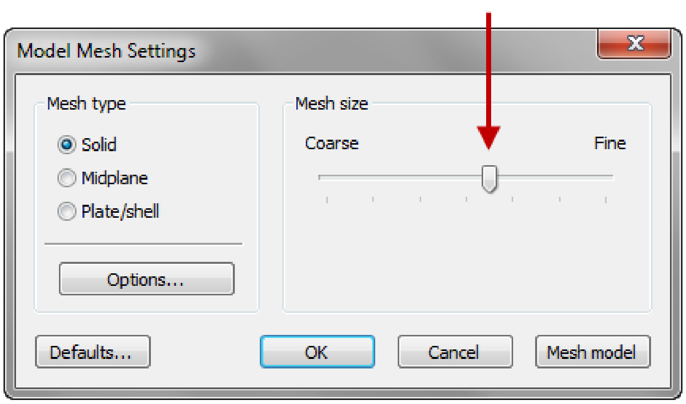


6. The 3D CAD model is now open within Autodesk Simulation and ready to be meshed and setup for the linear analysis.



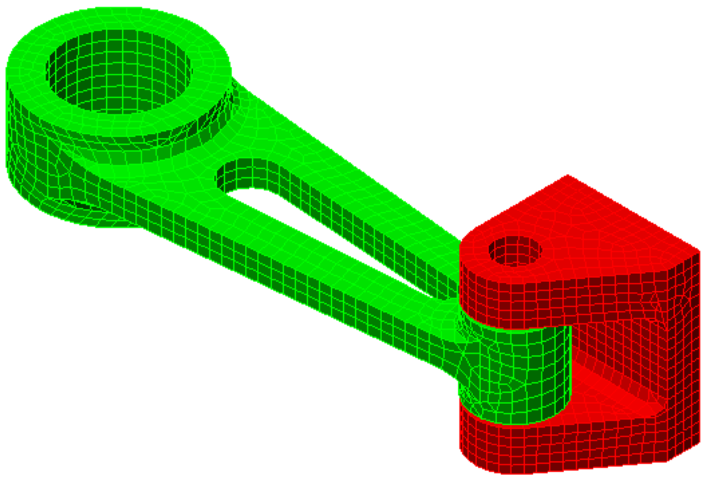
7. One of the first things to complete is the meshing of the model as most other items are established from the mesh instead of the CAD model geometry. Click on the **Mesh tab | Mesh panel | 3D Mesh Settings** to open the **Model Mesh Setting** dialog.

8. Move the **Mesh Size** slider to **85%** to use a finer mesh and ensure the **Mesh Type** is set to **Solid** to use brick elements. The overall model size and specified mesh size value is used to calculate the mesh size.

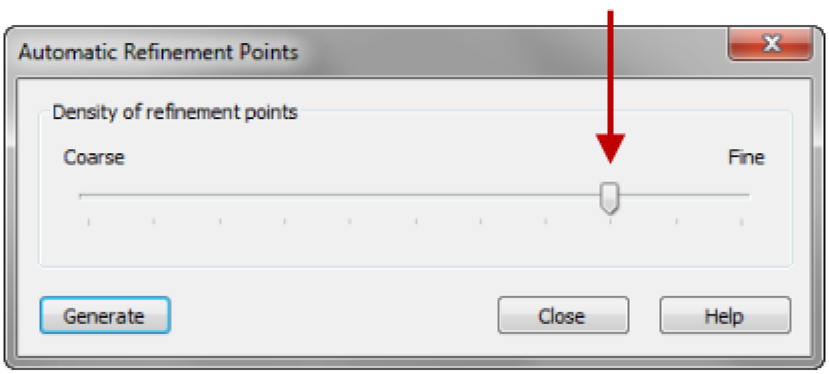


9. Click **Mesh Model** and this will create the mesh on the model and close the **Model Mesh Setting** dialog. When prompted to **View Mesh Results** at this time click **Yes** and you will notice there are over 3500 elements. Click **Close** to continue.

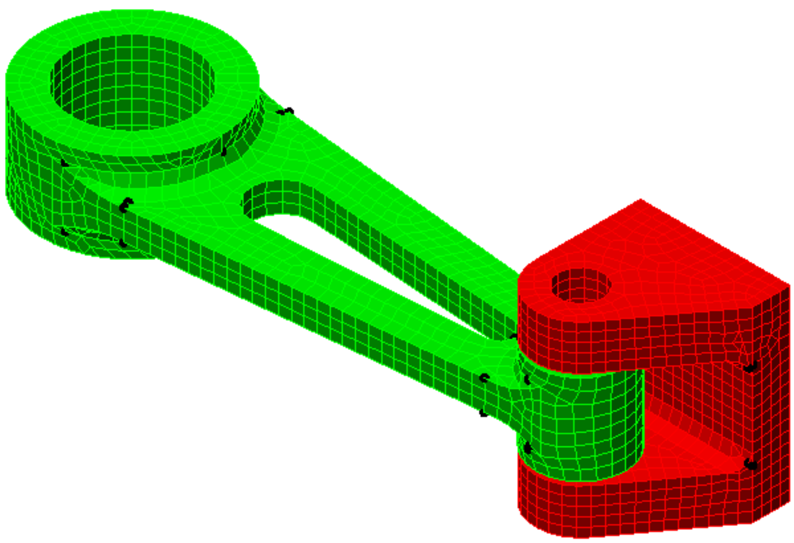
10. Use the **Orbit** command located number the **Navigate panel** within the **View tab** to rotate the model around and review the mesh results. You will notice that the general mesh size is fairly good but is maybe to coarse in some areas around the fillets of the yoke part.



