AUTODESK **INVENTOR**

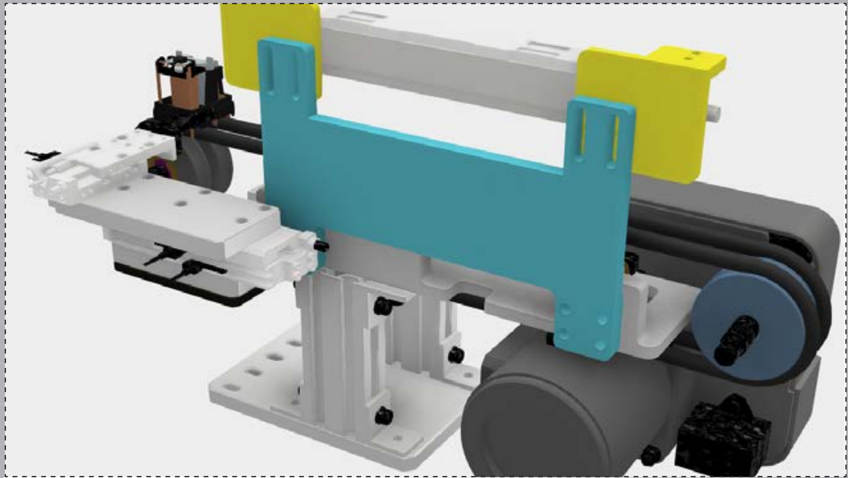
Trial Projects

Shape Generator

Design a stiff lightweight part

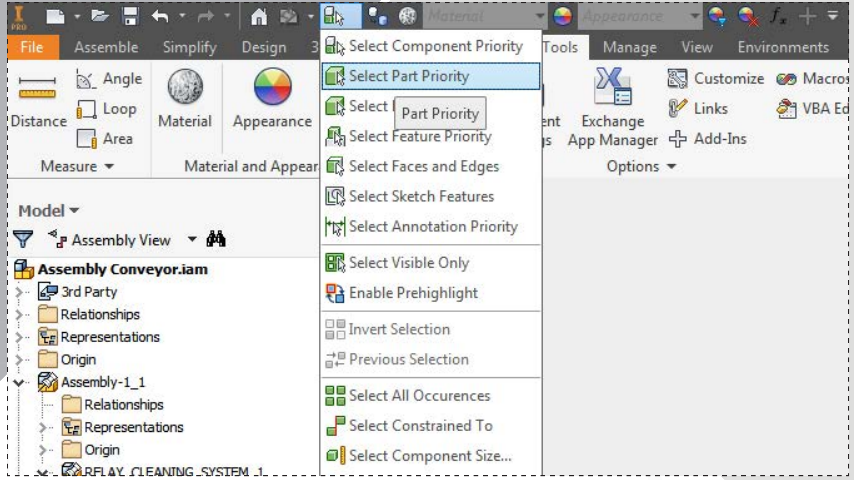
PART 1: SHAPE GENERATOR

1.



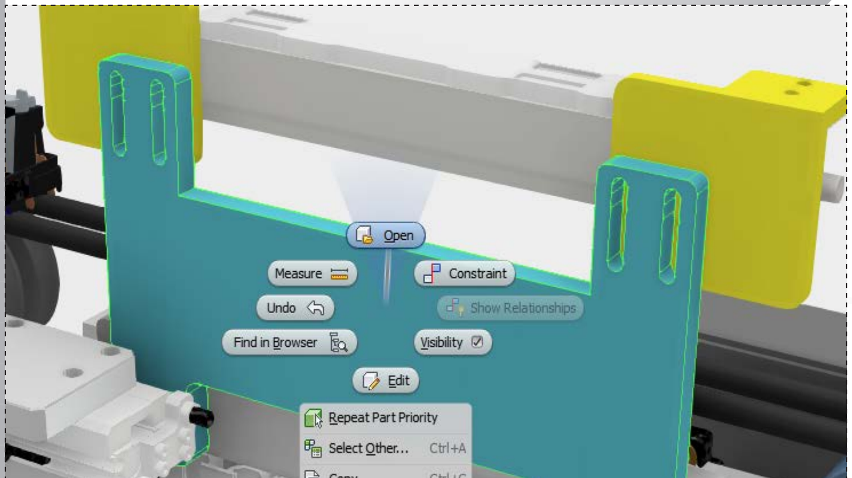
In Inventor, click the ‘Projects’ icon in the ribbon. Navigate to where you saved the project files and select A*ssembly Conveyor.ipj*. Then open ***Assembly Conveyer.iam***.

2.



