AUTODESK **INVENTOR**

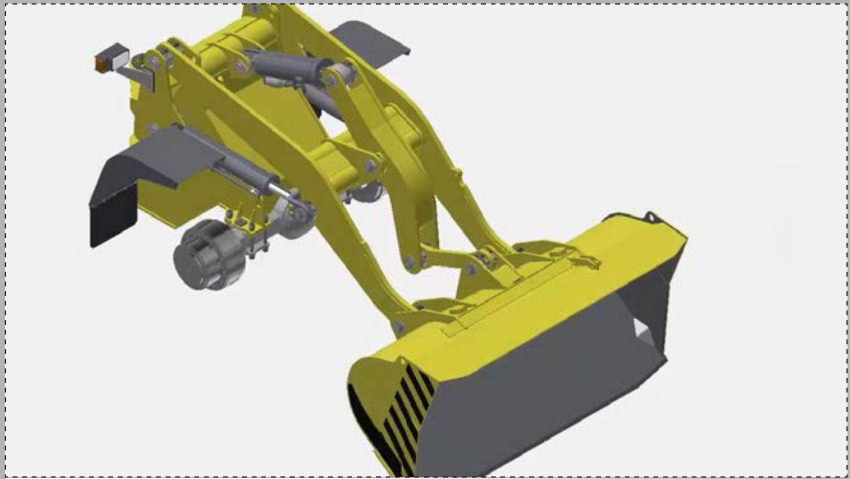
Trial Projects

Dynamic Simulation

Analyze front loader assembly

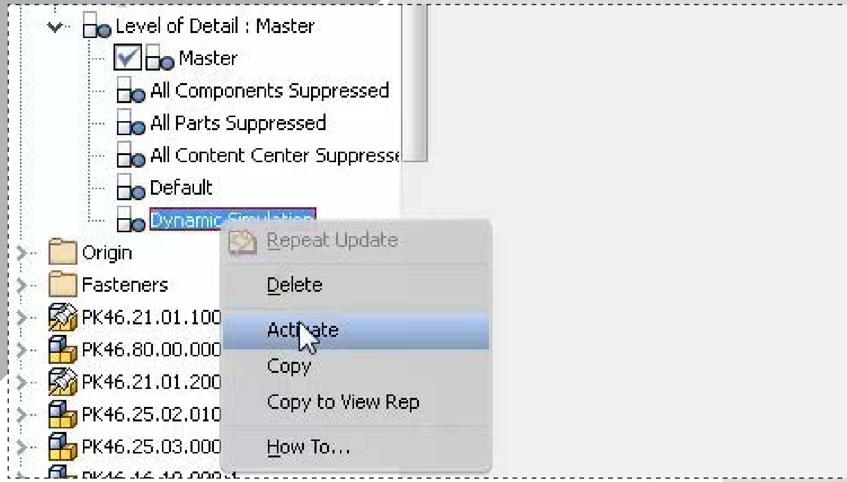
PART 1: DYNAMIC SIMULATION

1.



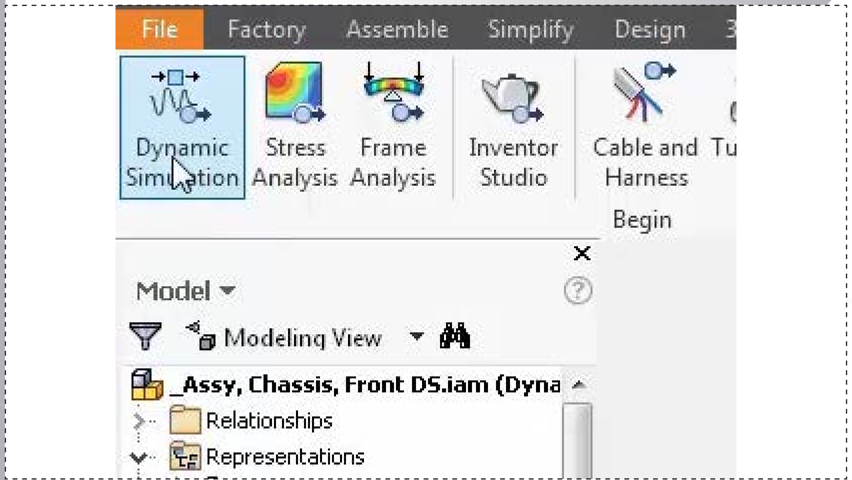
In Inventor, click the ‘Projects’ icon in the ribbon. Navigate to where you saved the project files and select ***Assy, Chassis, Front DS.ipj***. Then open ***Assy, Chassis, Front DS.iam***.

2.



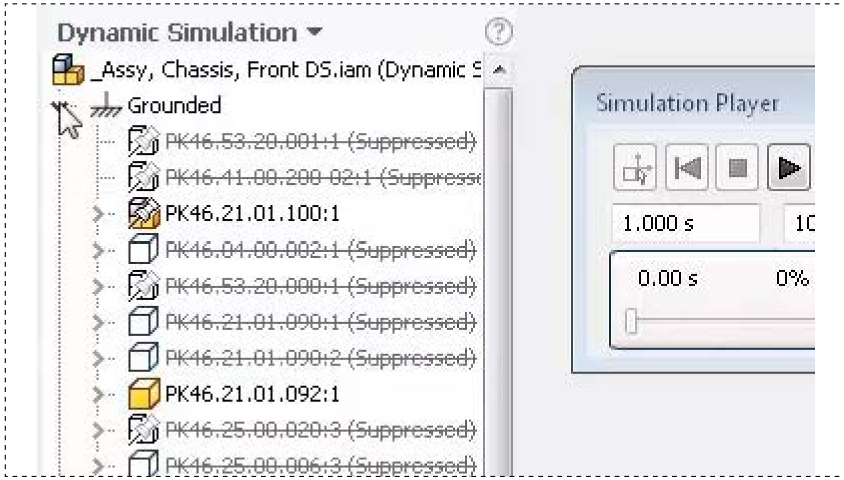
Right click the ‘Level of Detail Dynamic Simulation’ in the browser and select ‘Activate’.

3.



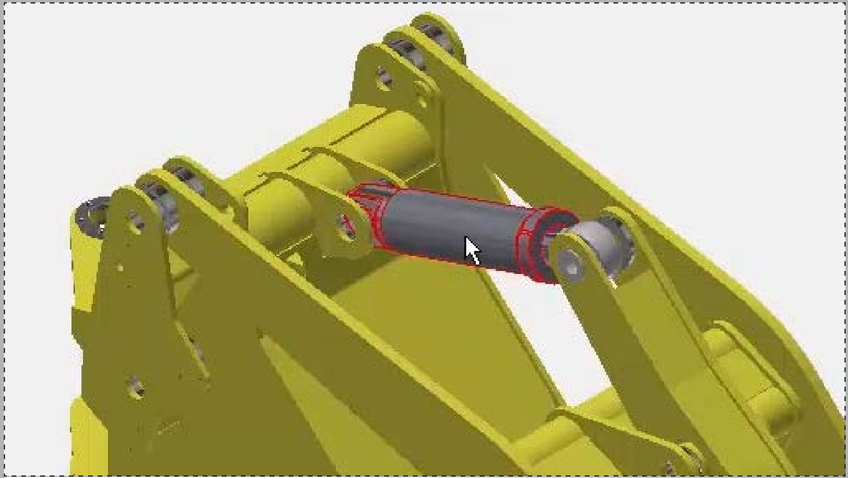
Select ‘Concentric’ from the drop-down for the placement.

4.



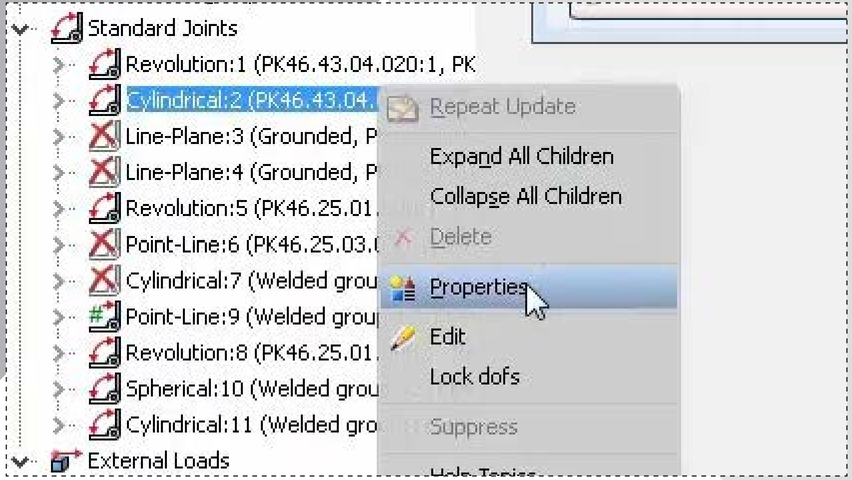
In the browser, collape the ‘Grounded’ item.

5.



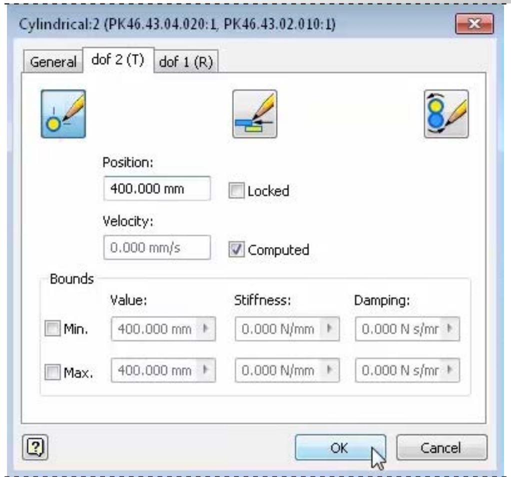
In the Graphics Window, select the hydraulic cylinder assembly, ‘PK46.43.04.0202:1 (~1)’.

6.



