ME 24-688 – Week 12

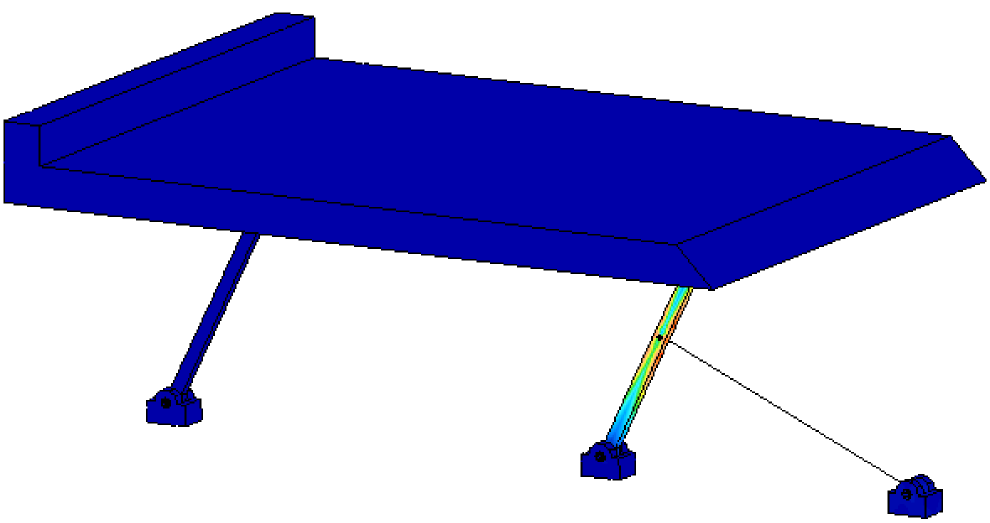
Lift Assembly Simulation

Mechanical Event Simulation

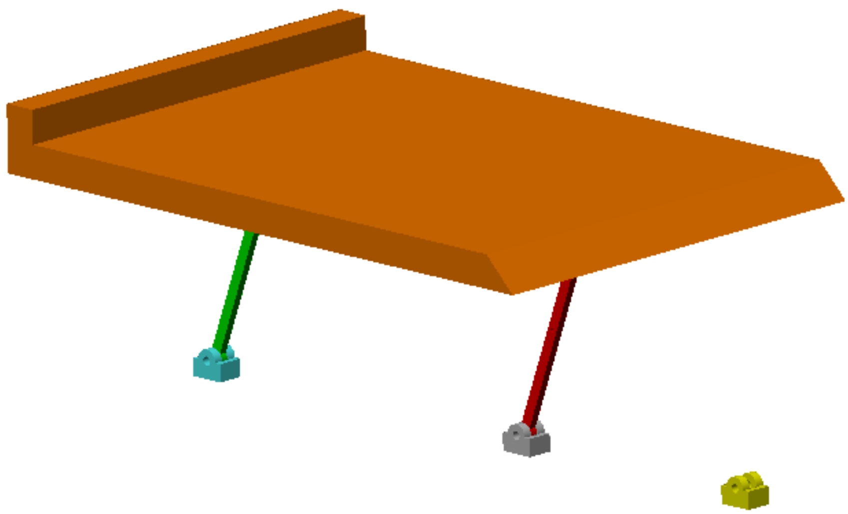
During this project assignment you will continue to explorer and learn about the Mechanical Event Simulation (MES) environment available within Autodesk Simulation. The Actuator element type with prescribed displacement will be used and you will also create several joints and constraints on components.

1.1 Project 3 – Lift Assembly Simulation

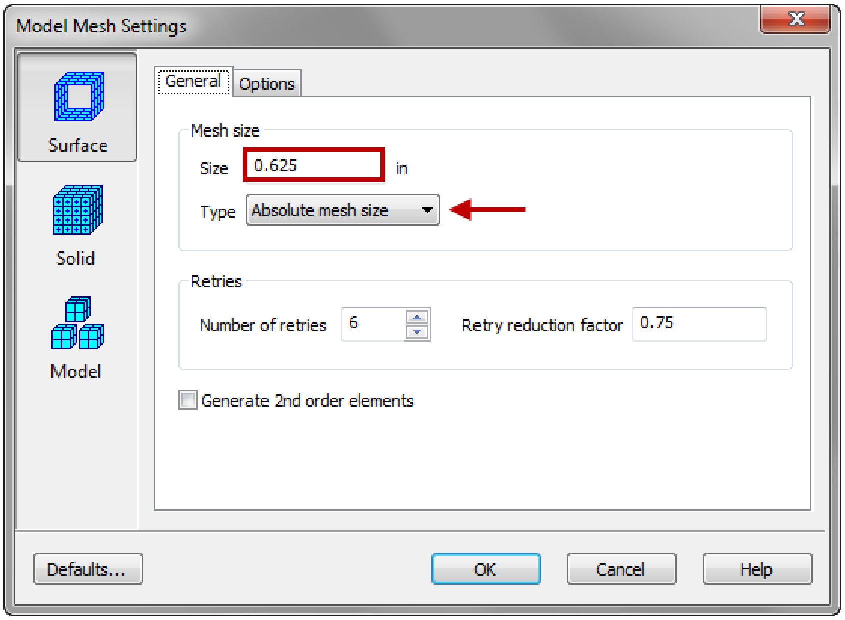
The objective of this project is to analysis the stress distribution and motion of the lift assembly mechanism shown in the following image. An actuator connects to the support bar on the lift with a stroke of five inches to raise and lower the main top plate part.