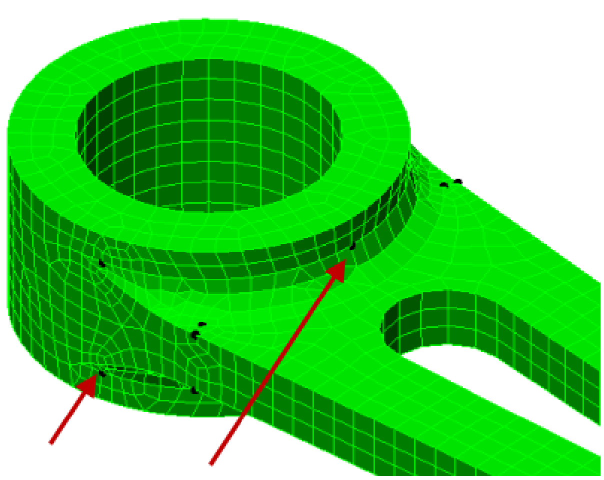
11. To add several refinement points to the model to increase the mesh around key smaller features click **Mesh tab | Refinement Points tab | Automatic** to open the **Automatic Refinement Points** dialog. Move the slider to the right so it is two tick marks away from the right fine end and then click **Generate**.



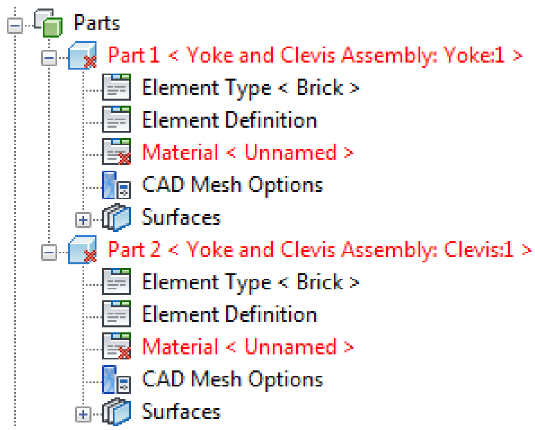
12. This will add 30+ points to the model as shown by block dots in the areas that require additional refinement. These refinement points will be used to create a finer mesh in these areas to ensure more accurate results. Click **Close** to exit the dialog.



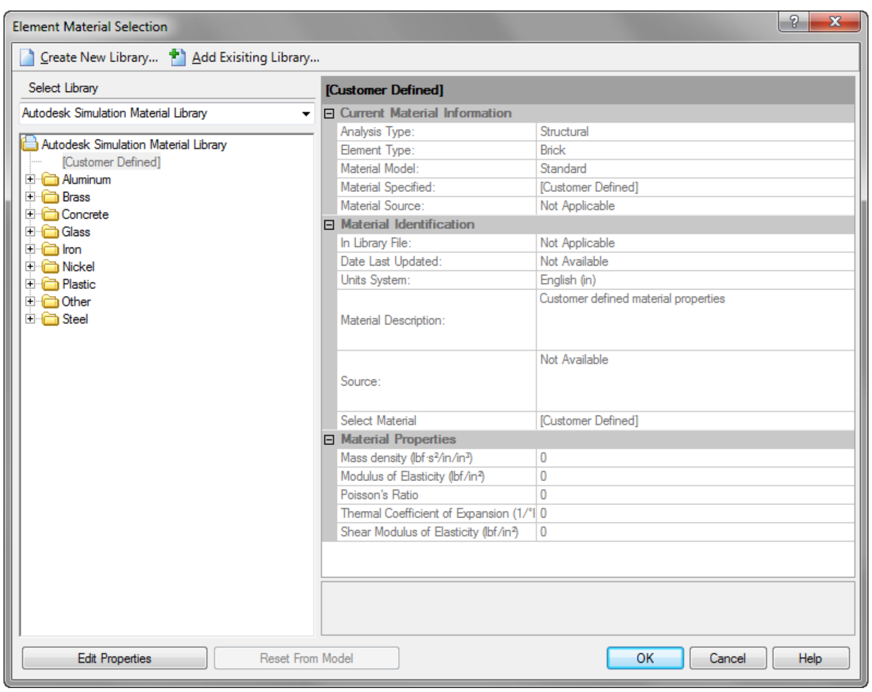
13. To recreate the mesh on the model using these refinement points click **Mesh tab | Mesh panel | Generate 3D Mesh**. This will create a new more refined mesh and click **Yes** to **View Mesh Results**. This will display the **Meshing Result** dialog showing the number of elements in the model which now should have over 4000 elements. Once complete click the **Close** button to exit.



14. Now that the model has been meshed you will notice that the **Element Type** and **Element Definition** have been established in the **FEA Editor Browser** for each part. These can be edited by right-clicking on them from the Browser if needed.