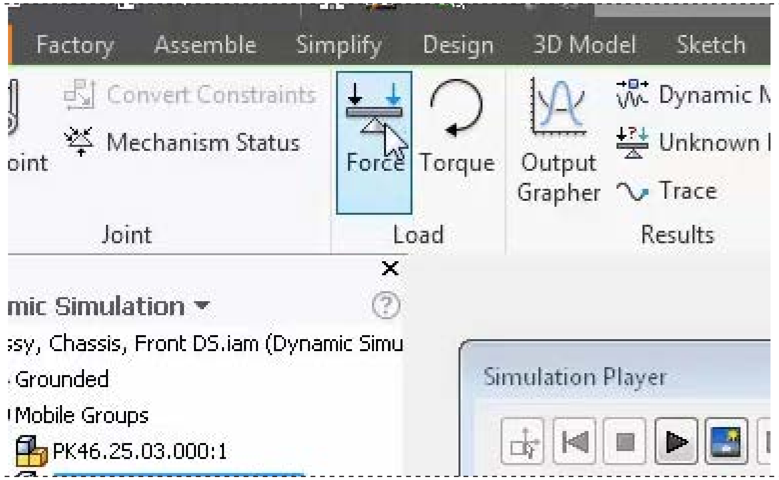
In the Browser, under ‘Standard Joints’, right click ‘Cylindrical:2’ and select ‘Properties’ from the short cut menu.

7.



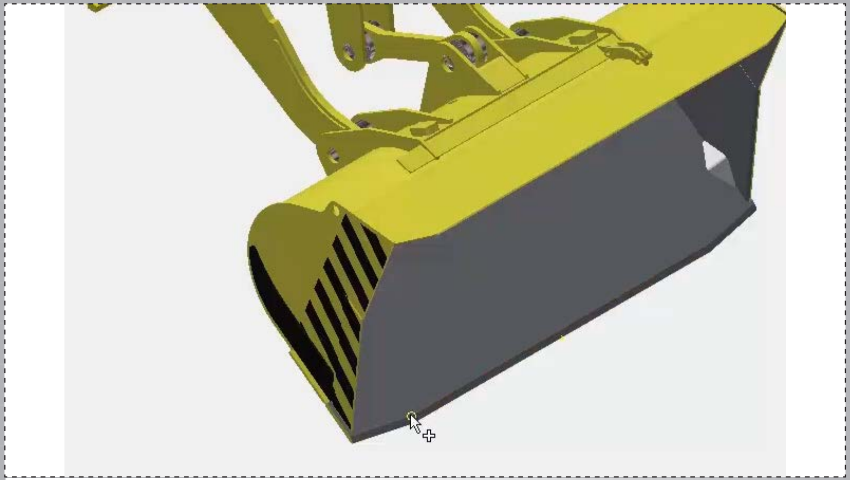
In the ‘Joint Properties’ dialog box, on the dof 2 (T) tab, enter initial condition value of 400 mm. Then click ‘OK’.

8.



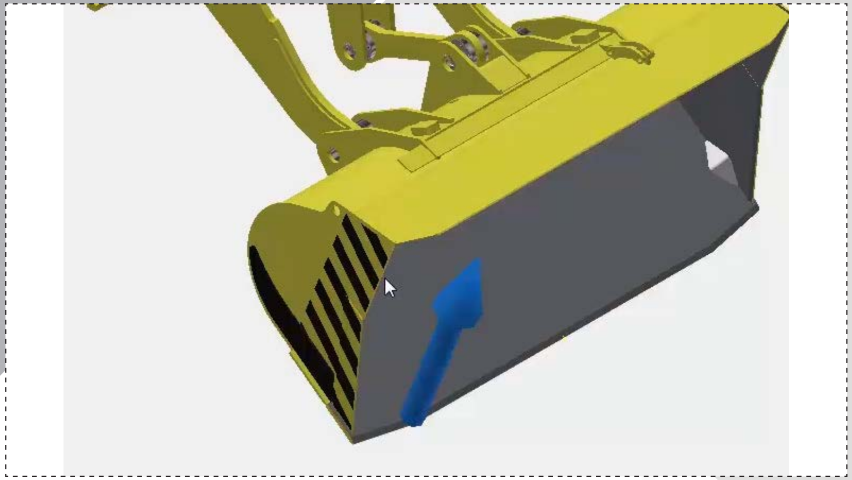
Apply load force ‘Dynamic Simulation > Load > Force’.

9.



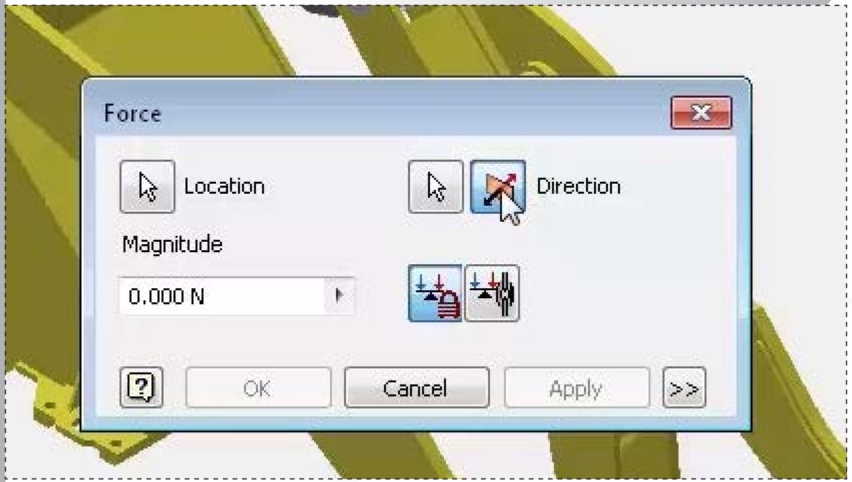
In the graphic window, select a point on the bottom of the bucket as shown.

10.



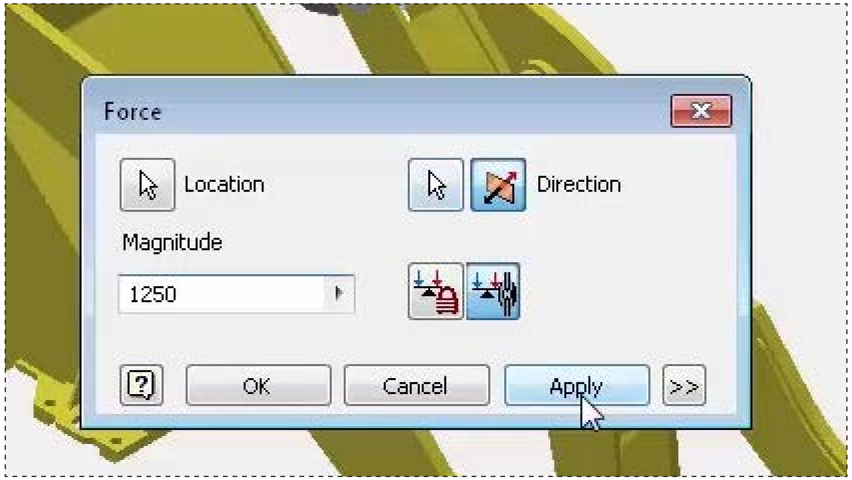
Select the edge shown to specify the load direction.

11.



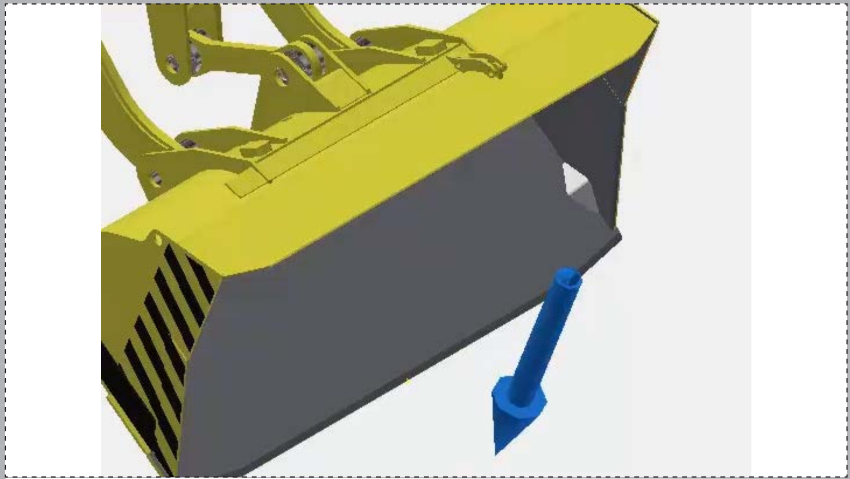
Flip the load direction so that it is pointing down.

12.



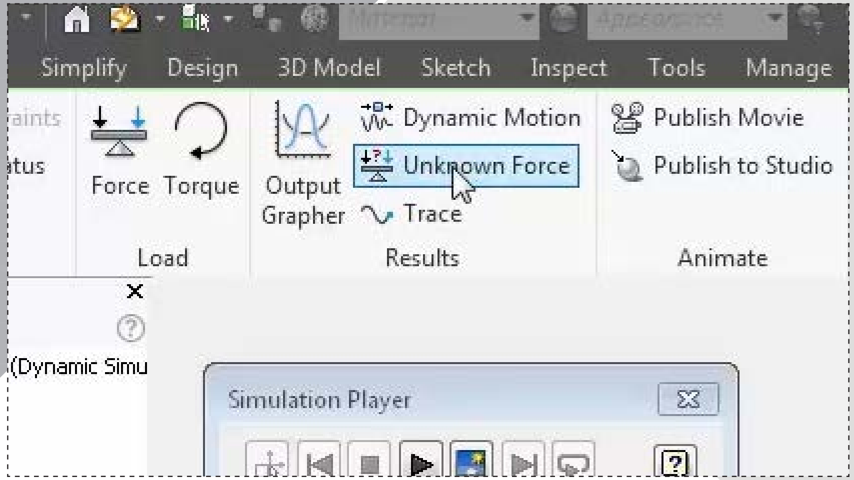
Switch to 'Associative Load Direction', and enter a value of 1250 N, then click Apply.

13.



Repeat step 9 - 12 on the mirror oposite side of the bucket.

14.