1. Open the *Lift Assembly.Step* file in Autodesk Simulation. Accept the use the default STEP file units of **Inches** and specify **MES with Nonlinear Material Models** as the analysis type. The starting model will look like the image below.



2. Click **Mesh tab | Mesh panel | 3D Mesh Settings**. Within the **Model Mesh Settings** dialog click the **Options** button. Select **Absolute Mesh Size** as the type and enter **0.625** as the **Mesh Size**. Click **OK** to accept the mesh options.

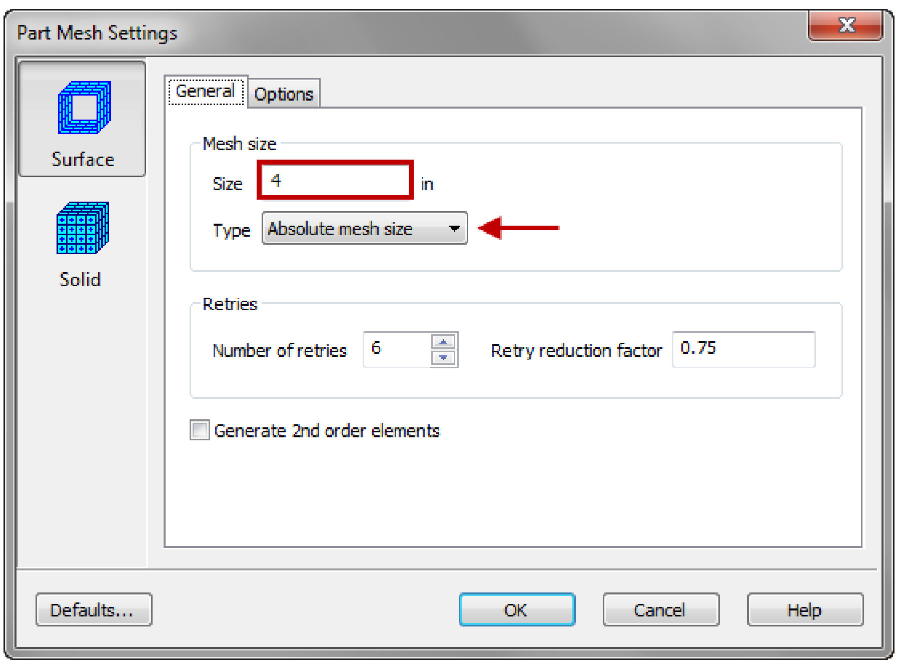


3. Click **OK** to exit the **Model Mesh Settings** dialog.

4. To set a different mesh size on the *Base* part Right-Click on the **CAD Mesh Options** node under *Part 5* in the **FEA Editor Browser** and select **Part**.

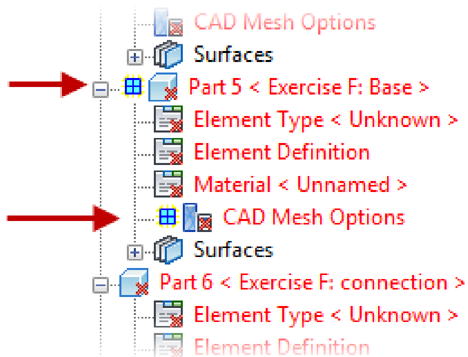
5. Within the **Part Mesh Settings** dialog click the **Options** button.

6. This part is much larger than the other parts so we want to use a larger mesh to improve performance. Select **Absolute Mesh Size** as the type and enter **4** as the **Mesh Size**. Click **OK** to accept the mesh options.