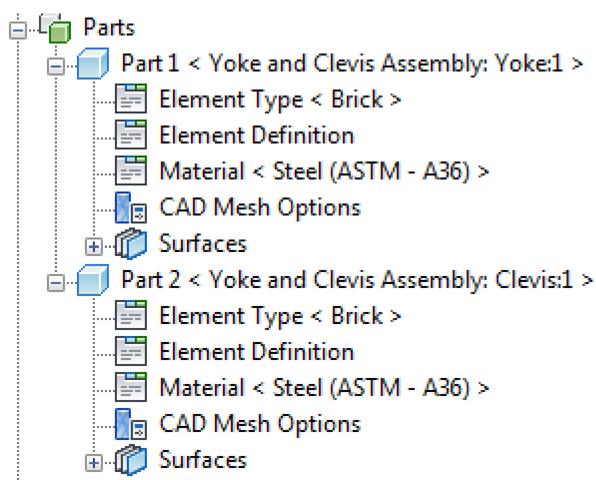
15. To set the material for each part go to the **FEA Editor Browser** and right-click on the **Material** heading of *Part 1* and select Edit Material. This will open the **Element Material Selection** dialog where you can assign, create, and edit materials to parts.



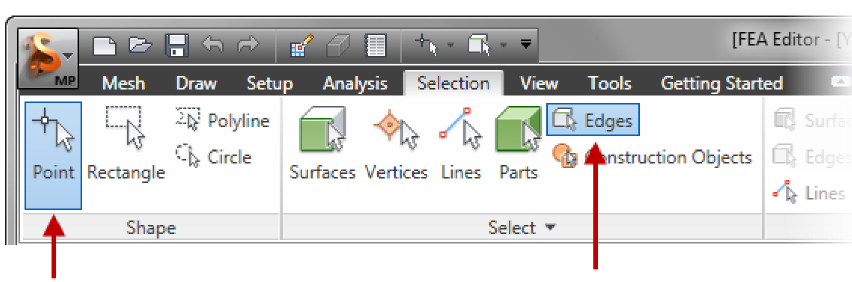
16. Expand the *Steel* folder and then expand the *ASTM* folder. Select the "*Steel (ASTM-A36)*" material by clicking on the name. Click the **OK** button to exit and assign this part to the part.

17. Repeat the process to assign the material "*Steel (ASTM-A36)*" to *Part 2* of the model.

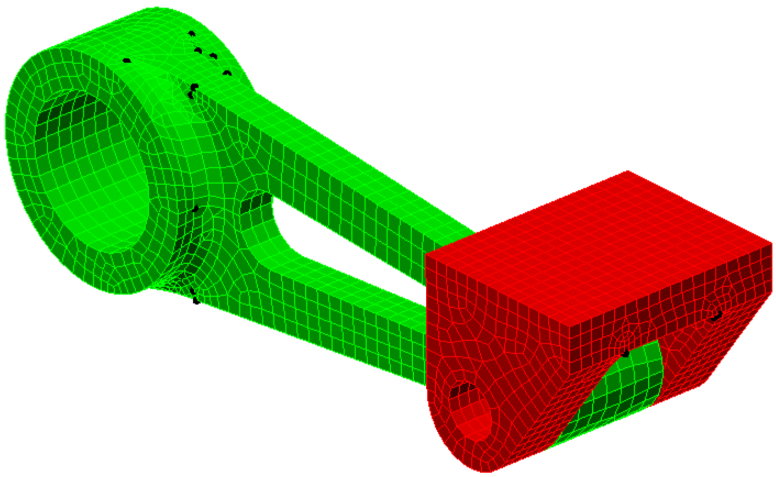
18. Once complete you will see the new material types assigned to the parts in the **FEA Editor Browser** as shown below. These material values will be used throughout the analysis and can be changed in the same manor if required.



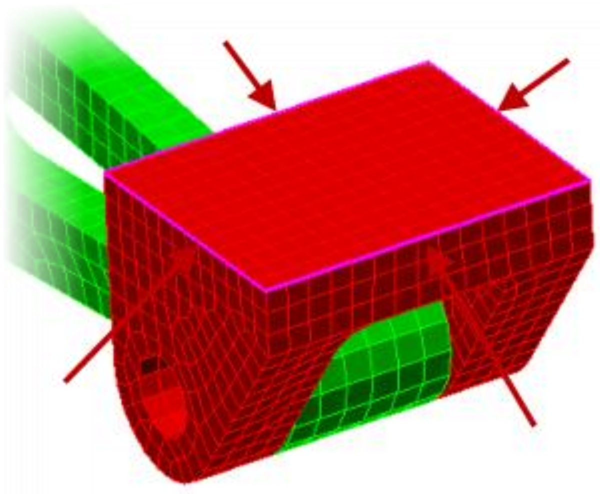
19. Before assigning the constraints click on the **Selection** tab. Here you will see the **Shape** and **Select** panels. Located on these two panels are the selection settings that control how items are selected on the mesh model. Select **Point** from the **Shape** panel so single points on the mesh model. Then select **Edges** from the **Select** panel to only select the edges from the point clicked.