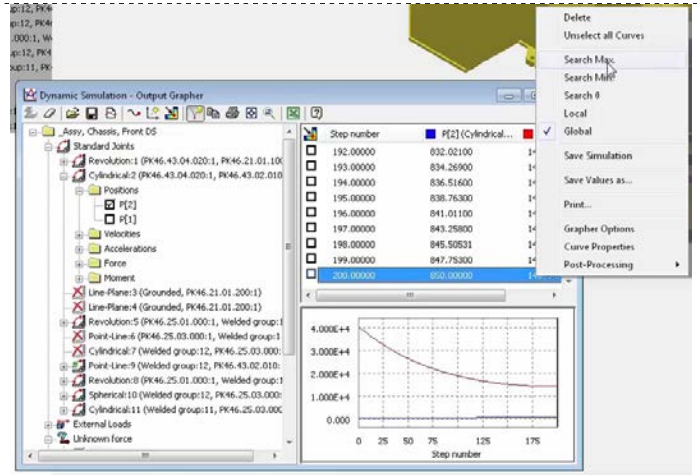


Select ‘Unknown Force Dynamic Simulation > Results > Unknown Force’

15.

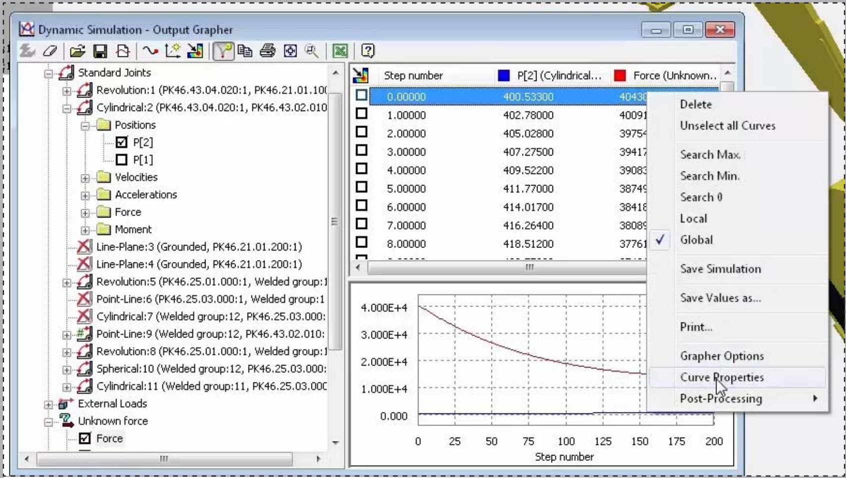
On the ‘Unknown Force’ dialog box, uncheck ‘Display’ then click ‘OK’.

16.



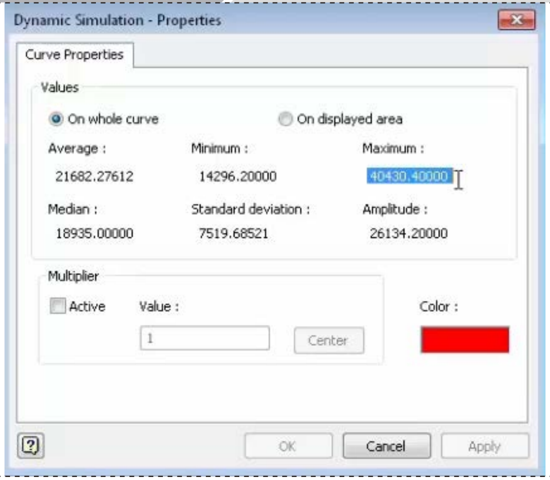
Once the simulation completes in the ‘Output Grapher’, right click on ‘Step 200’, select ‘Search Max’.

17.



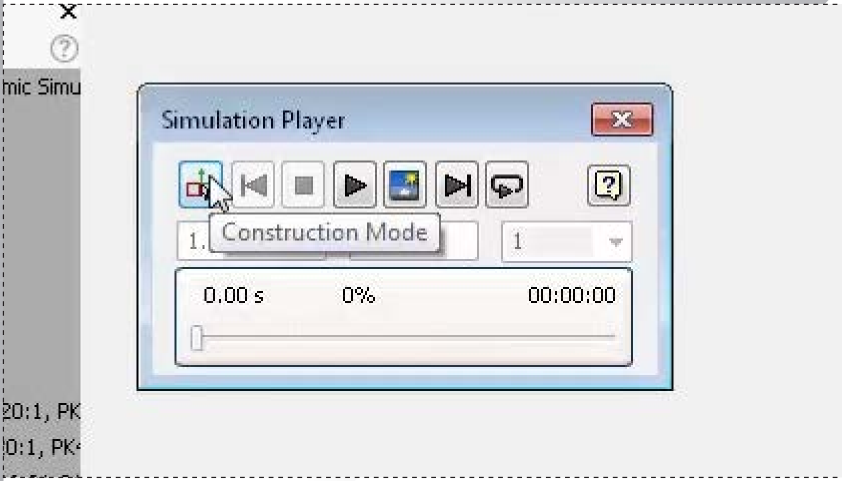
Right click the hightlighted step (Step 0), select 'Curve Properties'.

18.



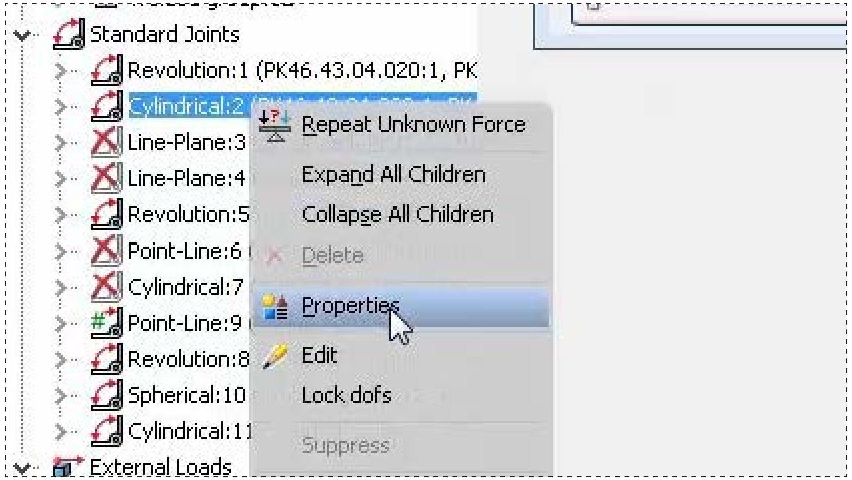
Note the Maximum force of 40430.40
Close the 'Properties' dialog box. Close the 'Output Grapher'.

19.



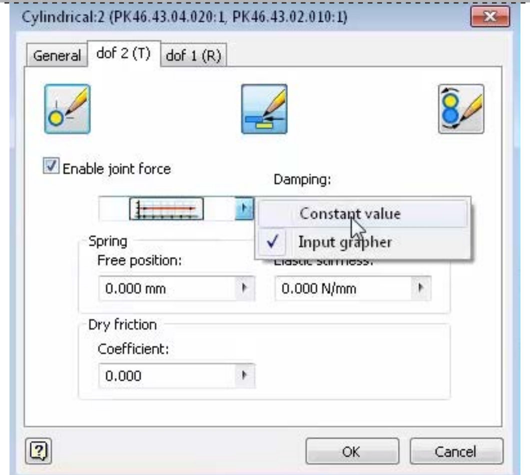
Exit the animation by clicking 'Construction Mode' on the 'Simulation Player'. Save your work.

20.



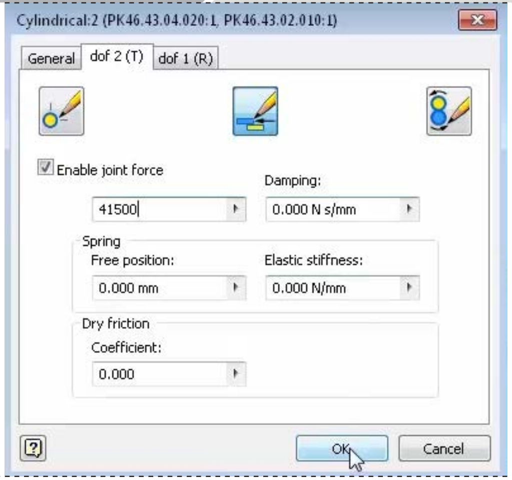
In the browser, under 'Standard Joints', right click 'Cylindrical:2' and select 'Properties' from the short cut menu.

21.



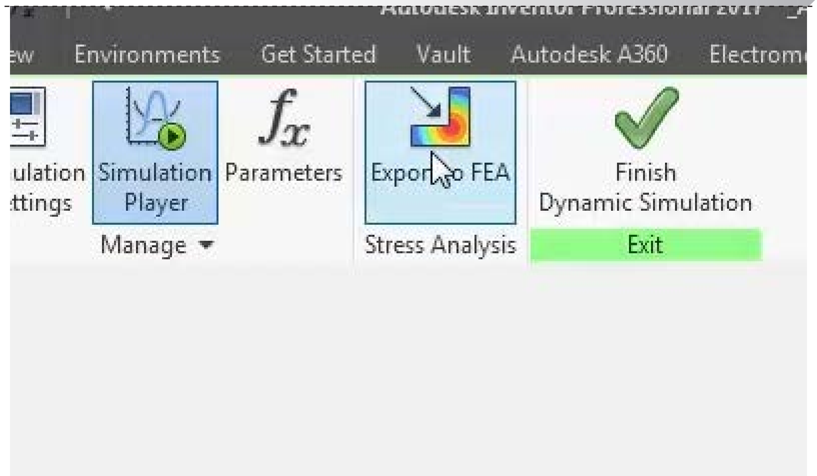
In the 'Joint Properties' dialog box, on the 'dof 2 (T)' tab, select 'Edit Joint Force', check on 'Enble Joint Force', select 'Constant Value'.

22.