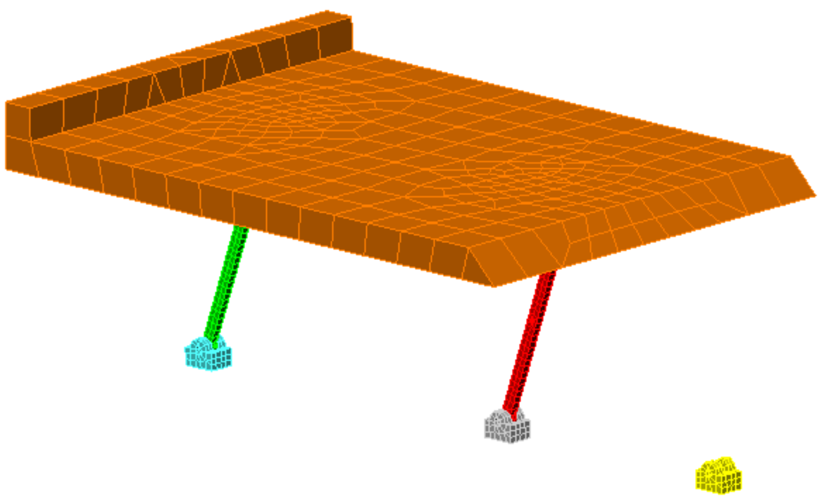


7. Click **OK** to exit the **Part Mesh Settings** dialog.

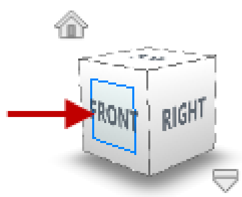
8. Notice in the **FEA Editor Browser** now has a new blue box icon next to *Part 5*. This indicates the mesh is using a part specific override setting.



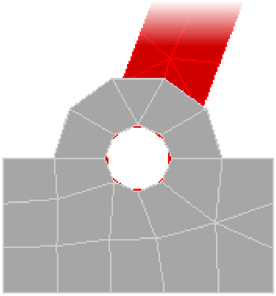
9. Now we will create the initial mesh. Click the **Mesh tab | Mesh panel | Generate 3D Mesh** and click **No** when asked to view mesh results. The completed model will look similar to the image shown below.



10. Click on **Front** view from the **View Cube**.

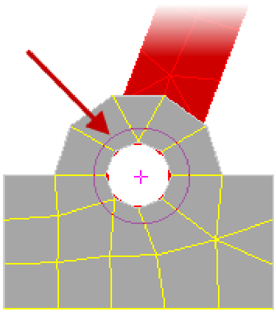


11. **Zoom** into the lower mounting clevis as shown below.



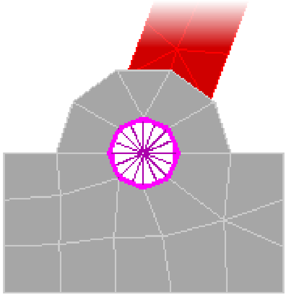
12. Set the section type to **Circle** for the **Shape** and **Surfaces** for the **Select**.

13. Draw a circle around the hole of the mounting clevis as shown below to select the inner hole surfaces of the parts.

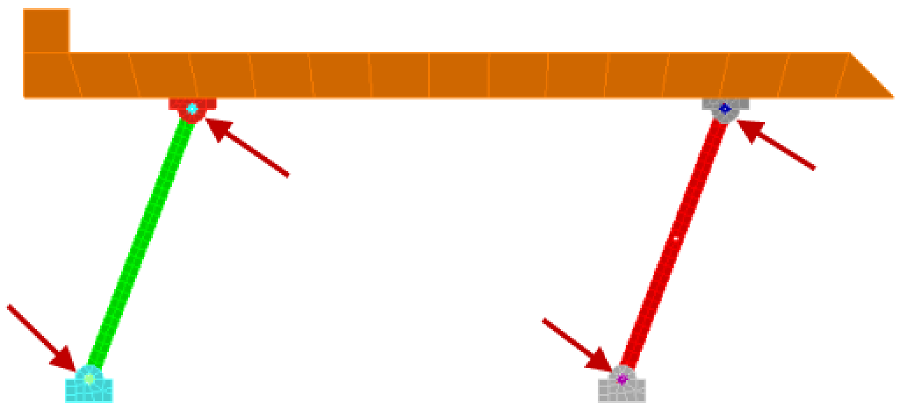


14. Click **Mesh tab | CAD Additions panel | Joint** to open the **Create Joint** dialog. Verify the joint type is **Pin Joint** and click **OK** to create the joint.

