ME 24-688 – Week 12

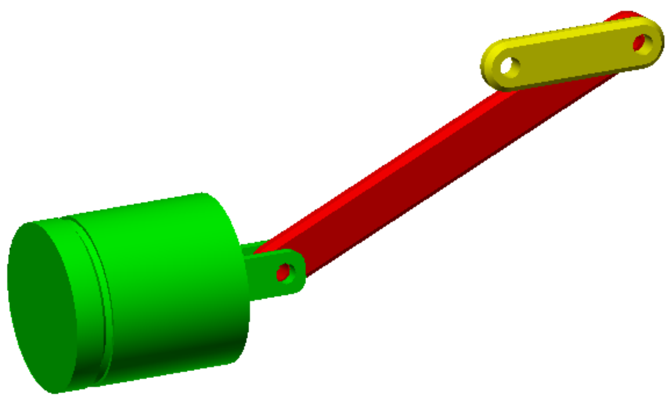
Piston Mechanical Event Simulation

Introduction to Mechanical Event Simulation

The purpose of this introduction to Mechanical Event Simulation (MES) project is to explorer the dynamic simulation environment of Autodesk Simulation. This environment allows you to perform dynamic simulations of designs. This type of simulation is used on assemblies to simulation time varying behavior of a system.

1.1 Project 2 – Engine Piston Simulation

In this project, you will step through the meshing of the engine piston assembly shown below. We will also create joints to connect the parts and to provide points for the crank part. Autodesk Simulation will be used for this mechanical event simulation (MES).

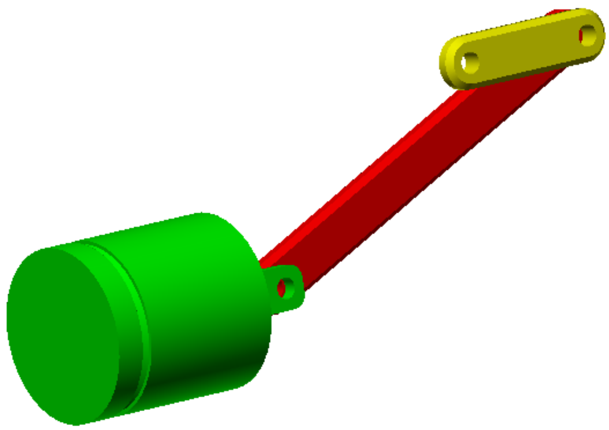


1. Open the *Piston Assembly.stp* file in Autodesk Simulation. You will need to change the **Files of Type** to **STEP (*.stp, *.ste, *.step)**.

2. Click **OK** to accept the default settings for the **Select Length Units** dialog box. This will keep the units set to inch from the STEP file.

3. Select **Nonlinear: MES with Nonlinear Material Models** for the analysis type in the **Choose Analysis Type** dialog. Click **OK** to complete opening the file in the **FEA Editor** environment.

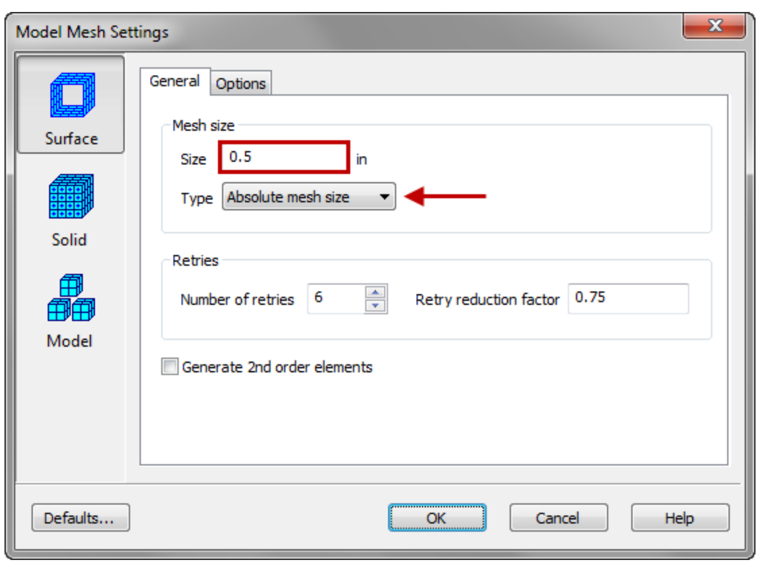
4. The engine piston model is now open as shown below.



5. To begin meshing the model click **Mesh tab | Mesh panel | 3D Mesh Settings**.

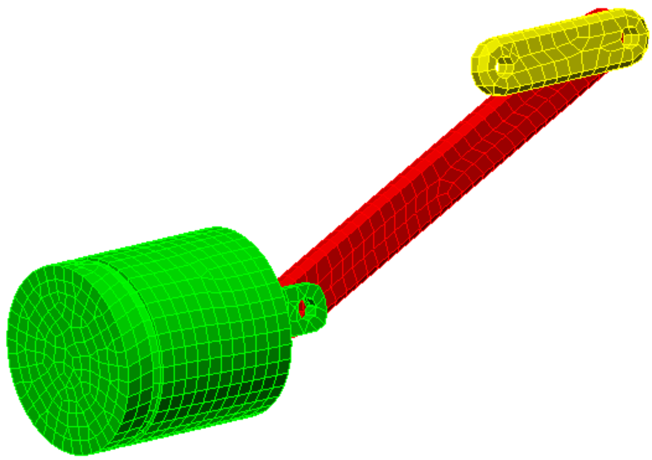
6. Click the **Options** button in the **Model Mesh Settings** dialog.

7. Select **Absolute Mesh Size** as the type and enter **0.5** as the mesh size. This will produce a consistent mesh size throughout the model. Click **OK** to complete setting the options.