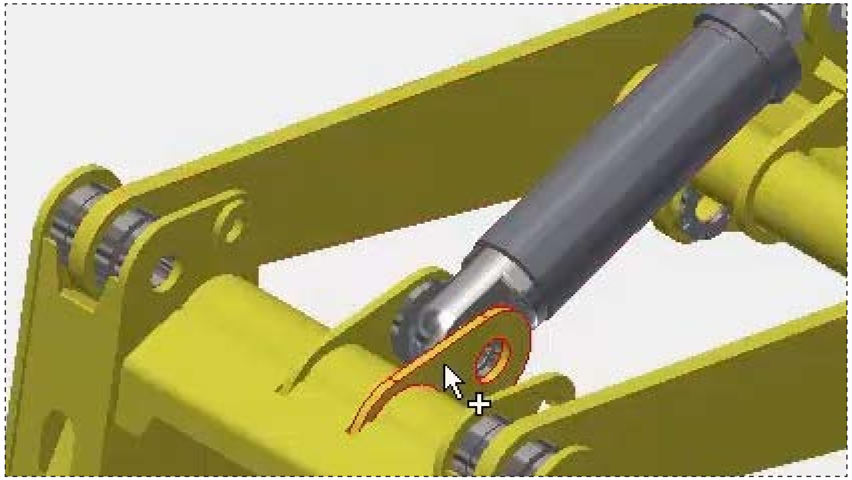
Enter a 'Constant Value' of 41500 N, Click 'OK'.

23.



Select 'Export to FEA Dynamic Simulation > Stress Analysis > Export to FEA'.

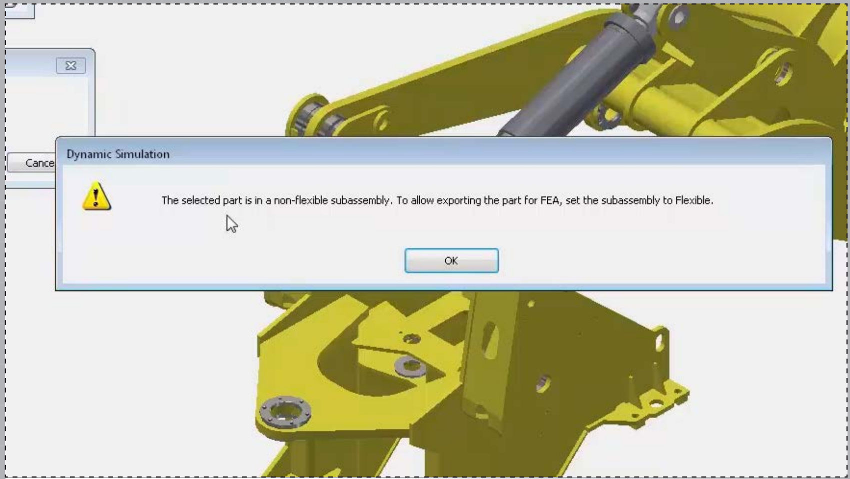
24.



In the graphics window, select the part shown.

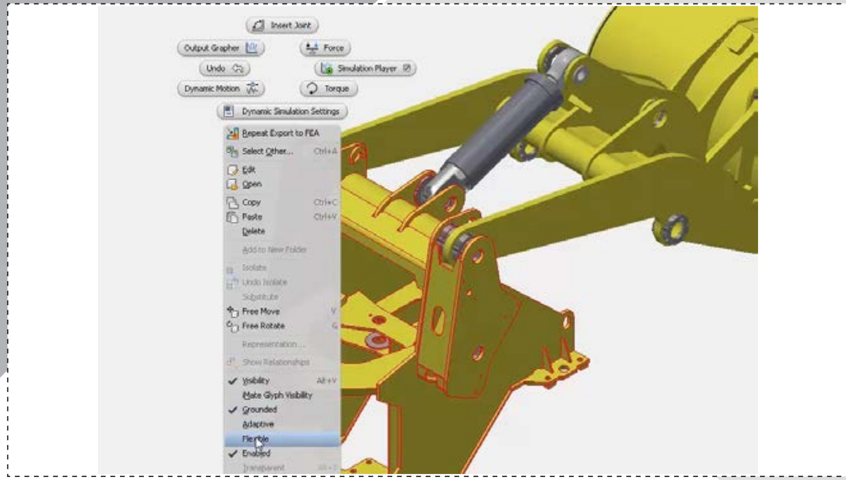
PART 2: DYNAMIC SIMULATION

25.



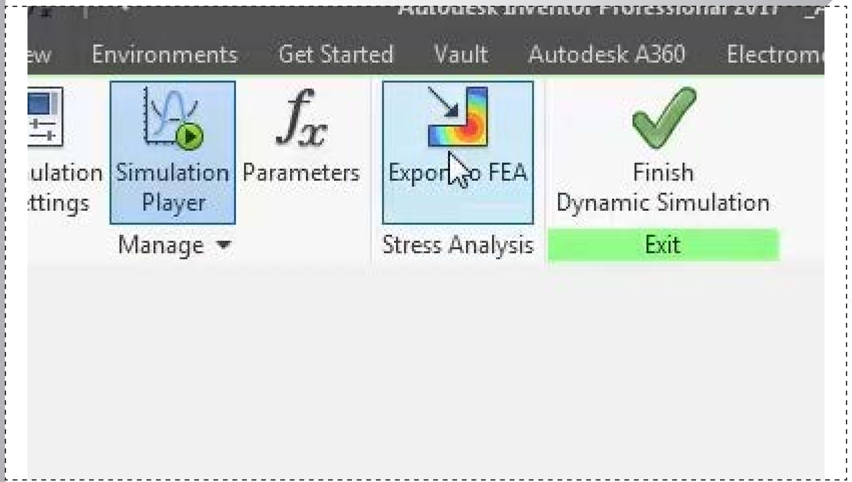
Select ‘OK’ on the warning message dialog box.

26.



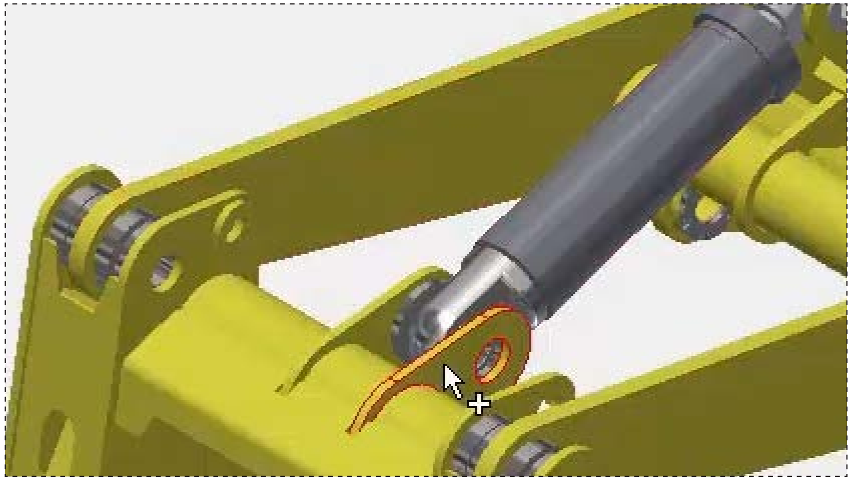
In the graphics window, select the sub-assembly shown, right click and select ‘Flexible’ from the short cut menu.

27.



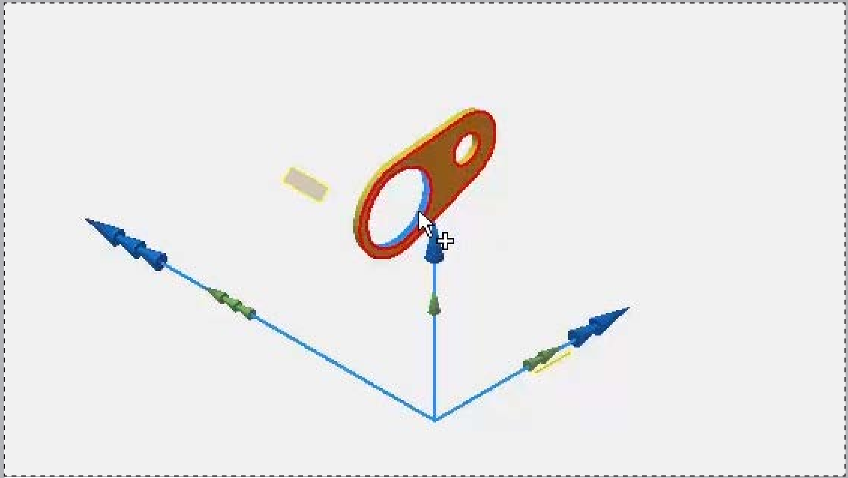
Select ‘Export to FEA Dynamic Simulation > Stress Analysis > Export to FEA’.

28.



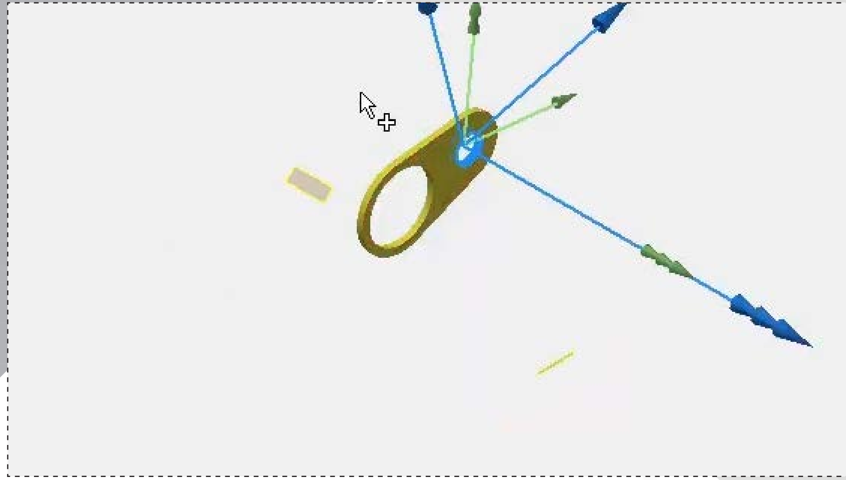
In the graphics gindow, select the part shown. On the ‘Export to FEA’ dialog box, click ‘OK’

29.