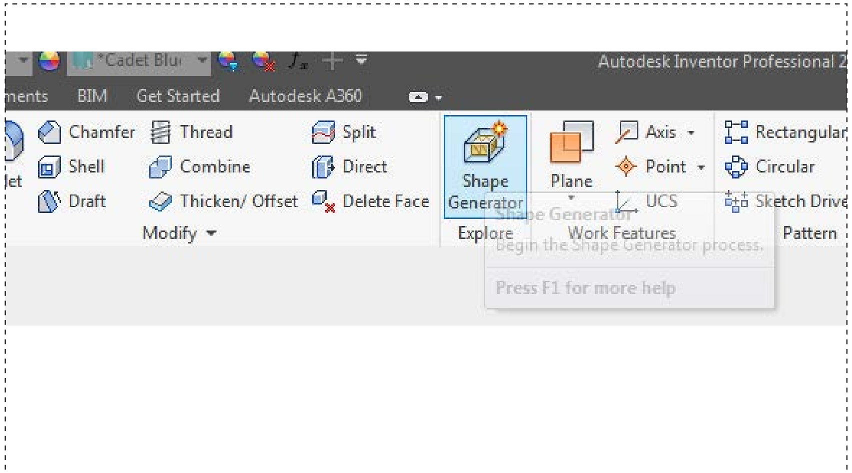
Next you will separate the blue part to lighten it. Begin by separating it from the assembly using the ‘Select Part Priority’ from the ‘Selection Priority’ drop down menu.

3.



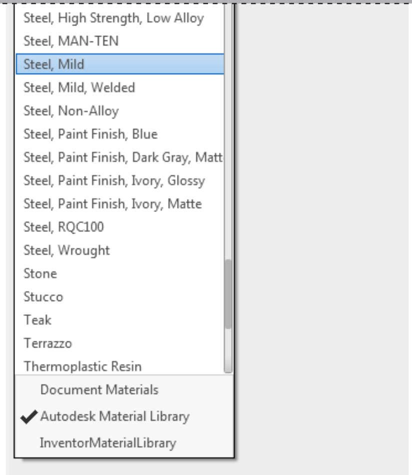
Next select the blue part, right click on it and select ‘Open’ to open the part. This will allow you to work on this part only.

4.



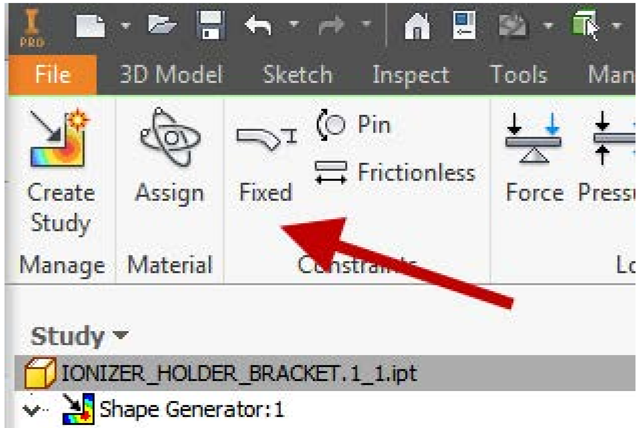
On the 3D Model tab select ‘Shape Generator’ in the ‘Explore’ panel.

5.



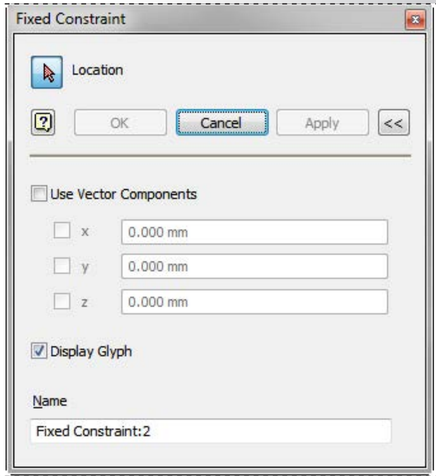
You will next need to set some variables for Inventor to make calculations. Go to the Material drop down menu and select 'Steel, Mild' in the 'Autodesk Material Library' to set the material for this part.

6.



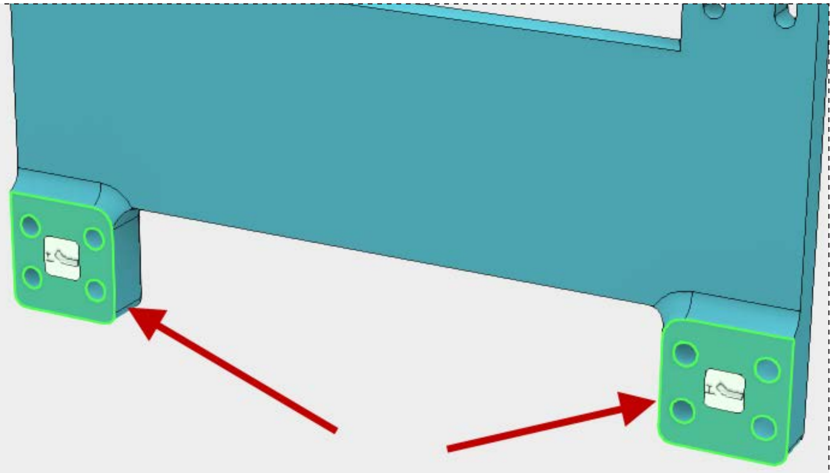
Next create constraints by going to the 'Constraints' panel and select 'Fixed'.