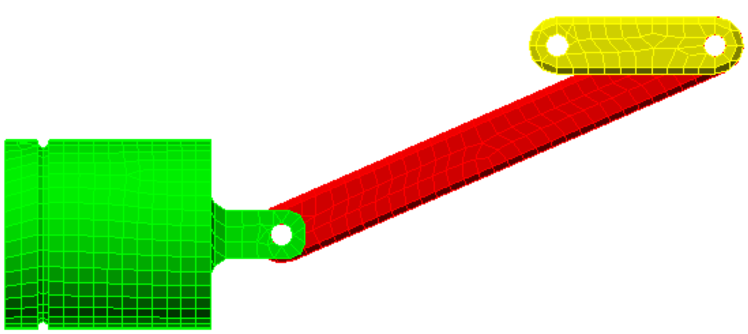
8. Click **Mesh Model** to start the modeling meshing process and close the settings dialog. When asked to review the mesh results click **No**.

9. The completed meshed model will look similar to the image shown below. At this time only the surface mesh has been created for the model. The interior mesh will be generated later during the analysis run or the check model process.



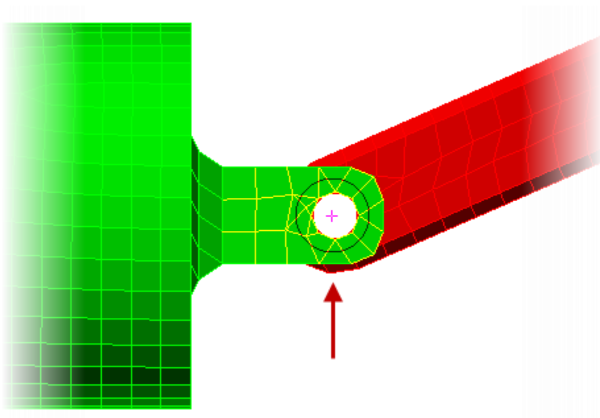
10. Use the **Orbit** viewing tool to review the mesh of the model in various locations to better understand the assembly.

11. We will now start to create joints between the components to create the mechanism. On the **View Cube** click the **Right** view to position model as shown below.

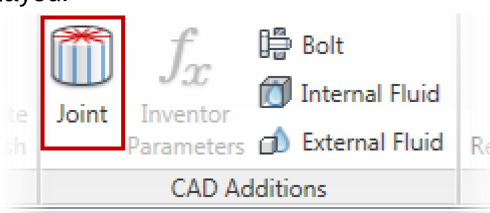


12. **Zoom** into the area where the green *Piston* part connects with the red *Arm* part. Under the **Selection** tab click **Circle** as the **Shape** and ensure **Surfaces** is selection as the **Select** type.

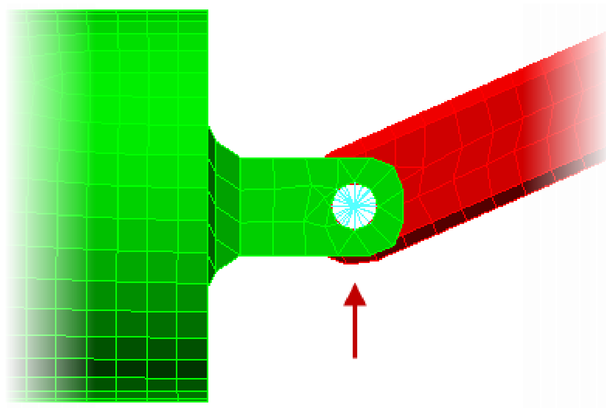
13. Draw a circle enclosing the hole where the piston and arm meet as shown in the image below. This will select the inner surfaces of the holes.



14. Click **Mesh tab | CAD Additions panel | Joint** to start the process of create a joint. The **Create Joint** dialog box is displayed.



15. The six (6) selected surfaces are shown in the **Participating Surfaces** list. Select **Pin Joint** as the **Joint Type**. Ensure that the automatic detection of the axis/center point is select which will place the joint origin in the center of the hole. Click **OK** to create the joint. The joint will be created using trusses.

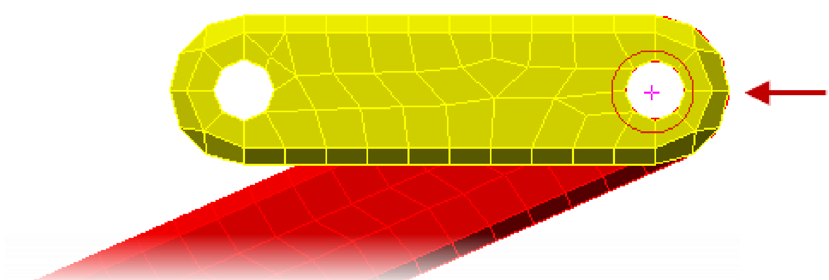


16. Press **Esc** to clear the selection set.

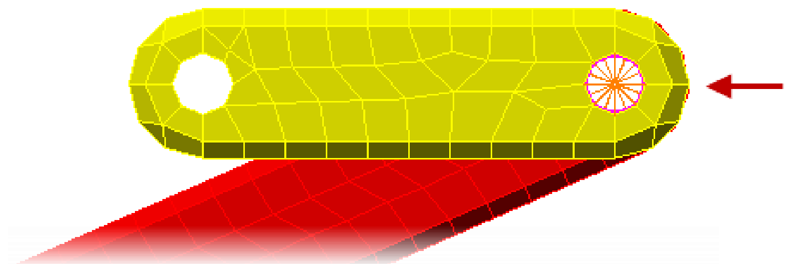
17. **Zoom** into the area where the yellow *Crank* part and red *Arm* part connect.