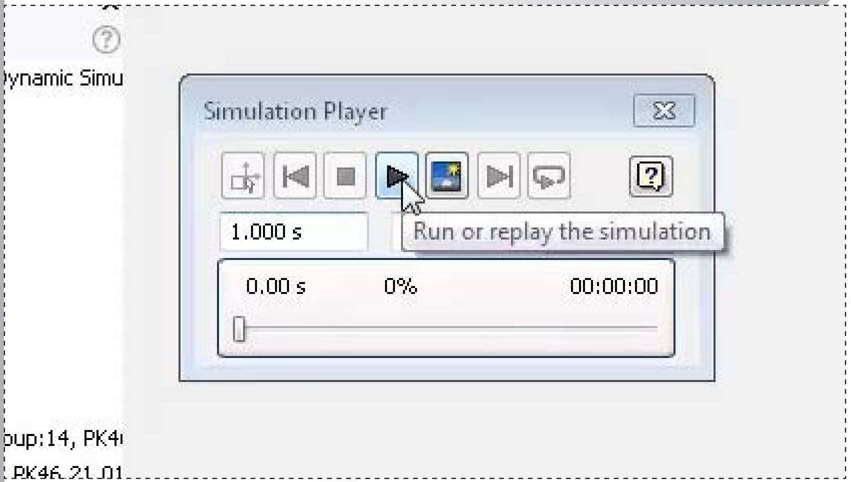
On the ‘FEA Load-Bearing Faces Selection’, under ‘Joints to complete’: ‘Load Bearing’, select ‘Welding:13’. Select the large inside circular face shown.

30.



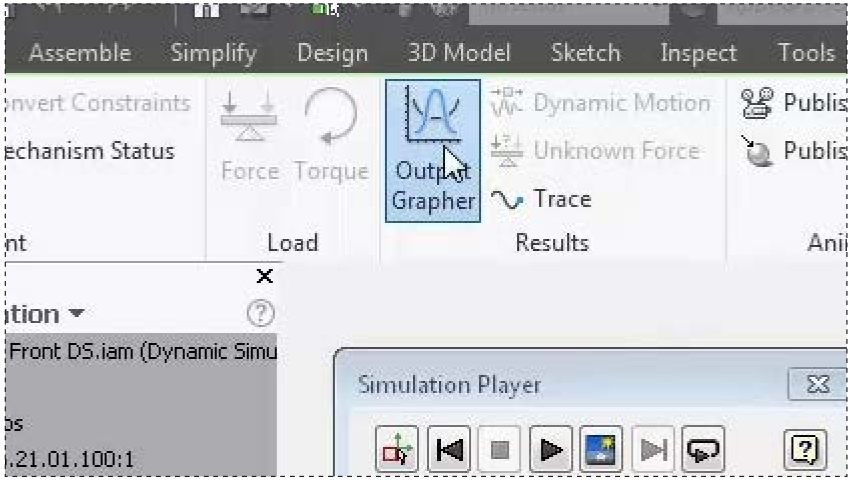
On the ‘FEA Load-Bearing Faces Selection’, under ‘Joints to complete’: ‘Load Bearing’, select ‘Revolution:14’. Select the small inside circular face shown. Click ‘OK’.

31.



Run the simualtion, click ‘Run’ on the ‘Simulation Player’.

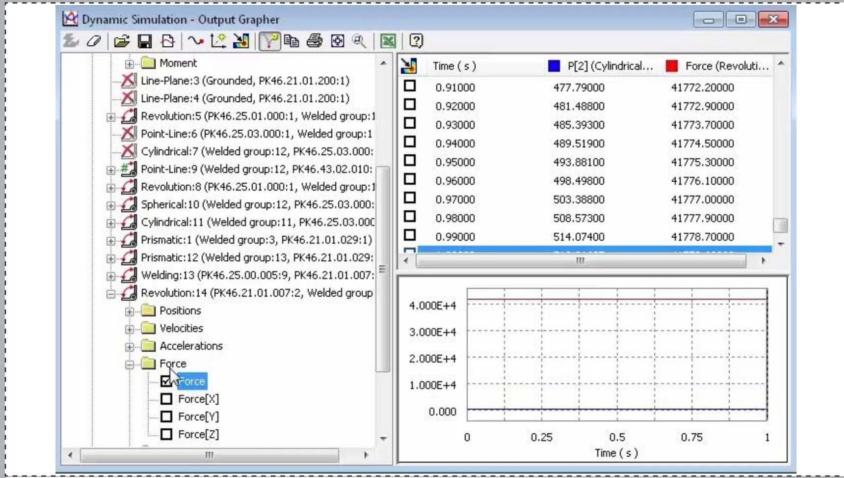
32.



Open the ‘Output Grapher’.
‘Dynamic Simulation > Results > Output Grapher’.

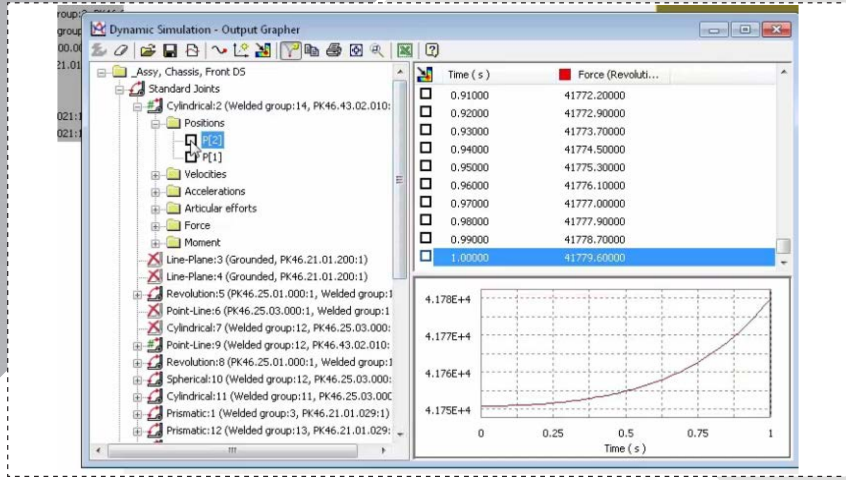
PART 2: DYNAMIC SIMULATION

33.



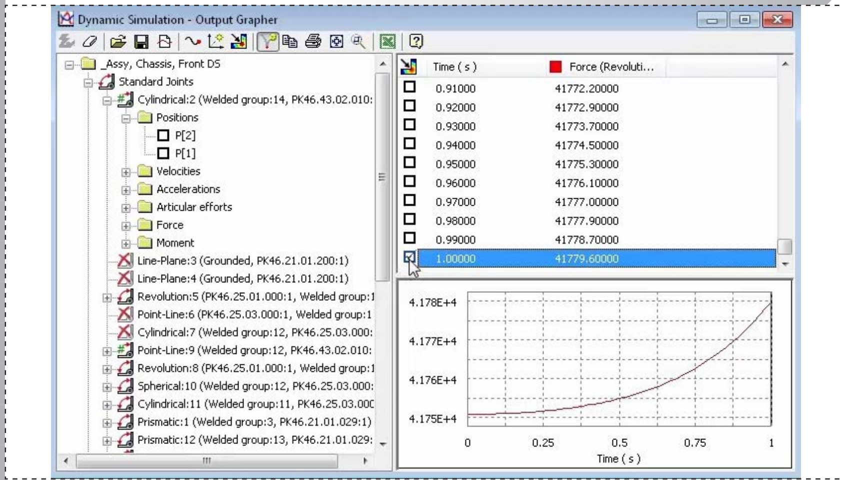
On the ‘Output Grapher’, expand ‘Joint Revolution:14’, then expand the Force folder, then check ‘On Force’.

34.



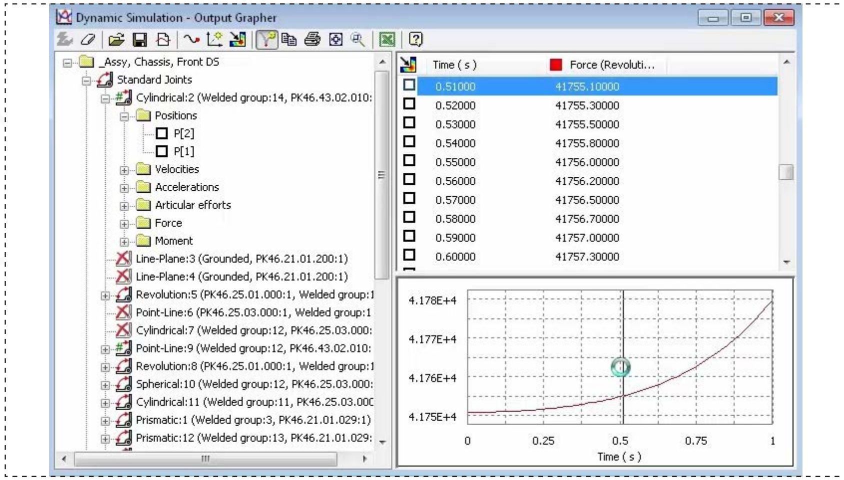
On the ‘Output Grapher’, under ‘Joint Cylindrical:2’, uncheck ‘P[2]’.

35.



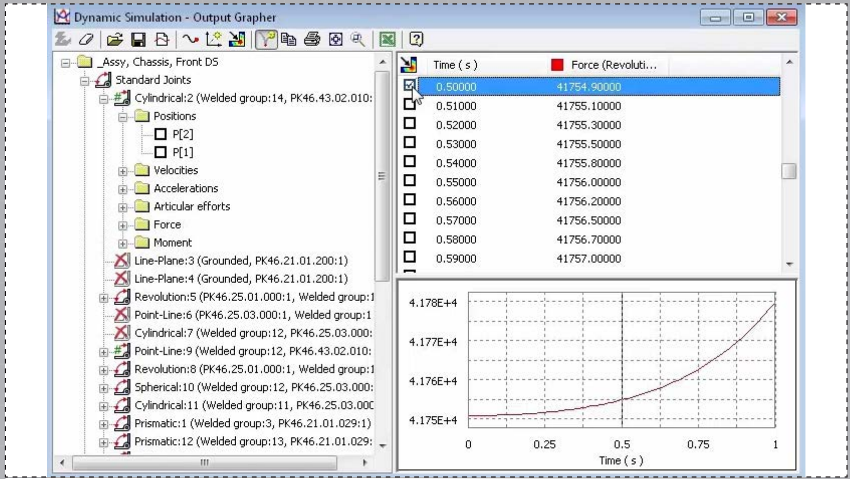
On the ‘Output Grapher’, check ‘Time Step 1.00000’ to export it.