20. **Orbit** the model so the bottom of the *Clevis* part can be viewed as shown below. Press ESC to exit **Orbit** when done.

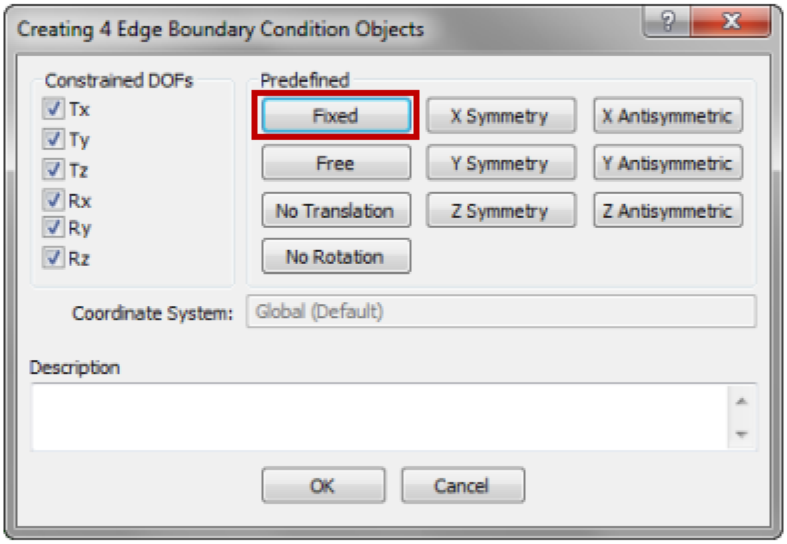


21. Select one of the bottom edges of the *Clevis* part and then hold down the CTRL key to also select the three other edges as shown below.



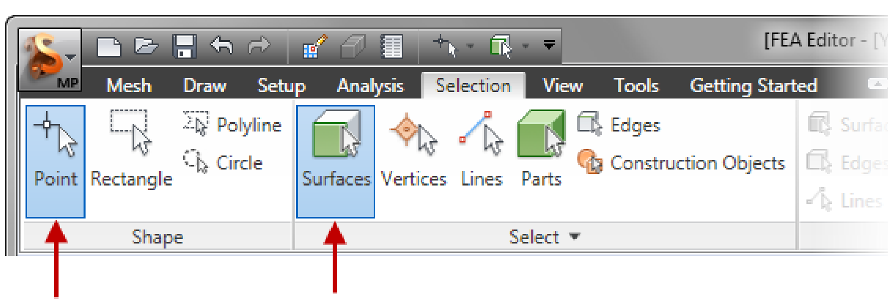
22. Select **Setup tab | Constraints panel | General Constraint** to open the **Boundary Condition Object** dialog.

23. Select **Fixed** as the **Predefined** setting. This will lock all six (6) degrees of freedom on the nodes of the four (4) selected edges preventing them from moving or rotating as shown in the **Constrained DOFs** values. Click **OK** to complete the assignment.



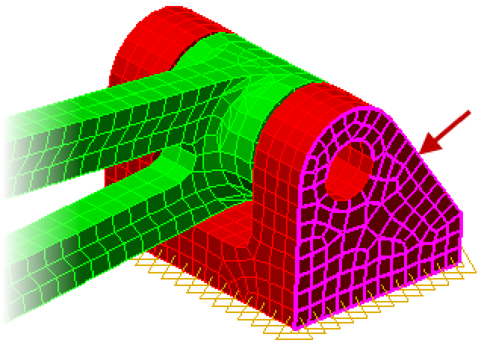
24. The *Yoke* part is held assembled to the *Clevis* part by a bolt in this case. To simulate the bolted connection that is not in the CAD model we will use the **Bolt** command available within Autodesk Simulation.

25. Change your selection setting to have the **Shape** set to **Point** and the **Select** set to **Surfaces** to select a complete surface of the mesh model.

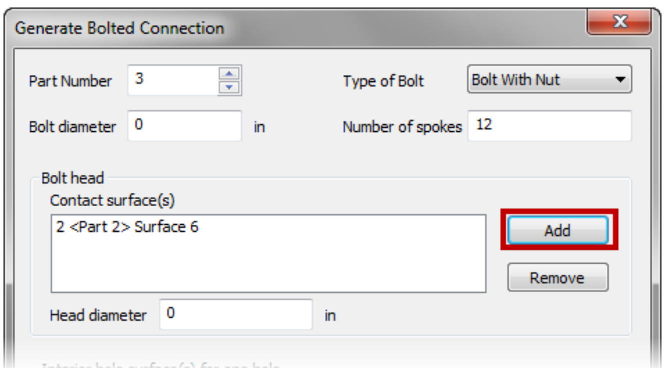


26. Start the bolted connection command by click **Mesh tab | CAD Additions panel | Bolt**.

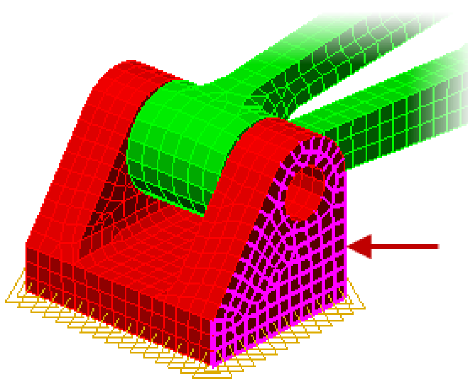
27. Select the outer surface of the *Clevis* part as shown below.