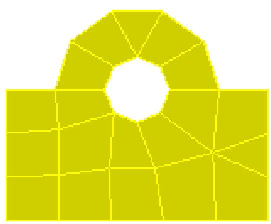
15. A new truss element part is created as shown below.



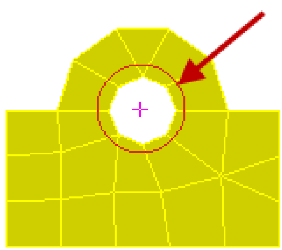
16. Repeat steps 11-15 for the other three support link clevis joints as marked below.



17. **Zoom** into the mounting clevis that is not connected where the actuator will connect as shown below.

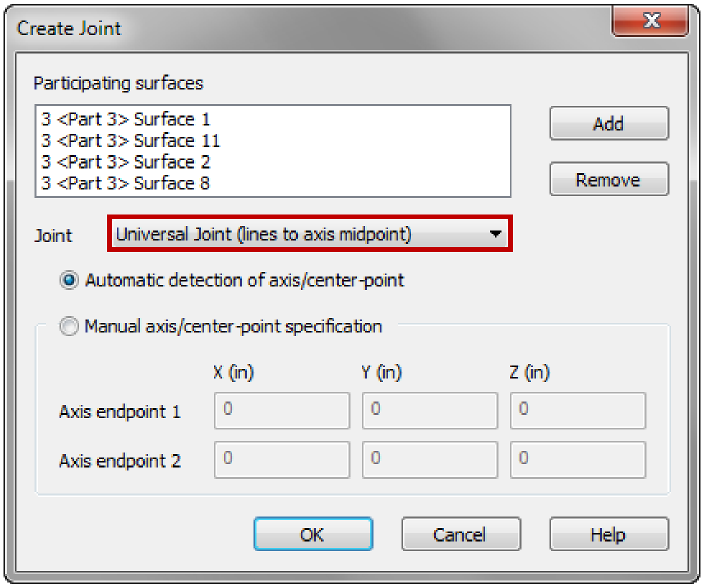


18. Draw a circle around the hole of the clevis as shown below to select the inner hole surfaces of the part.

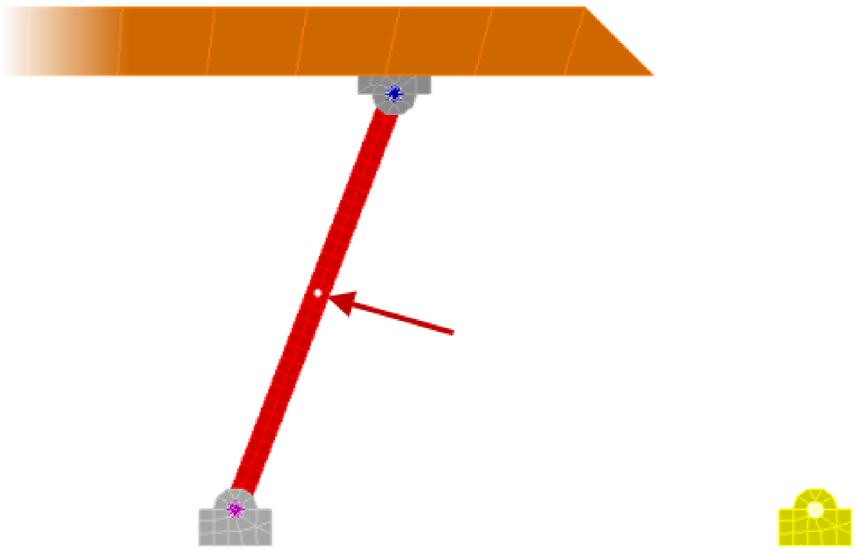


19. Click **Mesh tab | CAD Additions panel | Joint** to open the **Create Joint** dialog.

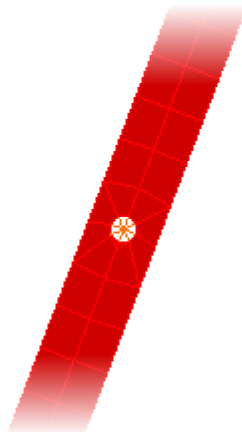
20. Select **Universal Joint** as the **Joint Type** as shown below. This will create a joint where the nodes at either end of the model selected surfaces will be connected to the midpoint of the axis. This type of joint will allow the model to rotate about the axis as well as swivel about the center point of the axis. Click **OK** to create the joint.