36.



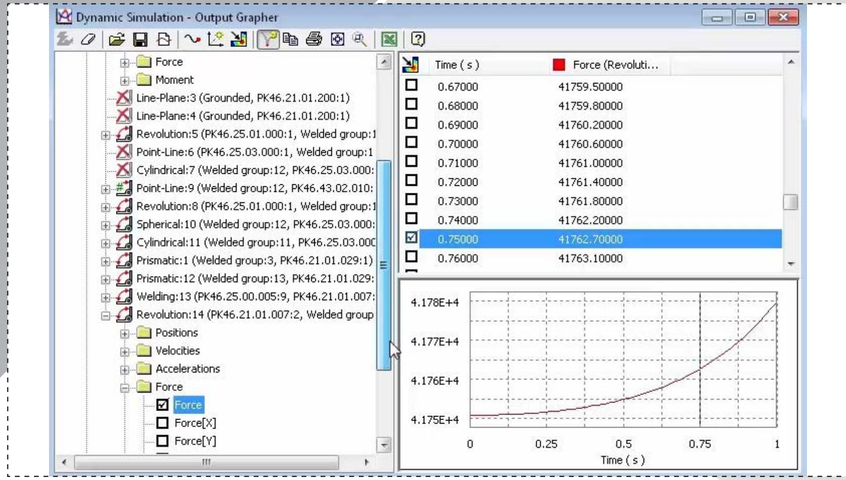
On the ‘Output Grapher’, double click on the graph at the ‘0.5 time mark’.

37.



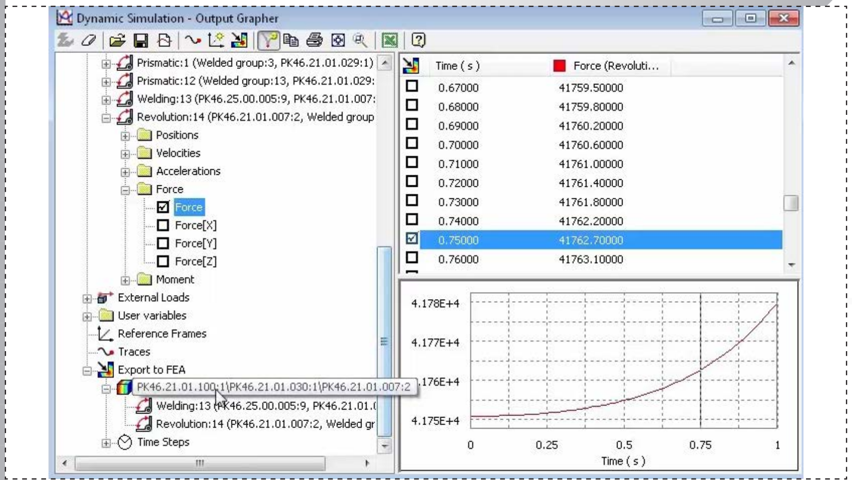
On the ‘Output Grapher’, check ‘Time Step 0.50000’ to export it.

38.



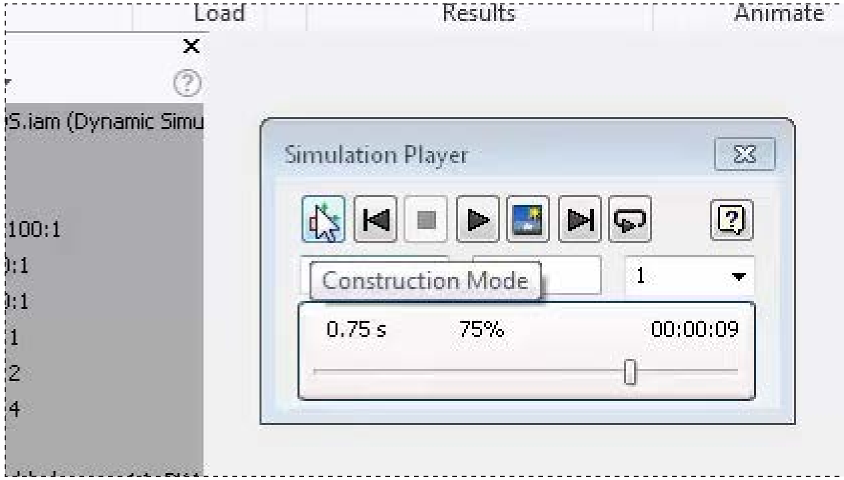
Repeat previous steps for the 0.75 time mark in the graph.

39.



Notice that the time steps have been added to the ‘Export to FEA’ folder in the ‘Output Grapher’. Close the ‘Output Grapher’.

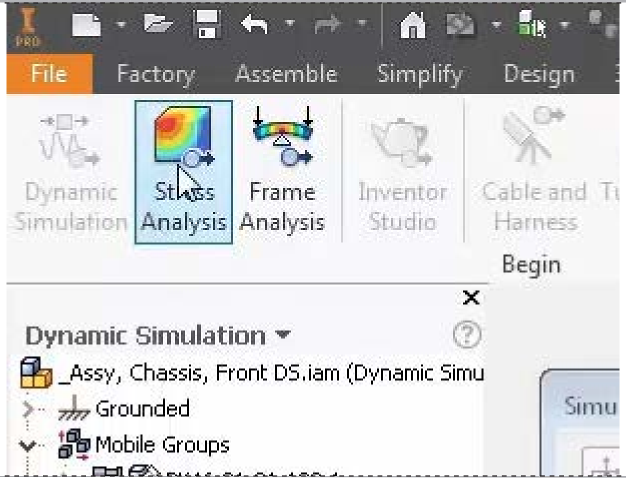
40.



Exit the animation by clicking ‘Construction Mode’ on the ‘Simulation Player’. Save your work.

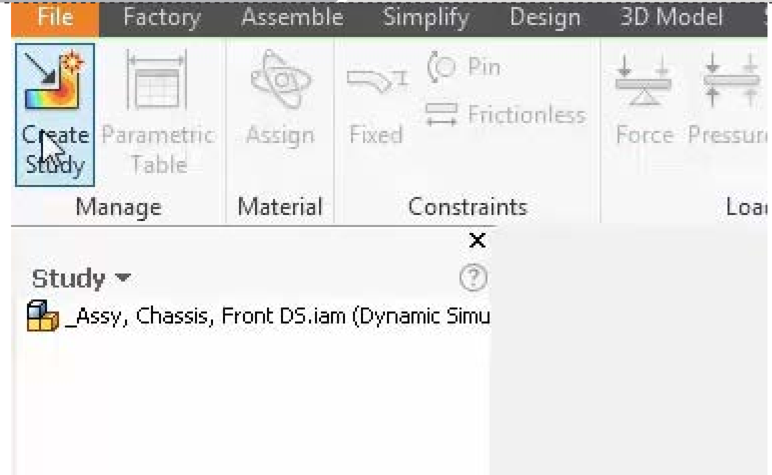
PART 3: DYNAMIC SIMULATION

41.



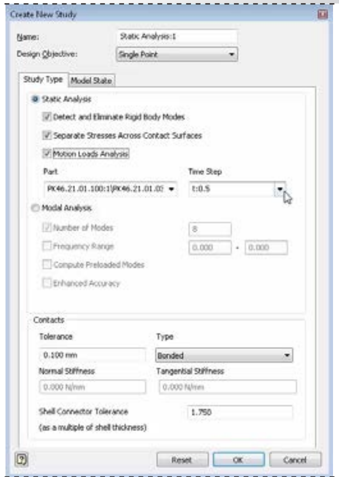
Start the 'Stress Analysis Environment, Environments > Beign > Stress Analysis'.

42.



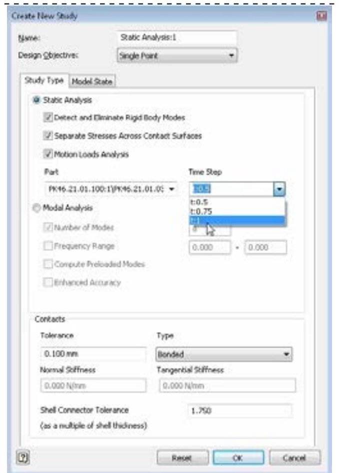
Create a 'New Study Analysis > Manage > Create Study'.

43.



On the 'Create New Study' dialog box, check on, 'Detect and Elminate Rigid Body Modes', 'Separte Stresses Across Contact Surfaces', 'Motion Loads Analysis'.

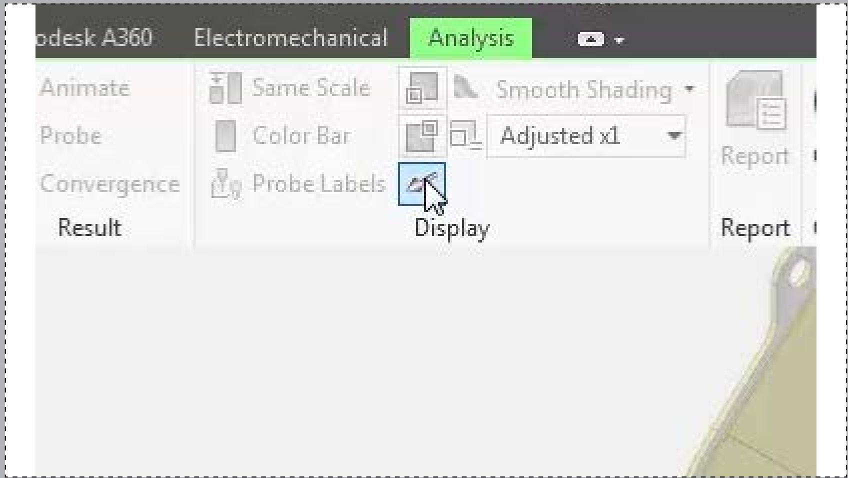
44.