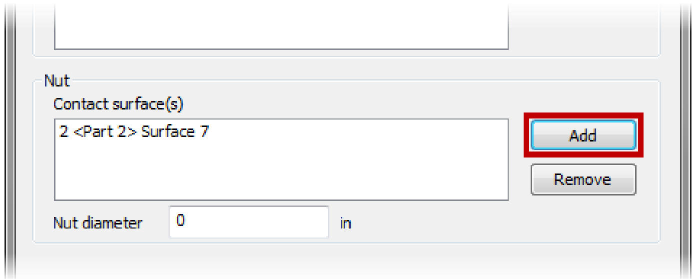
28. Click the **Add** button within the **Bolt Head** section of the **Generate Bolted Connection** dialog to add the selected surface as the bolt head surface.



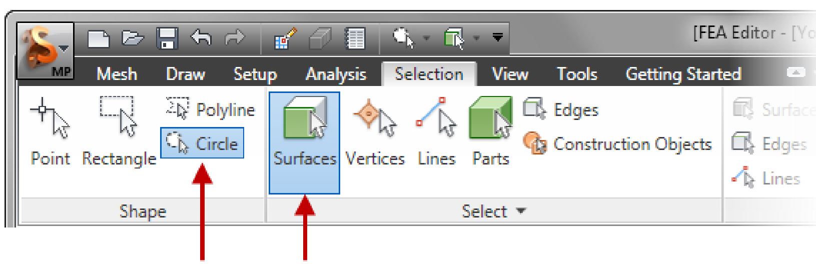
29. Select the opposite outer surface of the *Clevis* part.



30. Click the **Add** button within the **Nut** section of the **Generate Bolted Connection** dialog to add the selected surface as the surface the nut would contact.

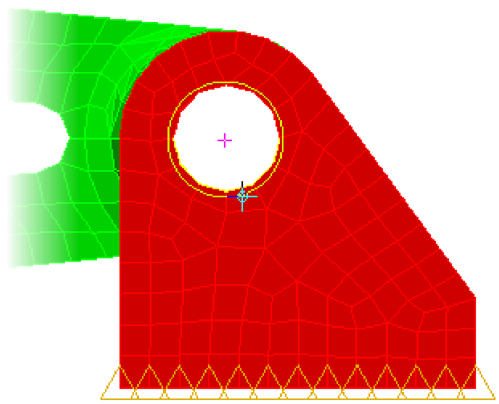


31. Set the selection to use **Circle** as the **Shape** and **Surfaces** as the **Select** value.

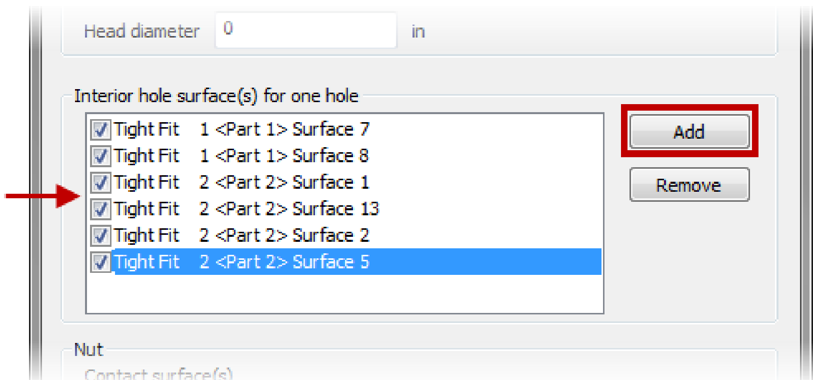


32. Go to the **View Cube** and select the **Bottom** view to view the model.

33. Position your cursor in the center of the clevis hole and drag a circle out just past the diameter of the hole to select all of the surfaces within this circle selection.



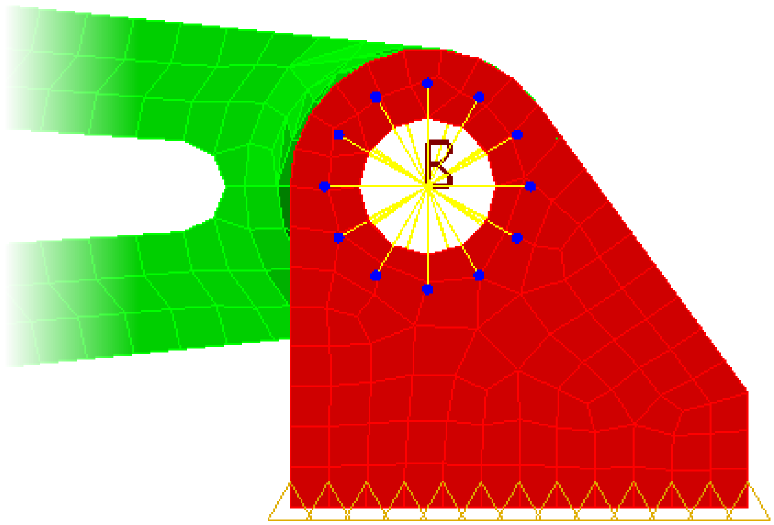
34. Click the **Add** button in the **Interior Hole Surfaces for One Hole** selection to add the six (6) selected surfaces. Then check the **Tight Fit** option next to each surface to indicate the bolt is a tight fit into the hole surfaces.