7.



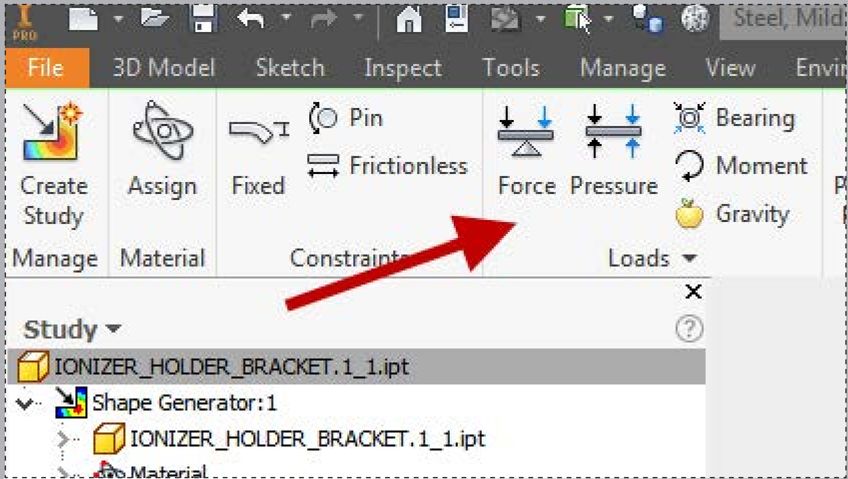
The 'Fixed Constraint' dialog appears.

8.



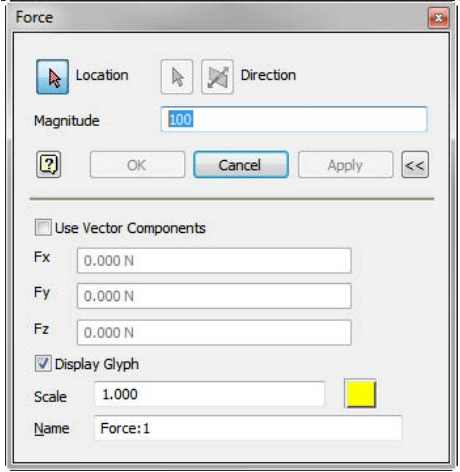
Select the two faces of the mounting pads as shown in the image below. Select 'OK'.

9.



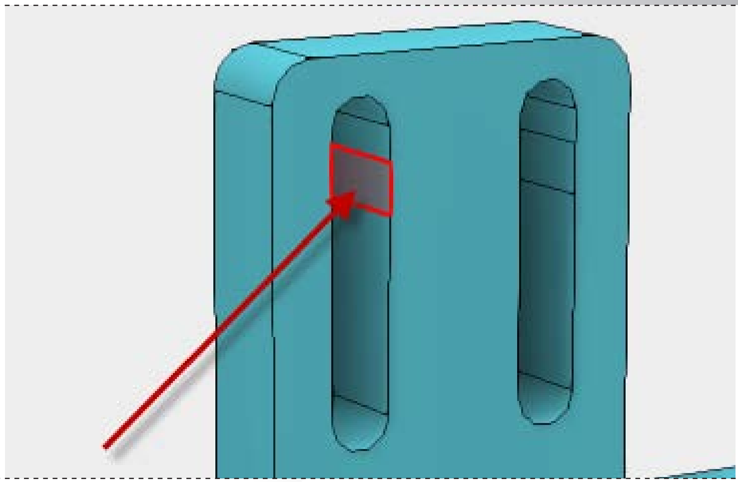
Next apply a load. Go to the ‘Loads’ panel and select ‘Force’.

10.