18. Ensure the section set is still set to **Circle** for the **Shape** and **Surfaces** as the **Select** type.

19. Draw a circle enclosing the hole where the *Arm* and *Crank* part meet as shown below.

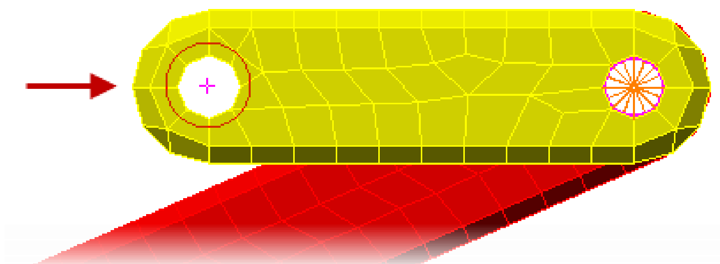


20. Click **Mesh tab | CAD Additions panel | Joint** to open the **Create Joint** dialog. Ensure that **Pin Joint** is selected and click **OK** to create the joint.



21. Press **Esc** to clear the selection set.

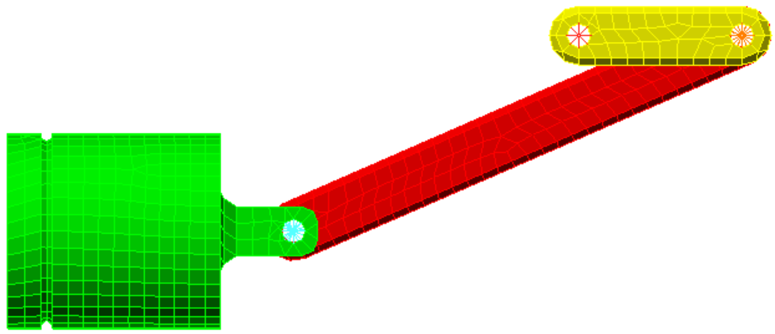
22. Draw a circle enclosing the hole at the free end of the crank as shown in the image below.



23. Click on **Joint** under the **Mesh** tab to begin creating another joint.

24. Click **OK** to create a **Pin Joint** with the two participating surfaces that are selected. This will create a joint using round beam elements.

25. Press **Esc** to clear the section set. Then click on **View tab | Navigate panel | Enclose (Fit All)** to zoom the screen to display the full model as shown below.

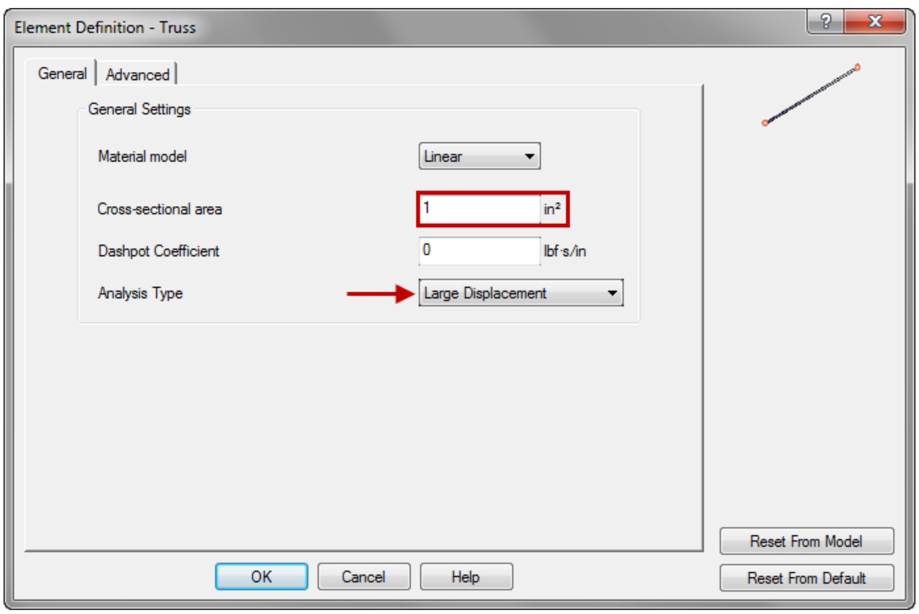


26. In the **FEA Editor Browser** select the **Element Definition** node for *Part 1* and then hold down **Ctrl** and select the **Element Definition** node for *Part 2* and *Part 3*. Then Right-Click one of the selected headings and select **Edit Element Definition**.

27. Ensure that **Large Displacement** is selected as the **Analysis Type**. Click **OK** to applied the values and continue.

28. Next we will assign element definitions for the trusses (Part 4 and 5). Select the **Element Definition** node for *Part 4* in the Browser and hold down **Ctrl** and select the **Element Definition** node for *Part 5* also. Right-Click and select **Edit Element Definition**.

29. Enter **1** in the **Cross-Sectional Area** field. This area is arbitrary and is intended only to provide a relatively rigid joint. Ensure that **Large Displacement** is selected as the **Analysis Type**. Click **OK** to exit the **Element Definition** dialog.