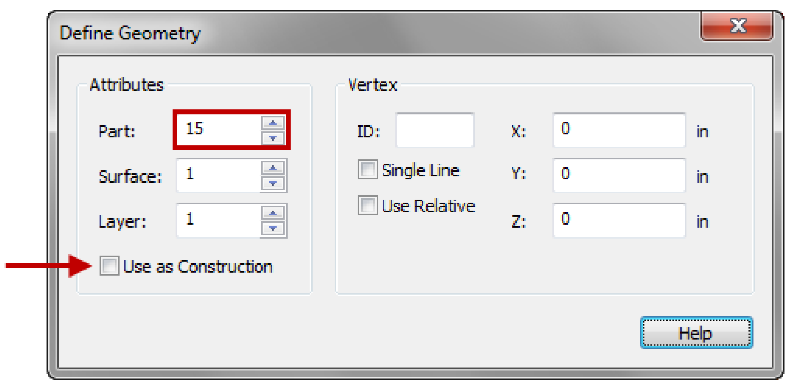
21. Repeat step 18-20 for the hole that is in the middle of the *Rod* part shown below where the other end of the actuator will connect.



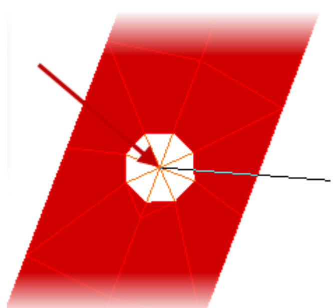
22. **Zoom** into the middle section of the right support link as shown below.



23. To create the actuator that will control the height of the lift assembly click **Draw tab | Draw panel | Line**. Enter **15** in the **Part** field and ensure the **Use as Construction** checkbox is not selected.

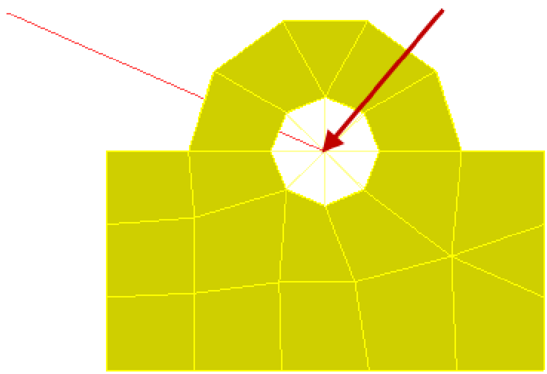


24. Click the vertex at the center of the joint in the hole in the middle of the support as shown below.



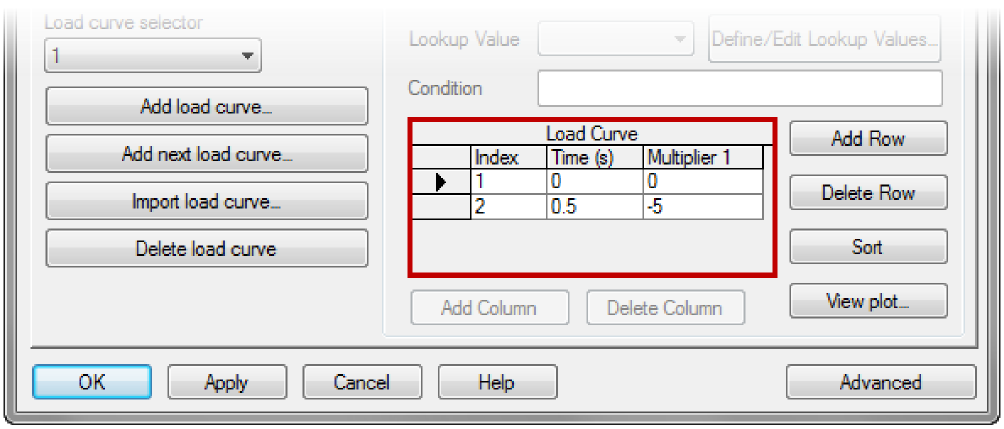
25. **Zoom** out and then **Zoom** into the mounting clevis part where the other end of the actuator will connect. Press **Esc** one to exit the zoom mode.

26. Click the vertex at the center of the joint in the clevis hole as shown below. Press **Esc** to exit the line selection mode.



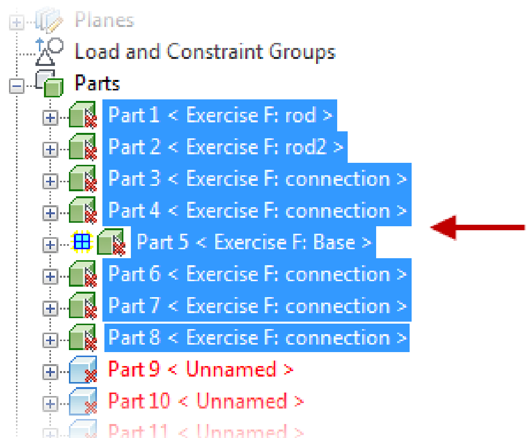
27. Press **Esc** key twice to exit the line creation command.