From the 'Time Step' drop down list, select the 't:1' time step Click 'OK'.

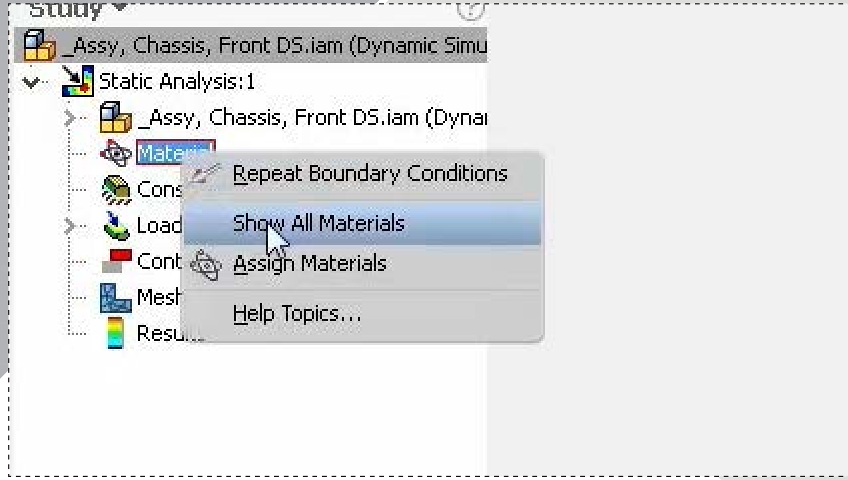
PART 3: DYNAMIC SIMULATION

45.



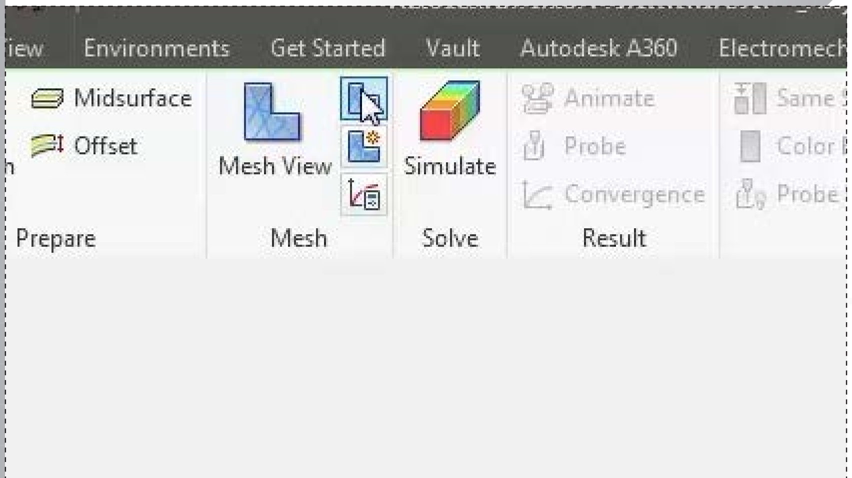
Turn Off the display of the load glyphs by unchecking ‘Boundary Conditions Analysis > Display > Boundary Conditions’.

46.



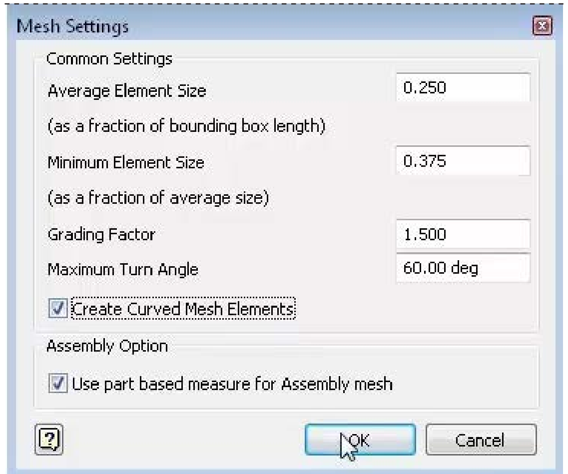
In the browser, right click on ‘Materials’, select ‘Show All Materials’ from the short cut menu.

47.



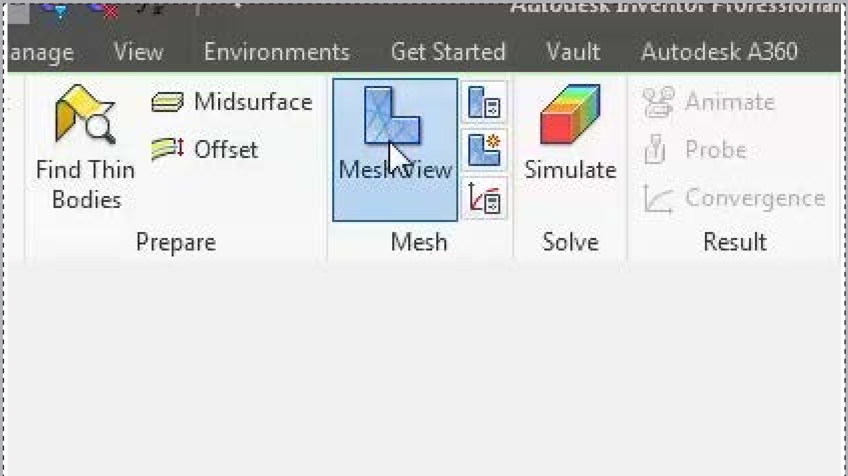
Click ‘Mesh Settings Analysis > Mesh > Mesh Settings’.

48.



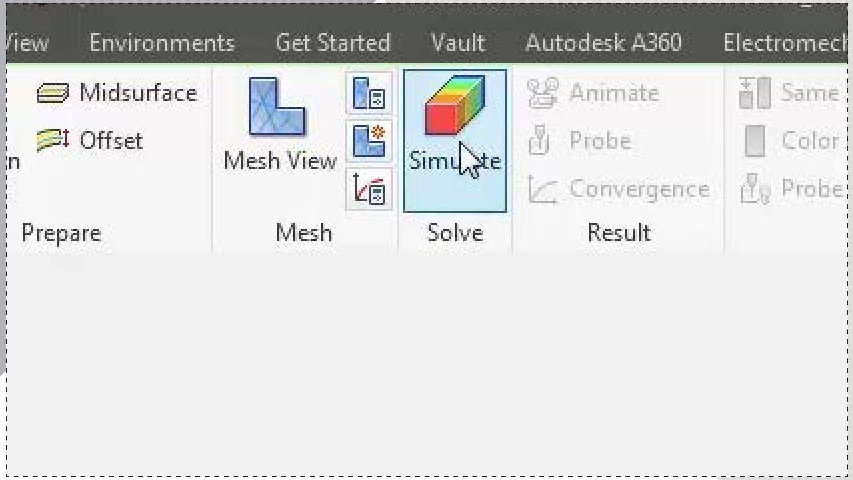
On the Mesh Setting dialog box, enter the values: Average Element Size: 0.250, Minimum Element Size: 0.375, Check ‘ON’, ‘Create Curved Mesh Elements’. Click ‘OK’.

49.



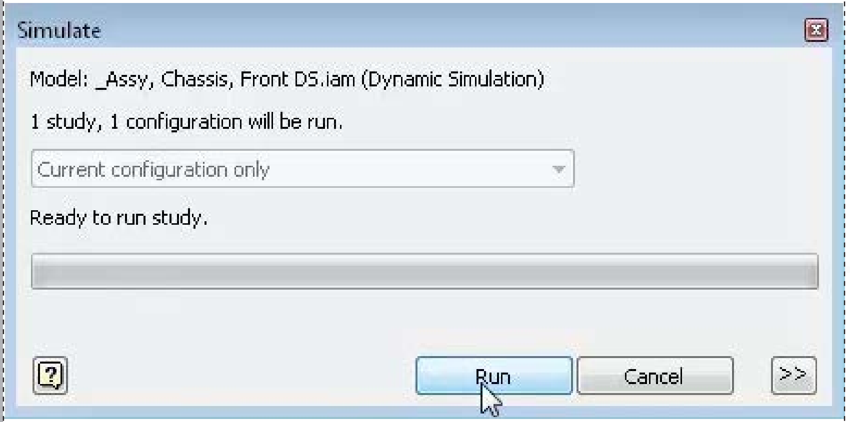
Click 'Mesh View Analysis > Mesh > Mesh View'. Turn off Mesh View by clicking 'Mesh View' again.

50.



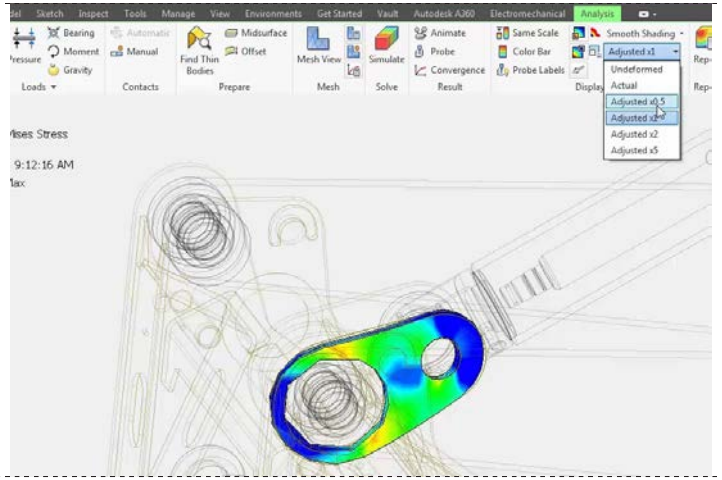
Run simulation, click 'Simulate Analysis > Solve > Simulate'.

51.



Click 'Run', on the 'Simulate' dialog box.

52.