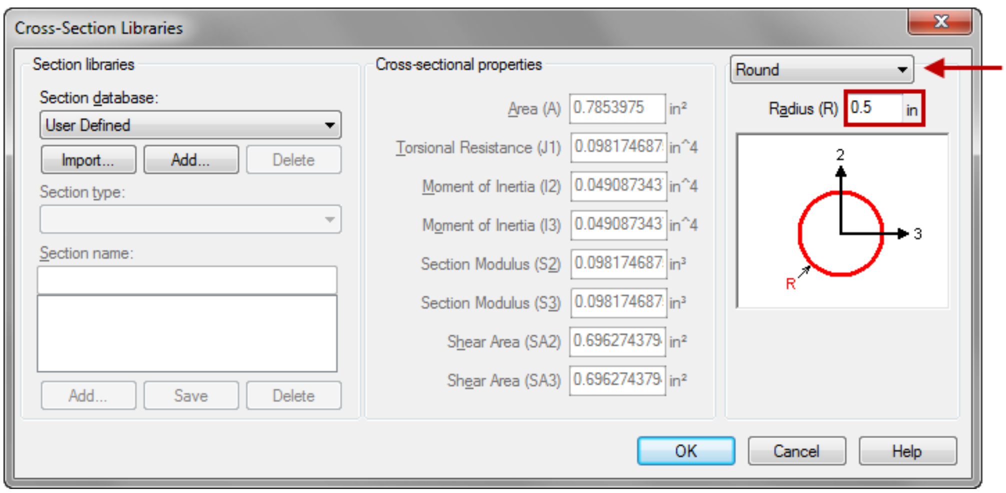


30. Now we will assign the element definition for the beam Part 6. Right-Click the **Element Type** node for *Part 6* in the Browser and select **Beam** as the element type. Later in the project we are going to prescribed rotation on the center of this joint so we need an element type that has rotational degrees of freedom (DOF). Trusses do not have rotational DOF.

31. Right-Click the **Element Definition** node for *Part 6* and select **Edit Element Definition**. This will allow us to set the properties of the element.

32. Select the first row of the **Sectional Properties** table then click the **Cross-Section Libraries** button.

33. Select **Round** option in the drop-down at the upper right corner. Then enter **0.5** in the **Radius** field. The cross section is arbitrary and is intended only to provide a relatively rigid joint. Click **OK** to accept the new properties and exit.



34. Ensure **Large Displacement** is selected as the **Analysis Type**

35. Click the **Advanced** tab. Select the **Large Rigid Body Rotation** checkbox. This implements an alternate beam element formulation and is necessary to ensure accurate stress results when the beams will experience rigid body rotation. Click **OK** to complete the element definition settings.

36. Now we will assign a material to each of the parts in the simulation. Select the **Material** node for *Part 1* in the **Browser** and then hold down **Ctrl** and select the **Material** node for *Part 2* and *Part 3* also. Right-Click one of the selected nodes and select **Edit Material**.

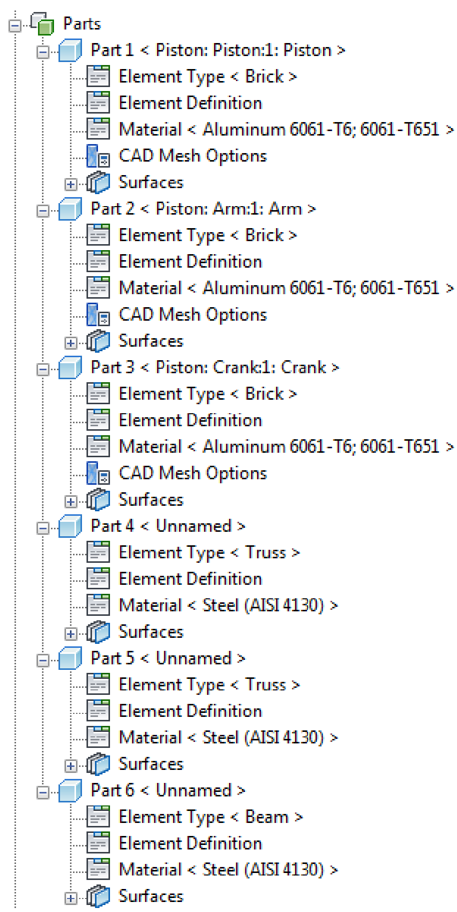
37. Select **Aluminum 6061-T6; 6061-T651** as the material and click **OK**.

38. Select the **Material** node for *Part 4* and then hold down **Ctrl** and select the **Material** node for *Part 5*. Right-Click and select **Edit Material**.

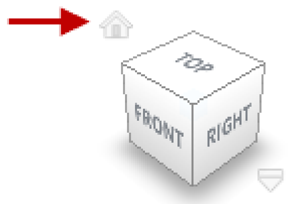
39. Select **Steel (AISI 4130)** as the material and click **OK**.

40. Select the **Material** node for *Part 6* and right-click and select **Edit Material**. Select **Steel (AISI 4130)** as the type and click **OK**. You can only specify the material for parts with the same element type at once.

41. The **FEA Editor Browser** will look like the image below at this point.

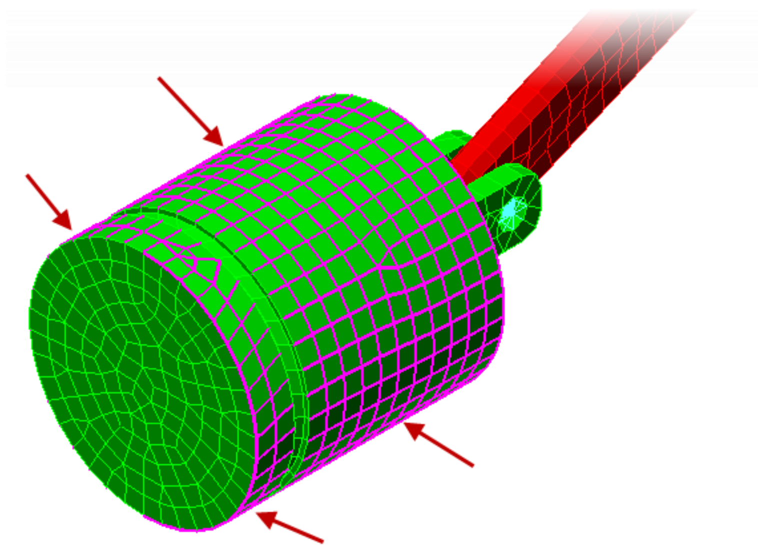


42. Click the **Home** view from the **View Cube**. This will position the model in an isometric view.



43. Change the selection controls to **Point** for the **Shape** and **Surfaces** for the **Select** option.

44. Select one of the cylindrical surfaces on the outside of the piston as shown below. Then hold down **Ctrl** and select the other three surfaces as shown below.