28. Click **Setup tab | Model Setup panel | Parameters** to open the **Analysis Parameters** dialog. Enter **0.5** for the **Duration** and **50** for the **Capture Rate**. Then enter **0.5** into the second row of the Load Curve for the **Time** field and enter **-5** into the **Multiplier** field. This will create the parameters for Load Curve 1 that we will utilize later to drive the actuator. Click **OK** to complete.



29. While holding down the **<Ctrl>** and **<Shift>** keys, press **M** key on the keyboard. This is a hot key to collapse the parts list in the Browser tree view.

30. Click on the heading for *Part 1* in the tree view. Holding down the **<Shift>** key, click on the heading for *Part 8* in the tree view.



31. Right-Click on one of the selected headings. Select the **Edit | Element Data** option to open the **Element Definition** dialog. Ensure **Large Displacement** is set as the **Analysis Type** and click **OK**.

32. With *Parts 1-8* still selected Right-Click one of the headings and select **Edit | Material** to open the **Element Material Selection** dialog. Select **Steel (ASTM – A36)** from the available list of materials. Click **OK** to complete the material assignment.

33. Click on the heading for *Part 9* in the tree view.

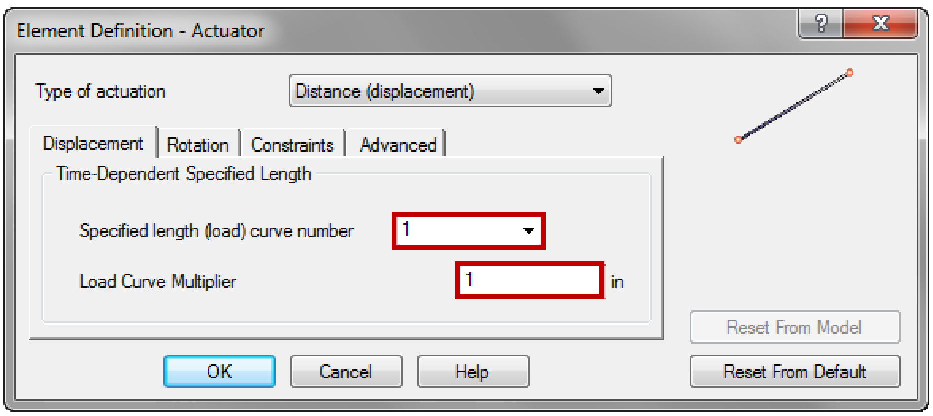
34. Holding down the **<Shift>** key click on the heading for *Part 14* in the tree view.

35. Right-Click on one of the selected headings and select **Edit | Element Data** to open the **Element Definition** dialog. Enter **1** for the **Cross Sectional Area** and ensure **Large Displacement** is set as the **Analysis Type**. Click **OK** to exit.

36. With *Parts 9-14* still selected Right-Click one of the headings and select **Edit | Material** to open the **Element Material Selection** dialog. Select **Steel (ASTM – A36)** from the available list of materials. Click **OK** to complete the material assignment.

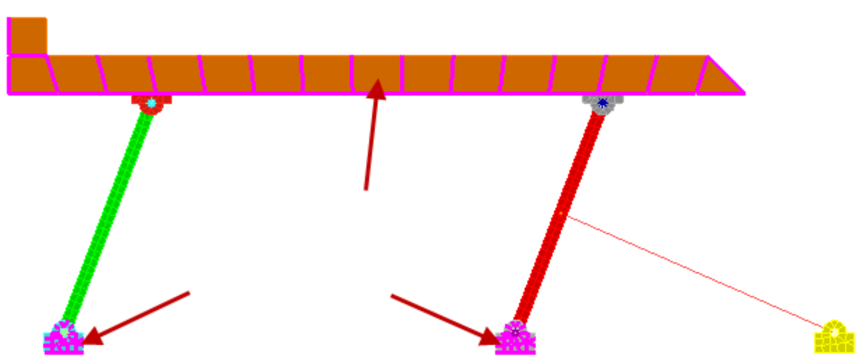
37. Select *Part 15* in the tree and Right-Click to select **Edit | Element Type | Actuator**. This will make the line that was created into the actuator.

38. To verify the settings required for the actuator Right-Click on *Part 15* and select **Edit | Element Data** to open the **Element Definition** dialog. Verify that Load Curve 1 is used for the Load Curve and that the Load Curve Multiplier is set as 1. Click **OK** to complete.