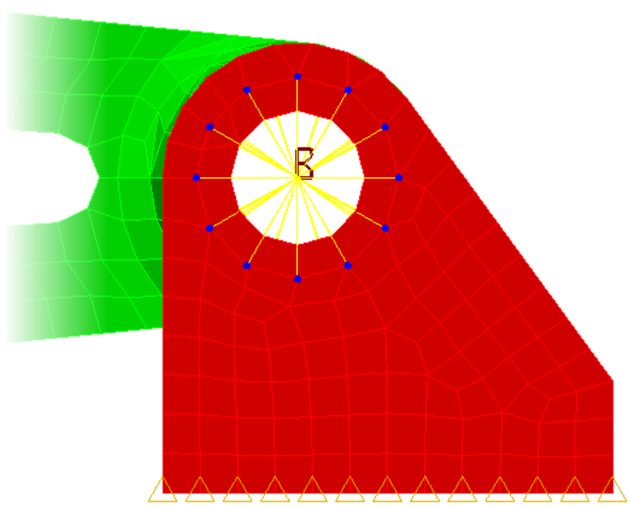
35. Complete the following values to define the size of the bolt:

- Enter 0.75 into the **Bolt Diameter** field
- Enter 1.125 into the **Head Diameter** field
- Enter 1.125 into the **Nut Diameter** field
- Enter 500 into the **Magnitude** field for the **Axial Force**

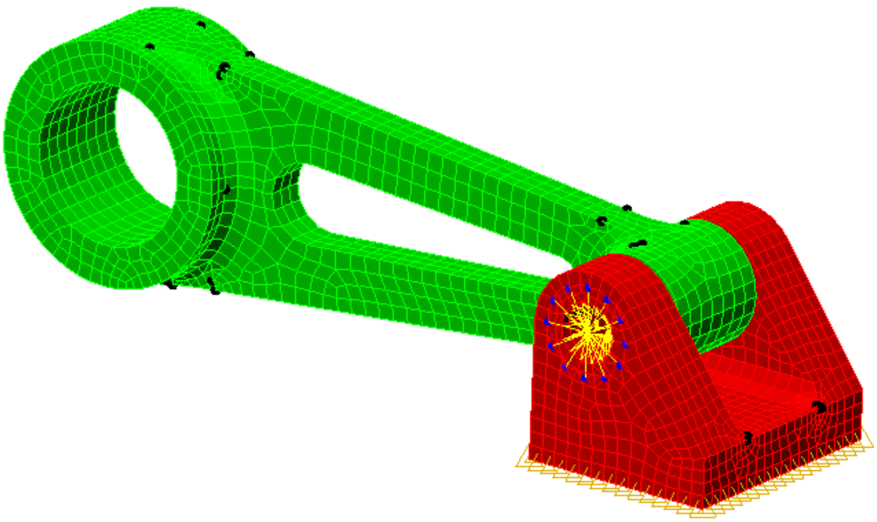
36. Click the **OK** button to complete the bolted connection. You will see several beam elements created within a new part that will represent the bolt and nut within the model. The blue dots are element nodes created at construction vertices that represent the bolt head and nut diameters.



37. Now we need to regenerate the mesh so that nodes will be created on the surface mesh of the clevis to connect to the new spoke nodes. The purpose of these construction vertices is to force the creation of a node at the same spot on the surface. To regenerate the mesh click **Mesh tab | Mesh panel | Generate 3D Mesh**.

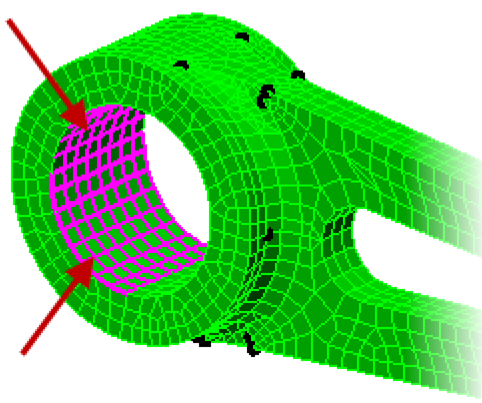


38. **Orbit** your model to a similar position as shown below so you can see the inside of the large diameter end of the *Yoke* part.