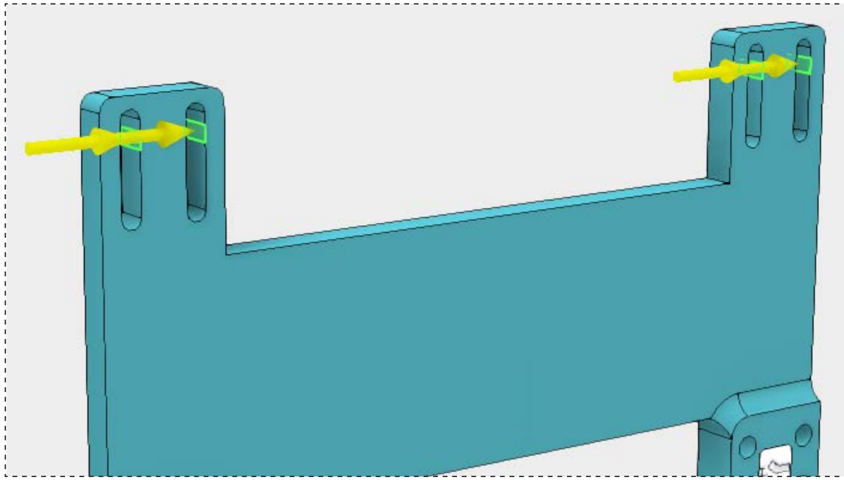
The ‘Force’ dialog appears, place a value of 100 in the ‘Magnitude box’.

11.



Next select the inside face edge on all for all four mounting slots as shown in the image to the right.

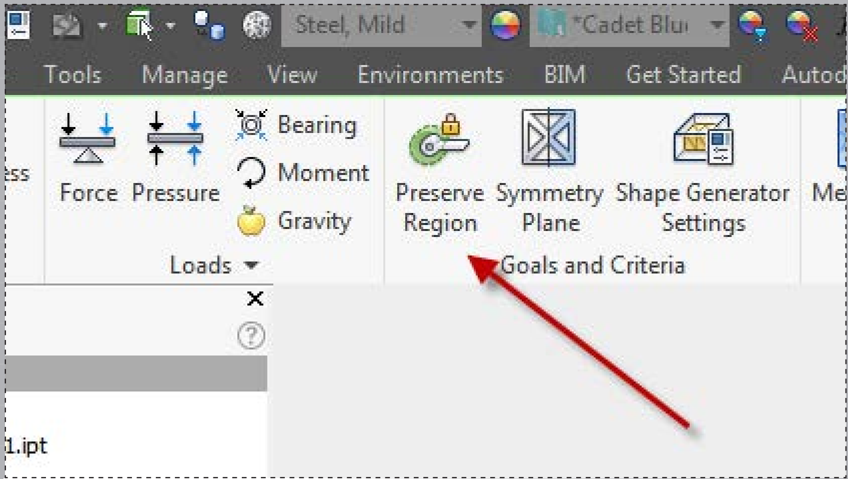
12.



Select ‘OK’.

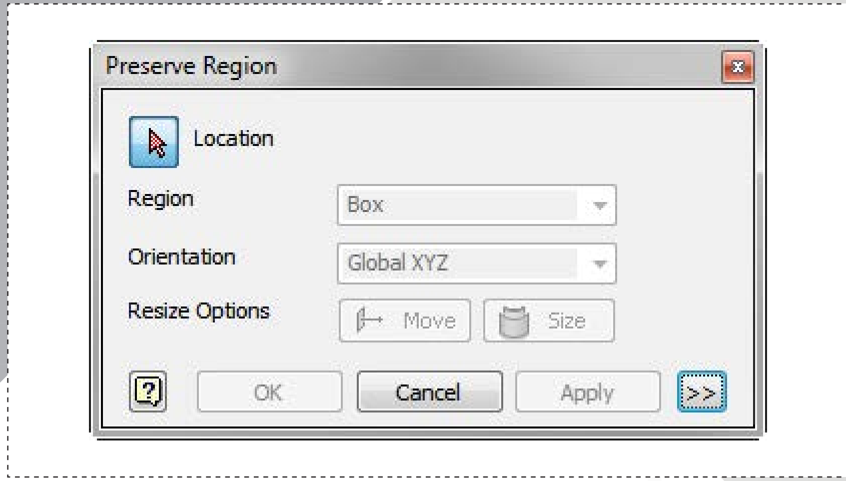
PART 1: SHAPE GENERATOR

13.



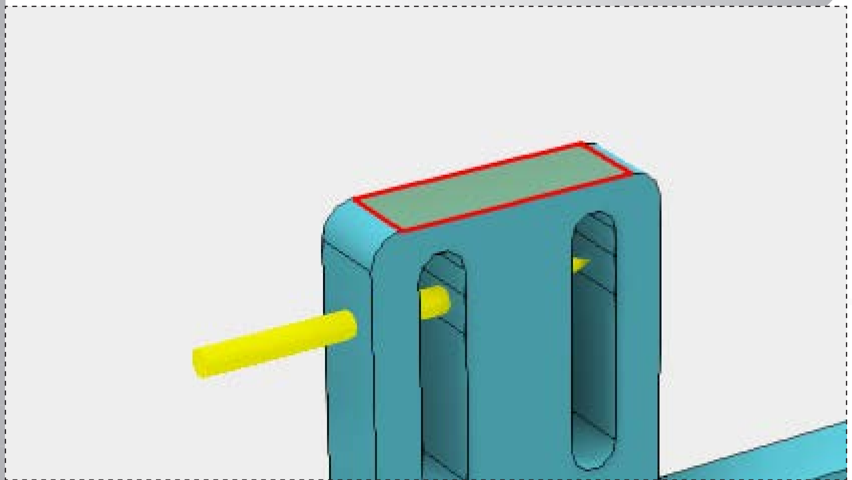
Next select 'Preserve Region' in the 'Goals' and 'Criteria' panel.