45. Rich-Click and in the graphics window and select **Add | Surface Boundary Conditions** from the menu.

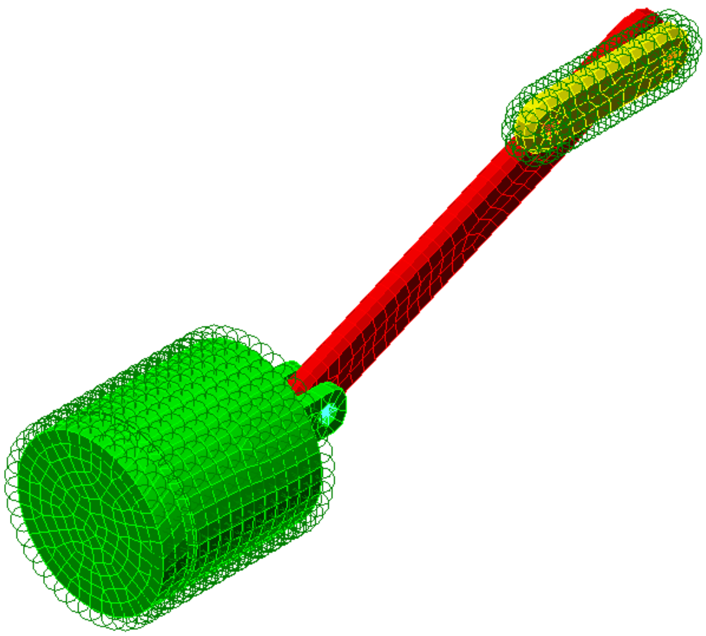
46. Select the **Tx** and **Tz** checkboxes for the **Constrained DOFs**. This will only allow the *Piston* part to move in the Y Axis direction. Click **OK** to complete the creation of the General Constraint.

47. Right-Click the *Part 3* heading in the Browser tree and select **Select Subentities | Surfaces**. All of the surfaces for *Part 3* are now selected.

48. Right-Click in the graphics window area and select **Add | Surface Boundary Conditions** to start the creation of a new General Constraint.

49. Check the **Tx** checkbox for **Constrained DOFs** to stop the part from moving in the X Axis direction. Click **OK** to create the constraint. This constraint will stabilize the crank by preventing any twisting or bending due to flexure of the joints and will make convergence of the solution easier.

50. The model will look like the image shown below at this time.

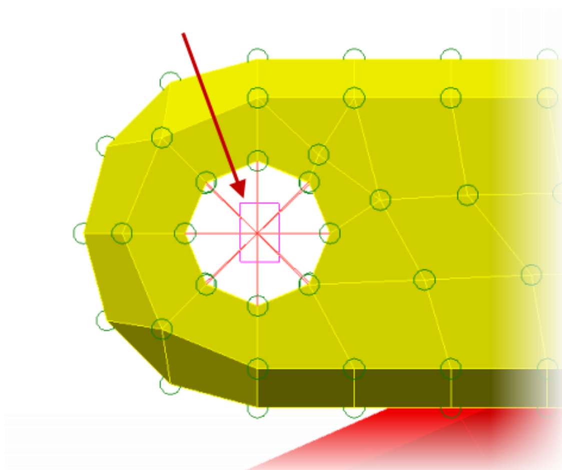


51. Click the **Right View** from the **View Cube**.

52. **Zoom** into the yellow *Crank* part area.

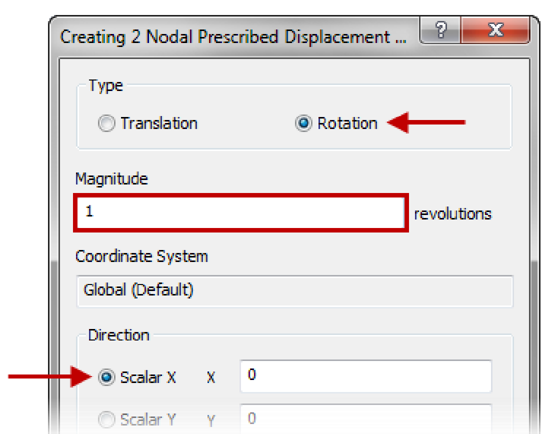
53. Change the section preferences to **Rectangle** for **Shape** and **Vertices** for **Select**. This will allow you to drag a rectangle shape to select nodes only.

54. Draw a box enclosing the center of the joint on the left side of the *Crank* part as shown below.



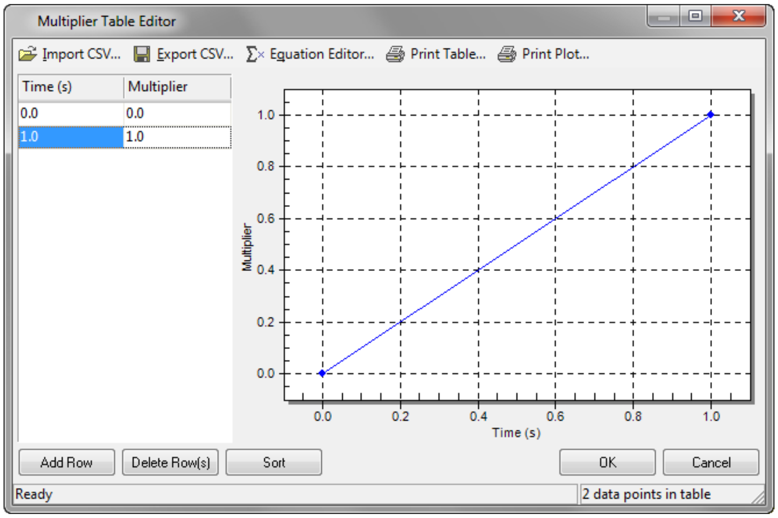
55. Right-Click in the graphics window and select **Add | Nodal Prescribed Displacement**.