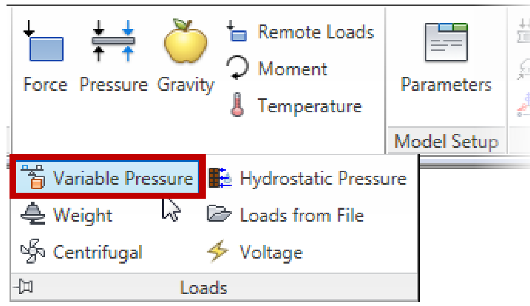


39. Change your selection settings to use **Point** as the **Shape** and **Surfaces** as the **Select** value.

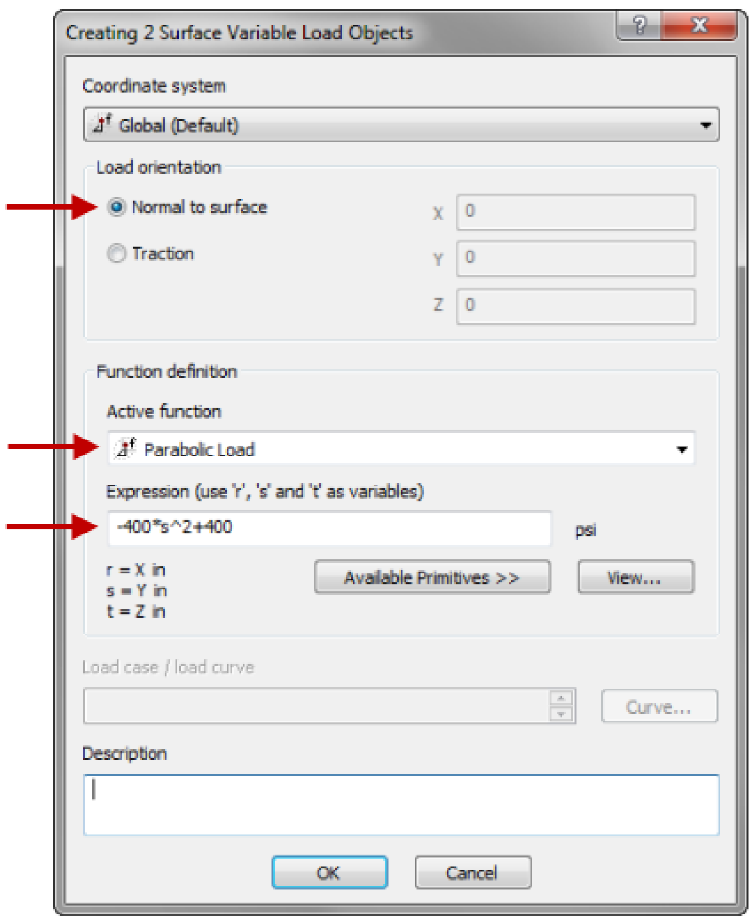
40. Select the two (2) left side inner surfaces of the large diameter hole.



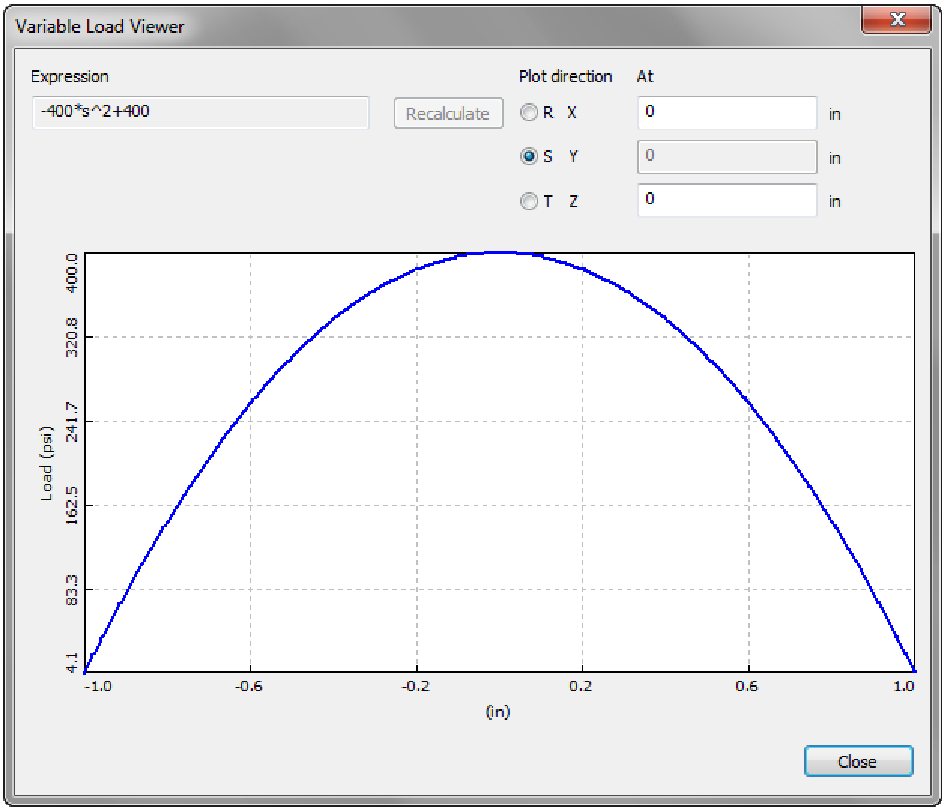
41. Click the down arrow on the **Loads panel** of the **Setup tab**. Select the **Variable Pressure** load to start the assignment of a parabolic profile pressure load.



42. Select **Normal to Surface** for the **Load Orientation** then enter "*Parabolic Load*" as the **Active Function** value. For the **Expression** enter "-400*s^2+400".



43. Click the **View** button to open the **Variable Load Viewer** dialog. Then select **S / Y** as the **Plot Direction**.