14.



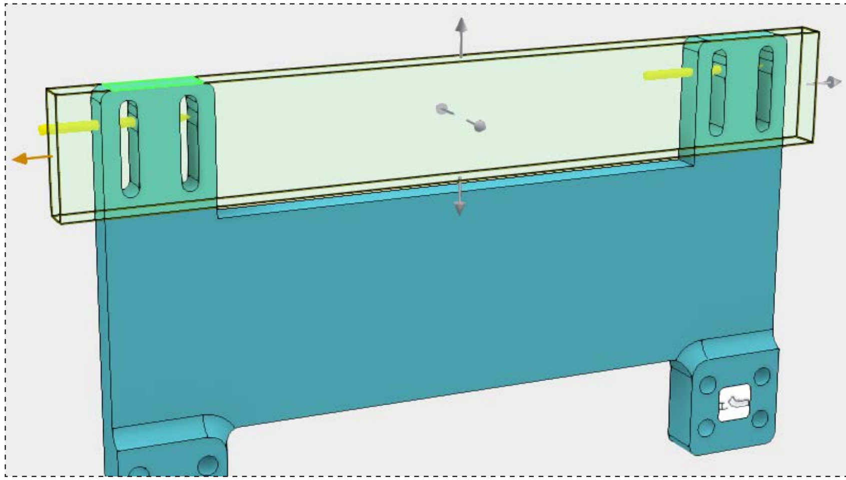
The 'Preserve Region' dialog appears.

15.



Next select the top face of the part as shown and then use the arrows on the box to expand it around the top portion of the part as shown in the images below.

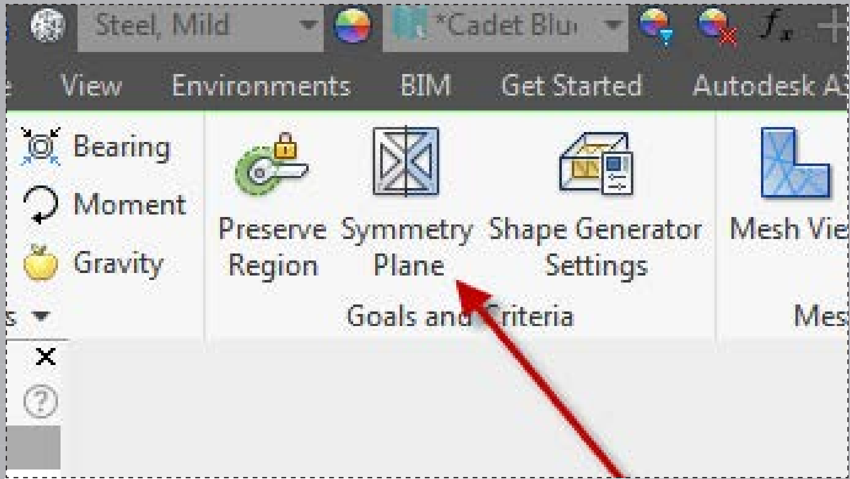
16.



Select 'OK'.

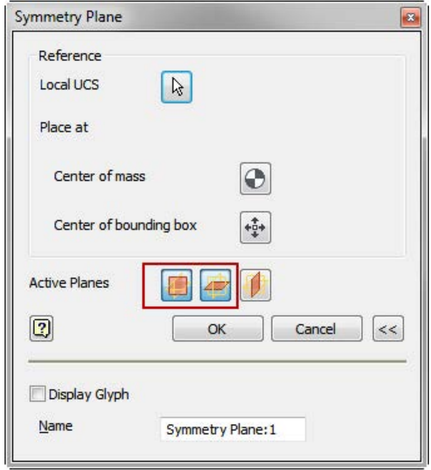
PART 1: SHAPE GENERATOR

17.