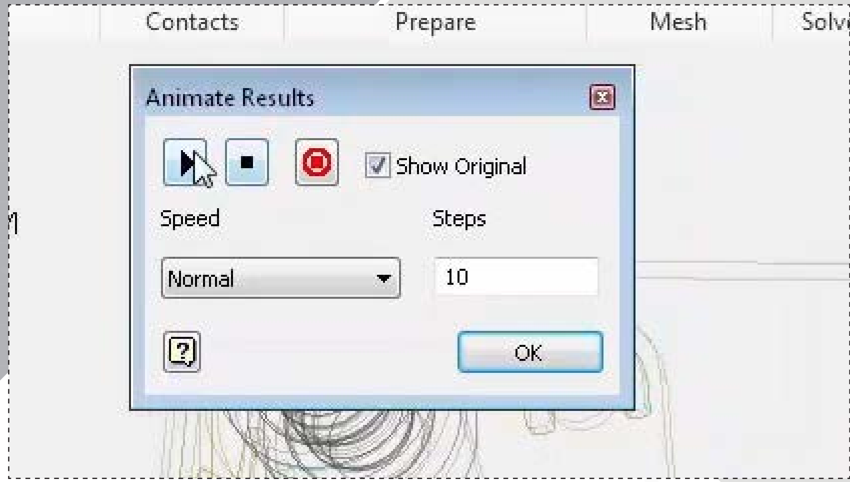


Adjust the displacement display, Select 'Adjusted x0.5 Analysis > Display > Displacement Adjustment'.

53.

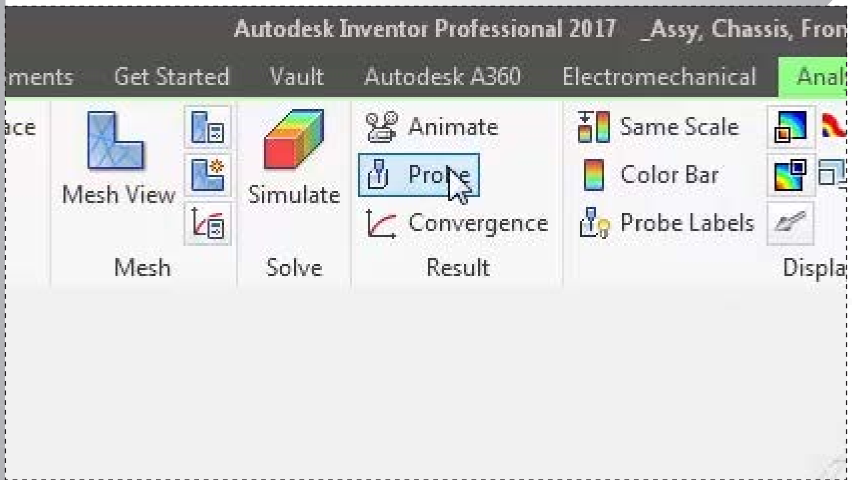
Animate the analysis results ‘Analysis > Result > Animate’.

54.



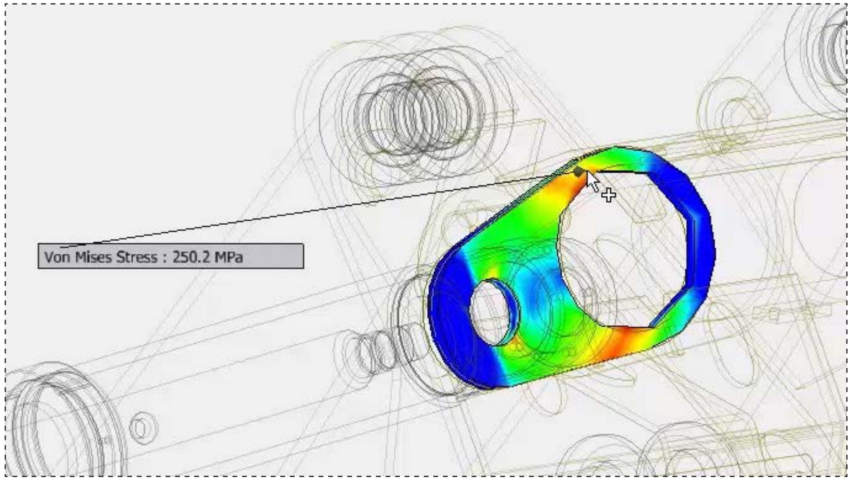
Click the ‘Play’ button on the ‘Animate Results’ dialog box. View the results animation. Areas of red are areas of concern. Click ‘OK’ to close the animation.

55.



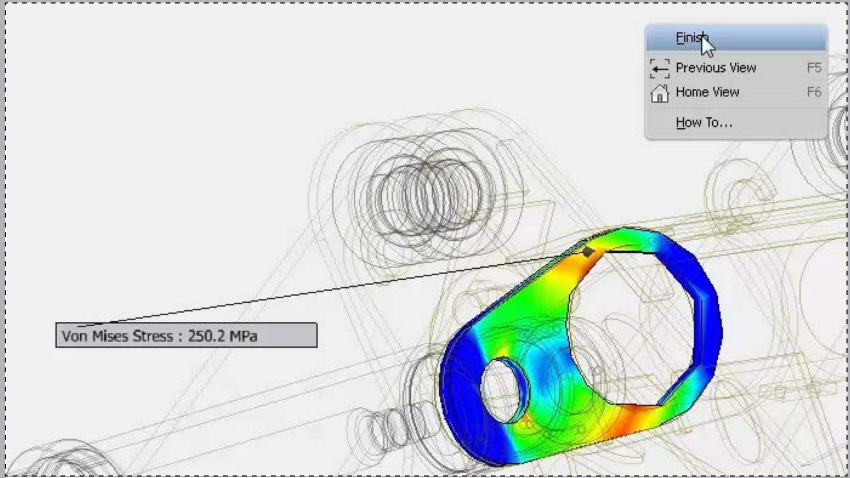
Place Probe to measure result at a selected point on the part ‘Analysis > Result > Probe’.

56.



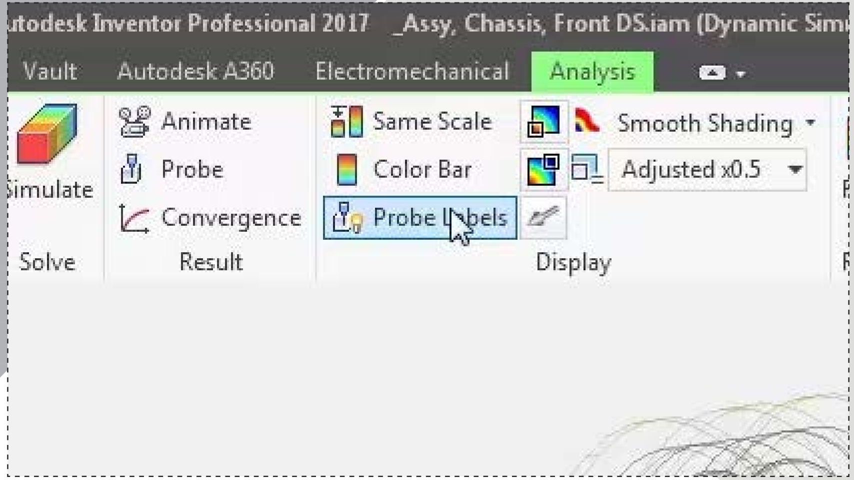
Click on the part in the graphics window to place a Probe.

57.



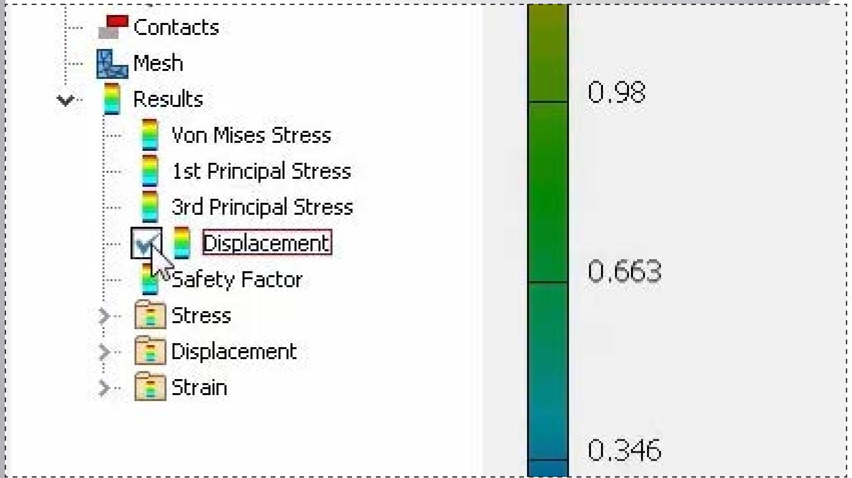
Right click in the graphics gindow, select ‘Finish’ from the short cut menu.

58.



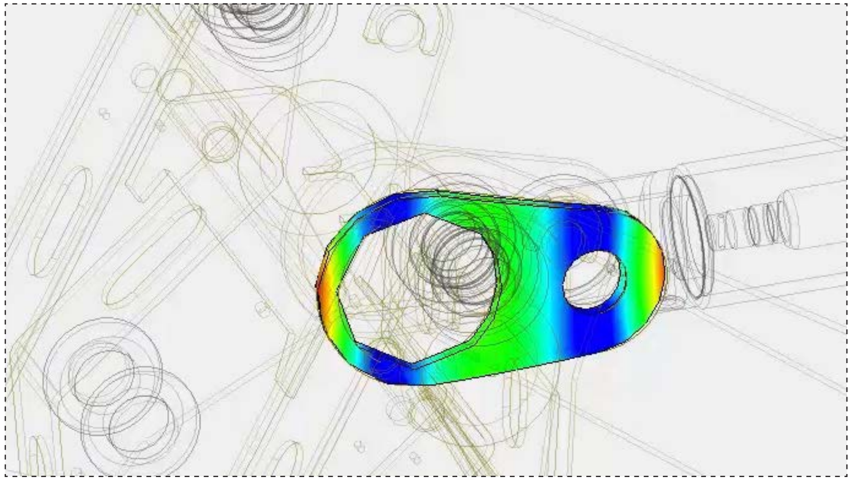
Hide the Probe display by clicking Off the Probe Lables ‘Analysis > Display > Probe Lables’.

59.



In the browser, double click ‘Displacement’ under the ‘Results’.

60.



View the Displacement results in the graphics window. Areas of red are areas of concern. To finsh the analysis, Click ‘Finish Analysis’.