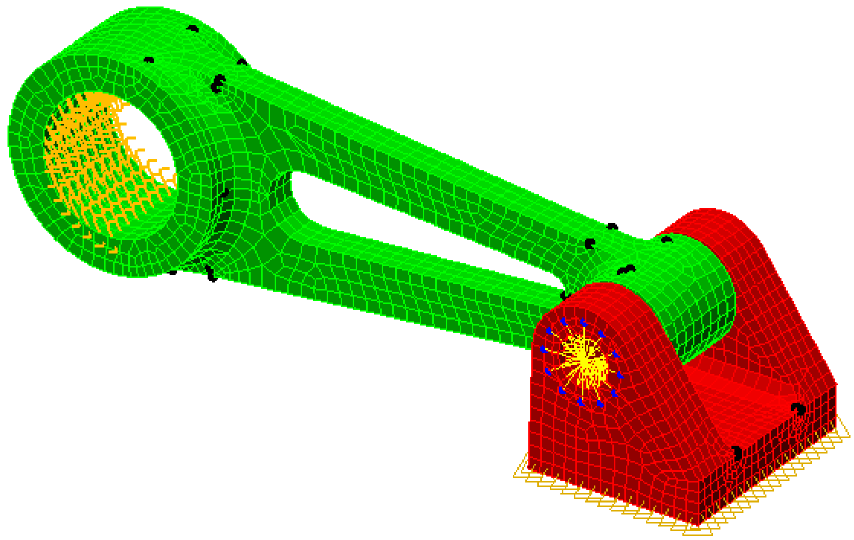


44. You will notice the variable pressure load will ensure that the load magnitude will be 0 at the ends of the diameter. Click **Close** to exit the dialog.

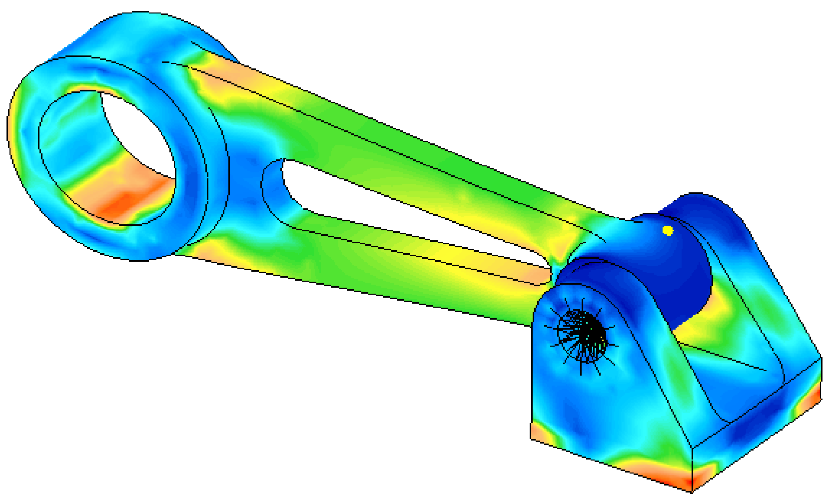
45. Click **OK** to complete the load assignment.

46. Earlier when the bolt connection as added a third part was added to the model representing the bolt. Assign the material *Steel (AISI 4130)* to this *Part 3* component in the FEA Browser.

47. The model is now ready to be solved.



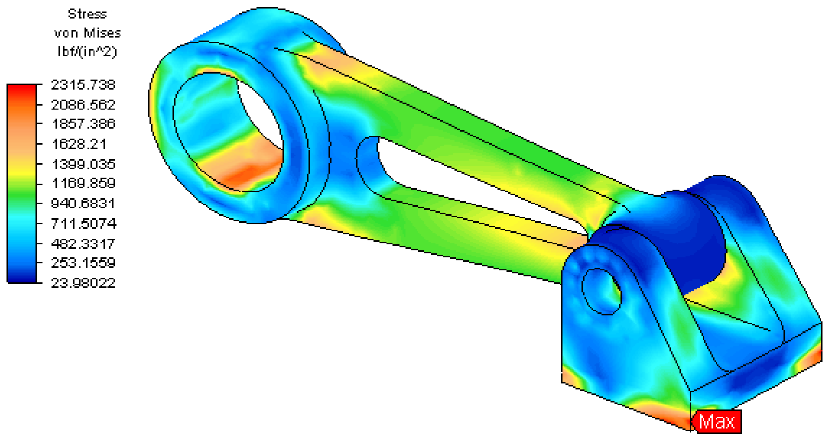
48. The model is now ready to be solved. Click **Analysis tab | Analysis panel | Run Simulation** to start solving the model.



49. Select Part 3 heading in the Browser and right-click **Visibility** off to hide the bolt.

50. Ensure that **Von Mises** is selected in the **Results Contours tab | Stress panel**. This will show the max stress of the model.

51. To display the maximum stress location click on **Results Inquire tab | Probes panel | Maximum**. This will place a probe at the maximum location of the model.



52. To display the displacement of the model click **Results Contours tab | Displacement panel | Displacement Magnitude**. The maximum probe will now show the new location which has a maximum displacement of 0.0008 in.