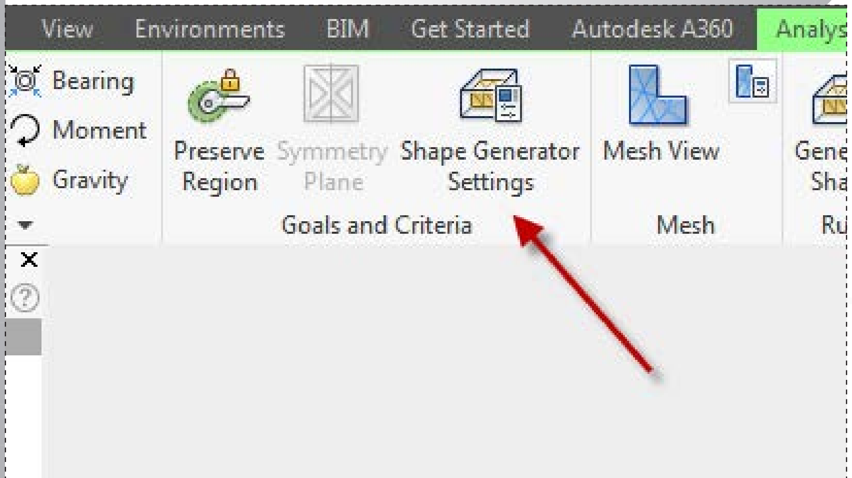
Next define symmetry planes by selecting ‘Symmetry Plane’ in the ‘Goals and Criteria’ panel. The ‘Symmetry Pane’ dialog appears. Under ‘Active Planes’ select ‘Horizontal’ and ‘Vertical’ for directions.

18.



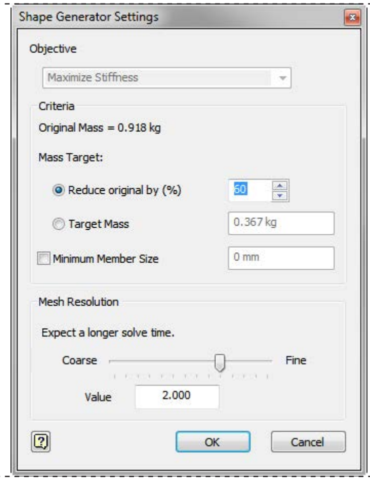
Select ‘OK’.

19.



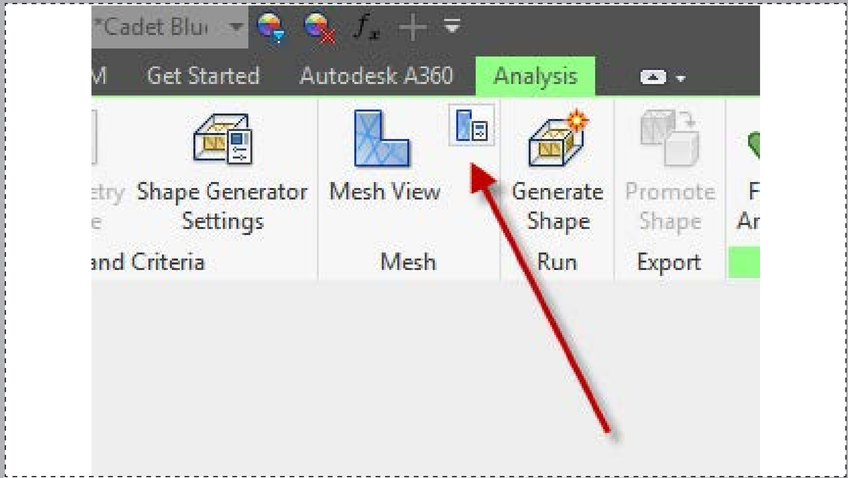
Next select ‘Shape Generator Settings’ in the ‘Goals and Criteria’ panel. The ‘Shape Generator’ Settings dialog appears.

20.



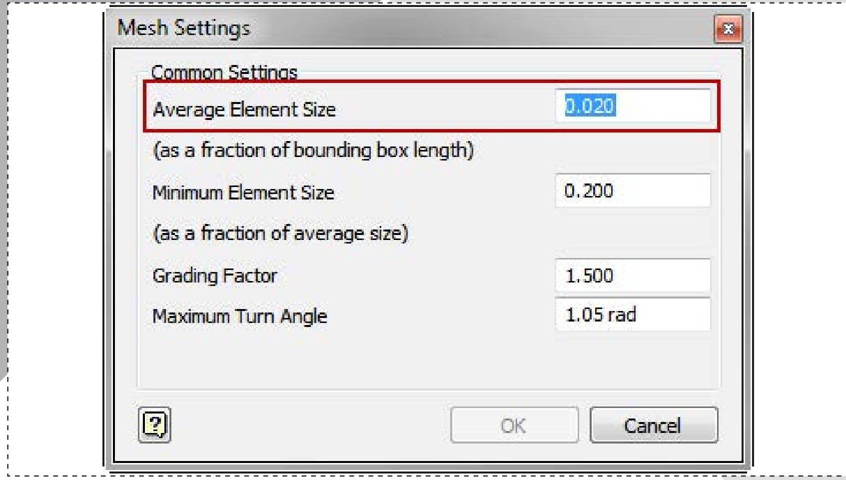
Under ‘Mass Target’ select ‘Reduce original by (%)’ and place a value of 60 in it’s box.