56. Select **Rotation** as the type and enter **1** as the **Magnitude** for revolutions. Ensure the **Scalar X** is selected as the **Direction**.

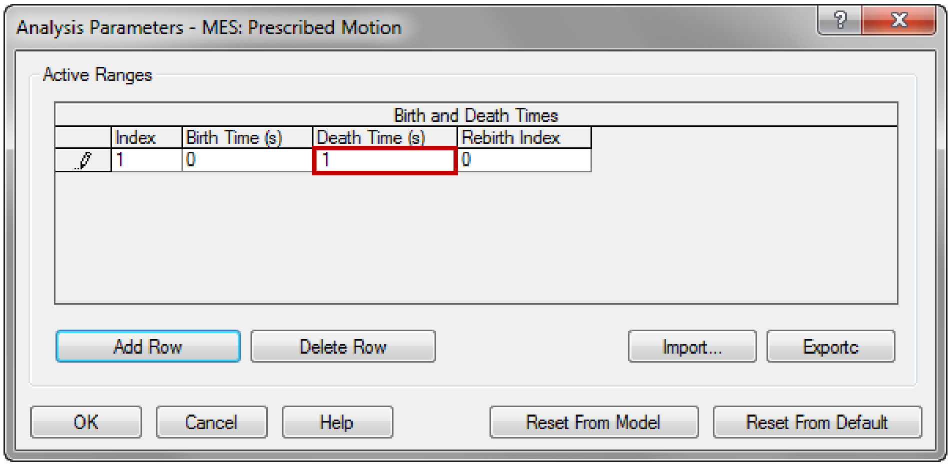


57. Click the **Curve** button to setup a load curve.

58. The default curve ramps up to the multiplier of 1 over a one second period of time. This is the load we want to keep so click **OK** to continue.



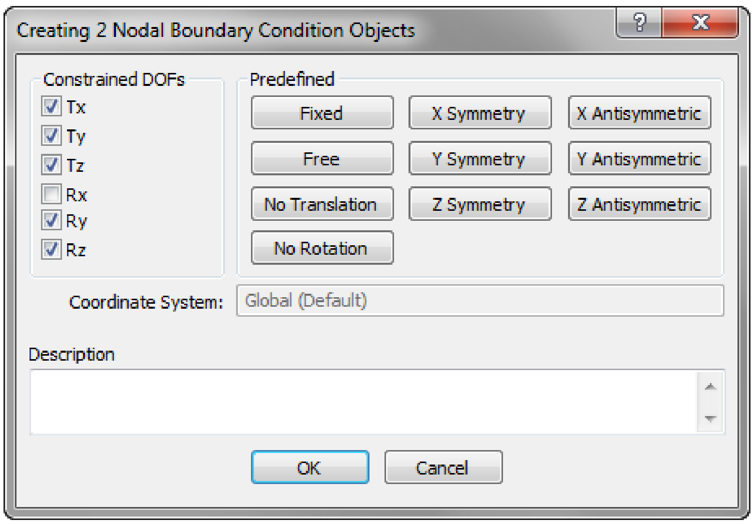
59. Click **Data** to setup the **Active Range** of the displacement. Enter **1** as the **Death Time** and click **OK**. Prescribed displacements can be deactivated during a simulation event to allow free motion when necessary. Here, a death time of 1 second ensures that the displacement is active throughout the 1 second simulation event. In addition, deactivated prescribed displacements can be reactivated at a later time via the Rebirth Index and subsequent Birth Time entries. Simply add a row to the table for each rebirth of a prescribed displacement.



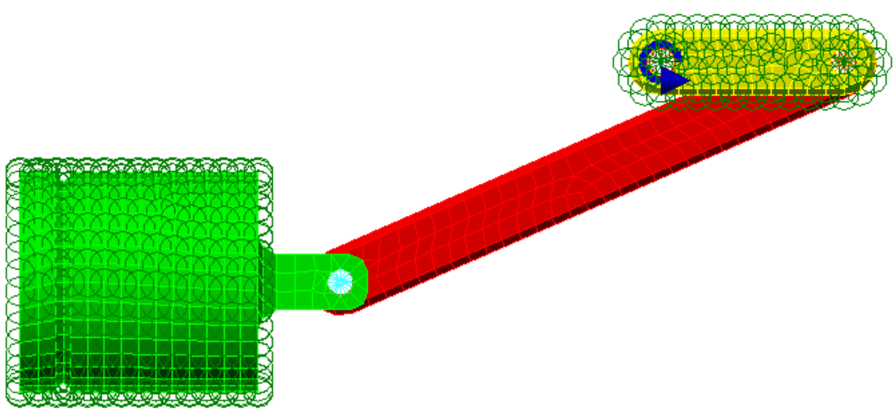
60. Click **OK** to complete the prescribed displacement setup.

61. With the center nodes of the joint still select Right-Click in the graphics window and select **Add | Nodal Boundary Conditions.**

62. Click the **Fixed** button and then clear the checkbox for the **Rx Constrained DOFs**. This will lock the position of the joint and only allow it to rotate about the X Axis. Click **OK** to complete.