39. Set the selection settings to **Point** for **Shape** and **Surfaces** for **Select**.

40. Click on the surface on the side of the large top part. Holding down the **<Ctrl>** key click on the side surface of both the bottom mounting clevis parts as shown below.

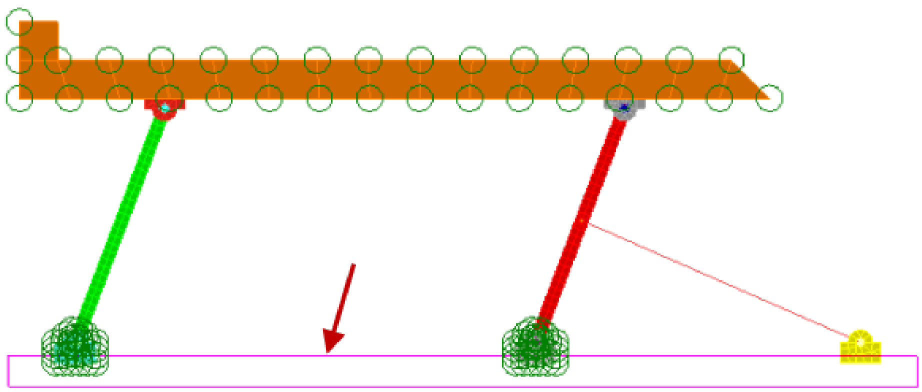


41. To control the position and movement of these parts click on **Setup tab | Constraints panel | General Constraint** to open the **Bounding Condition** dialog.

42. Check the **Ty** checkbox for **Constrained DOFs** to lock the translational movement in the Y Axis of the parts.

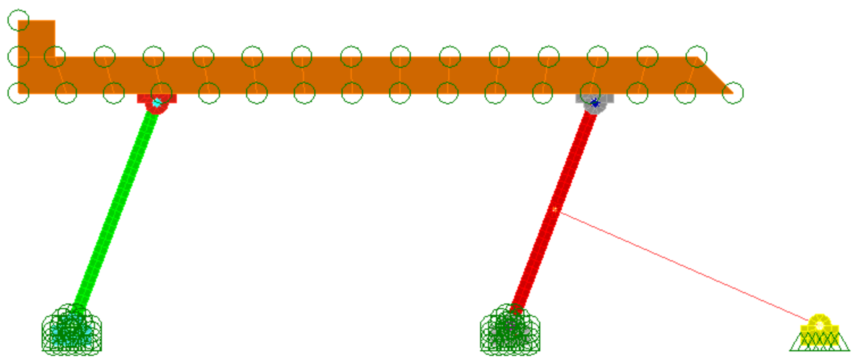
43. Set the selection settings to **Rectangle** for **Shape** and **Surfaces** for **Select**.

44. Draw a box enclosing the bottom edges of all three bottom clevis parts as shown below.



45. To lock the position of the clevis parts click on **Setup tab | Constraints panel | General Constraint** to open the **Bounding Condition** dialog. Click the **Fixed** button to lock all degree of freedom of the surfaces. Click **OK** to complete the assignment.

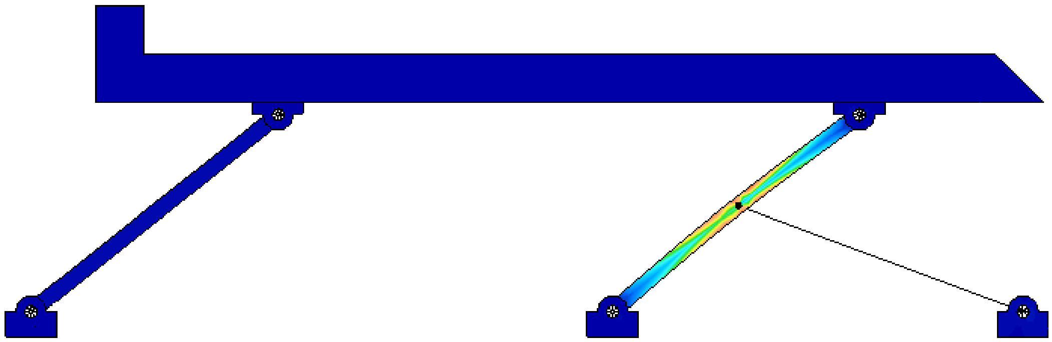
46. The model will look like the image below at this time of the project.



47. The model is ready to be solved. Click **Analysis tab | Analysis panel | Run Simulation** to start analyzing the model. The simulation will take approximately 8 minutes to complete depending on your computer resources. Once completed the results are loaded into the Results environment.