21.



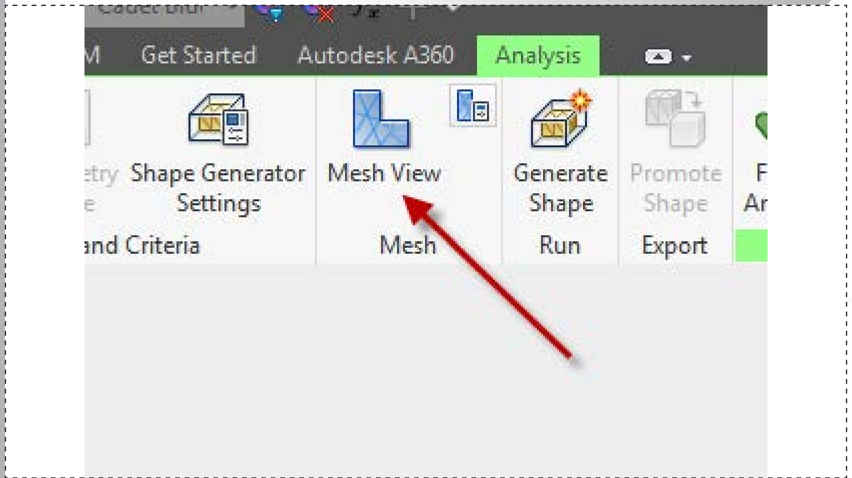
Next select 'Mesh Settings' button in the 'Mesh' panel. The 'Mesh Settings' dialog appears.

22.



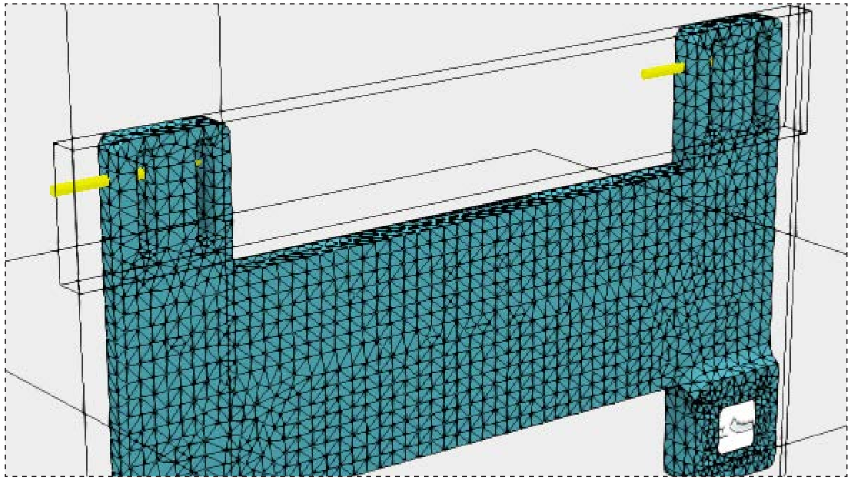
Under 'Common Settings' in the 'Average Element Size' box set the value to 0.02.

23.



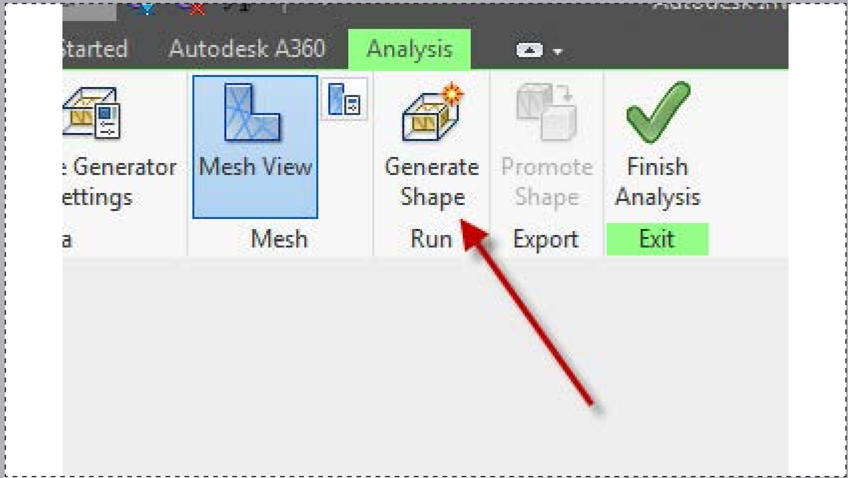
Select 'Mesh View' in the 'Mesh' panel. The part will begin to calculate.