63. **Zoom** out on the full model as shown in the image below.



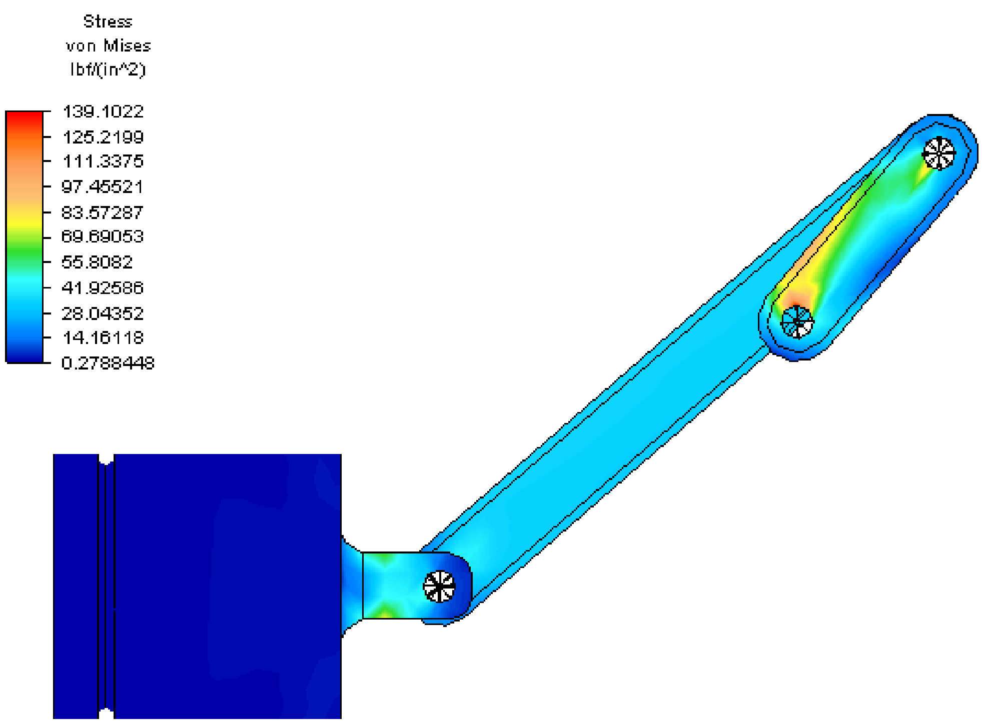
64. Click **Setup tab | Model Setup panel | Parameters**.

65. Enter **36** as the **Capture Rate** and ensure the **Duration** is set **1** second. This will set the output to be in 10 degree increments (360/36). Click **OK** to continue.

66. Click **Analysis tab | Analysis panel | Run Simulation**. This will start the simulation process by first processing the complete mesh of the model then solving the simulation. The simulation will take approximately 15 minutes to complete depending on your computer resources.

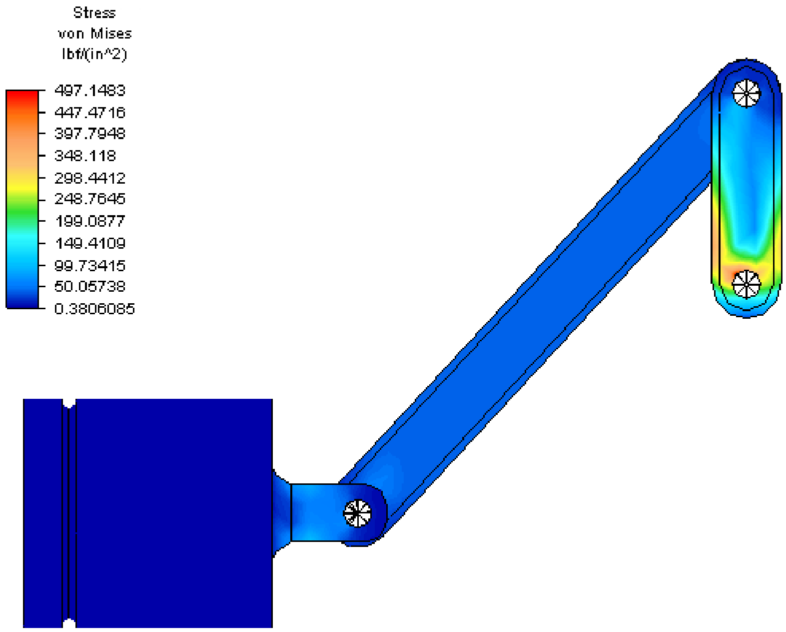
67. To turn off the visibility of the loads and constraints symbols click on **Results Options tab | View panel | Loads and Constraints**.

68. At time step 5 the Von Mises Stress results will look similar to the image below.



69. Click the **Results Contours tab | Load Case Options panel | Load Case – Set**. Enter **9** into the load case field and click **OK**.

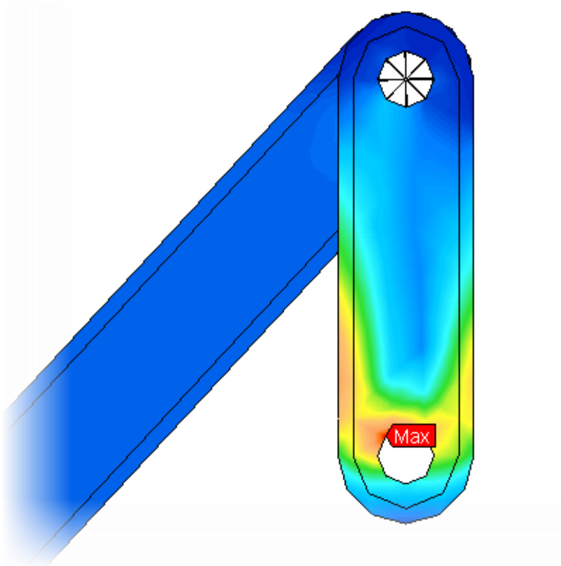
70. The results at the 0.25 seconds (Time Step 9) are displayed with a maximum Von Mises Stress of about 500 psi as shown below.