48. Click **OK** to close the pop-up message at the completion of the analysis. This is due to having an actuator element in the model.

49. Turn off the display of the loads and constraints by deselect **Results Options tab | View panel | Loads and Constraints**.

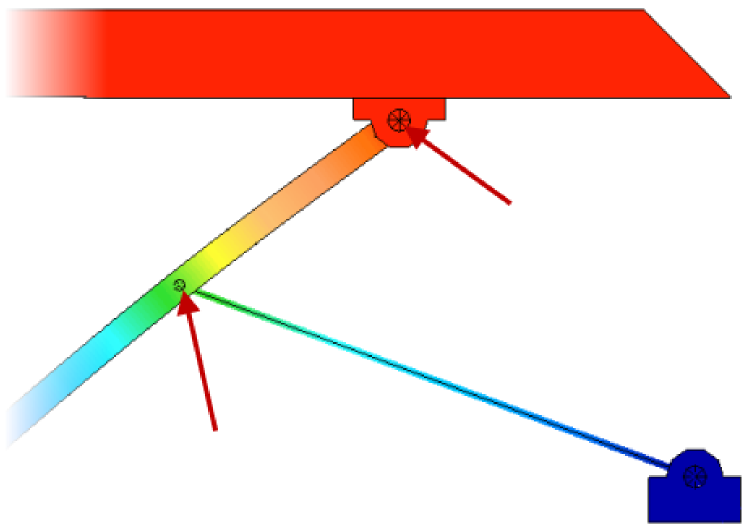


50. During the simulation over the half second time the actuator gets five (5) inches smaller pulling the lift into a lower position. Click **Results Options tab | Captures panel | Start** to play the simulation. You will notice an oscillatory bending of the support link to which the actuator is attached. This top part moves forward in surges because the motion at the center hole is steady but the motion of the top part lags because of the time needed to accelerate the heavier mass part. Once accelerated the top part inertia causes it to overshoot the actuator position, reversing the bend in the support link and producing the reaction forces that decelerate the top part. Click **Stop** when done viewing the animation of the results.

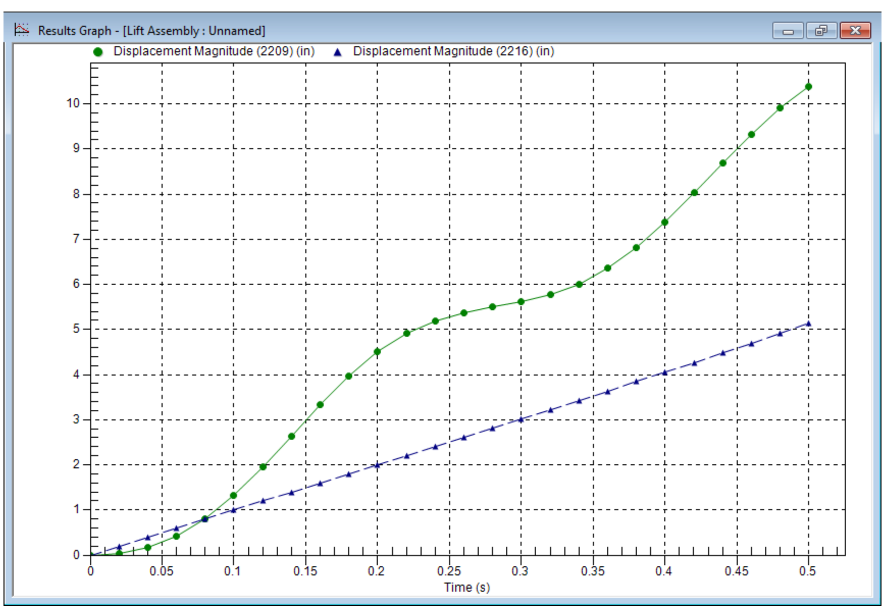
51. Switch the results view to displacement by clicking **Results Contours tab | Displacement panel | Displacement**.

52. Set the section values to **Point** for **Shape** and **Nodes** for **Select**.

53. **Zoom** in as required and select the center node of the of the middle support hole where the actuator mounts. Then hold down **<Ctrl>** and select the center node of the top mounting clevis hole as shown below.



54. Right-Click in the graphics window and select **Graph Value(s)**. This will open a graph showing the displacement of the two nodes over the time of the simulation. You will notice the node where the actuator mounts maintains a nice linear movement. Where the displacement for the node on the top is pulsating.



55. **Save** the simulation so you can later review the results as required.