71. Click **Results Inquire tab | Probes panel | Maximum** to display a probe where the maximum stress is located.

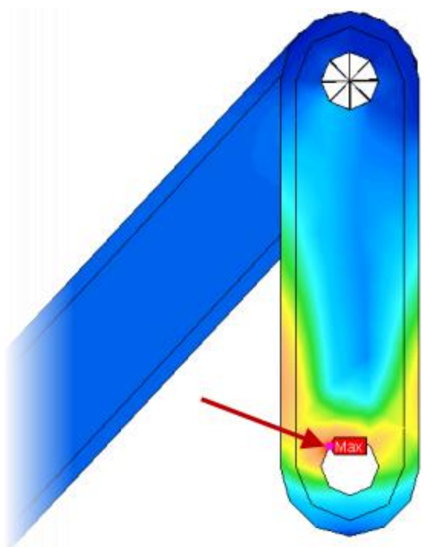
72. Right-Click on *Part 6* in the **Results Browser** and turn off the **Visibility**. This will allow us to review the results for only the *Crank* part.

73. **Zoom** into the *Crank* part as shown below.

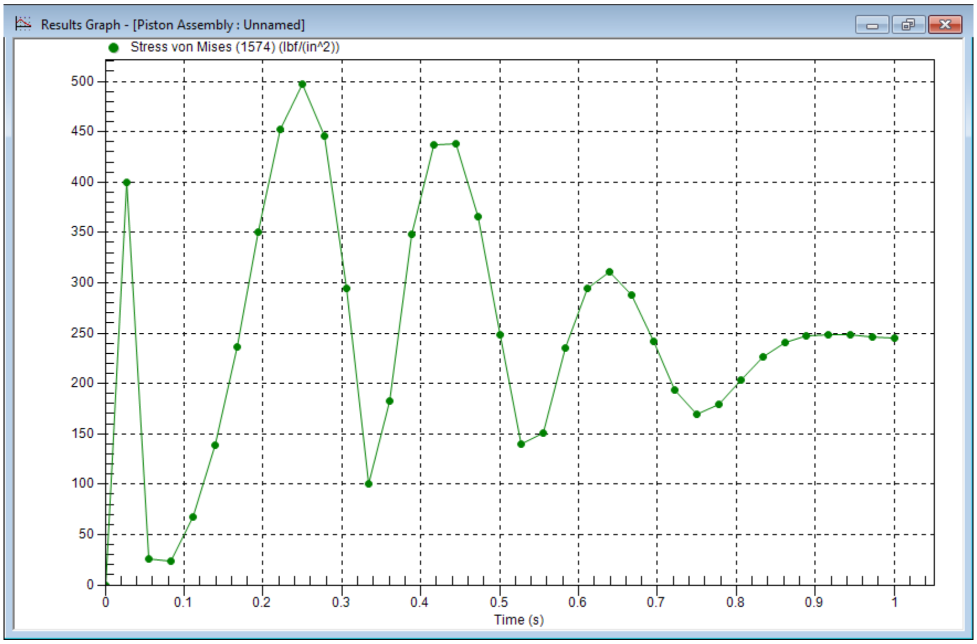


74. Set the selection type to **Point** for **Shape** and **Nodes** for **Select**. This will allow for the selection of a single node.

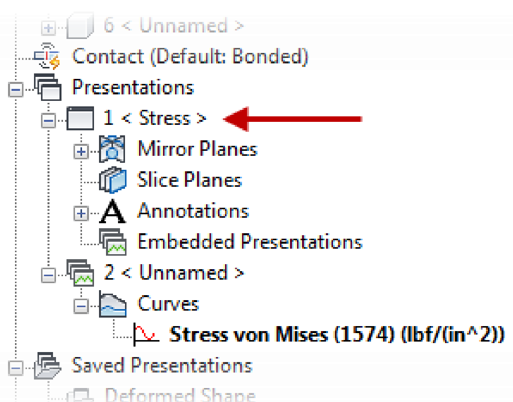
75. Pick to select the node that has the highest stress as marked by the probe as shown below.



76. Right-Click in the graphics area and select **Graph Value(s)**. The stress versus time for the selected node will be displayed as shown below.

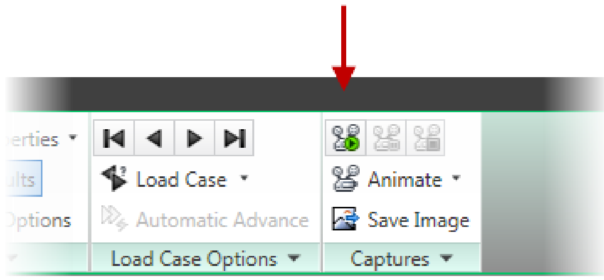


77. Click the **1 <Stress>** heading under **Presentations** in the **Results Browser** to return to the previous results plot.



78. Deactivate the **Maximum Probe** and **Zoom** out to view the complete model with some extra empty space around the model.

79. Click on **Results Contours tab | Captures panel | Start**. This will play an animation of the complete simulation time cycle. Click the **Stop** button when complete.



80. You can also save the results as an AVI movie by click **Results Contours tab | Captures panel | Animate – Save As AVI**. Specify a file name and location to save the AVI file and click Save to capture the movie.

81. **Save** the simulation so you can review the results later if required.