24.



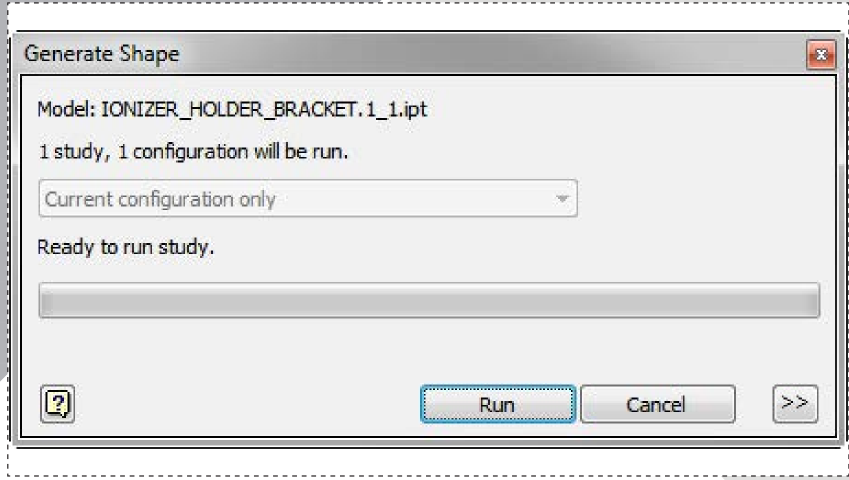
The final mesh appears after it's calculation, as shown in the image below.

25.



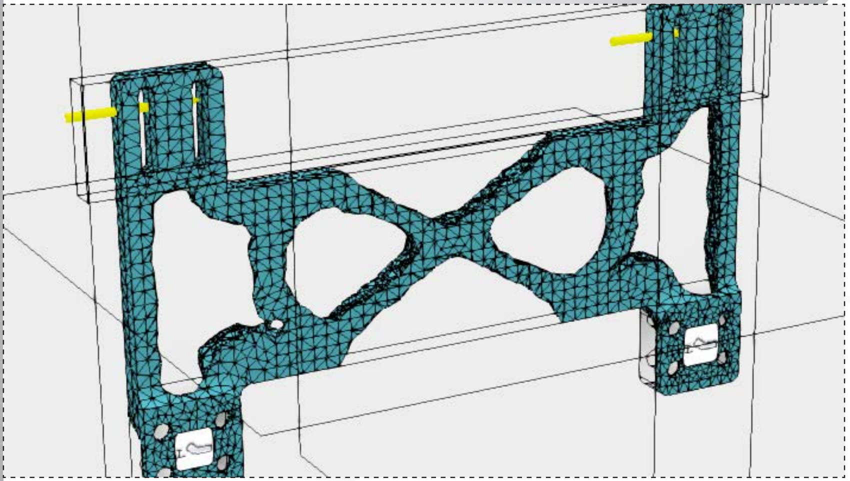
Finally select 'Generate Shape' in the 'Run' panel.

26.



The 'Generate Shape' dialog appears. Click 'Run'.

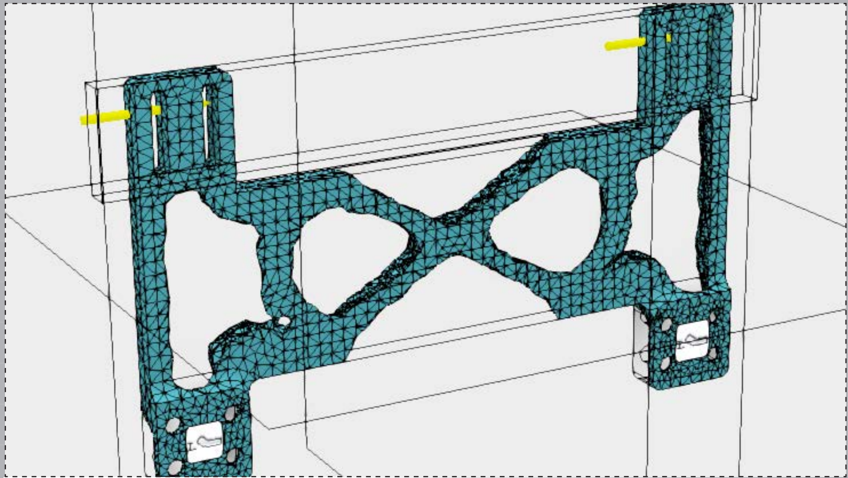
27.



The final shape study is completed and a final generative shape is created as shown in the image below.

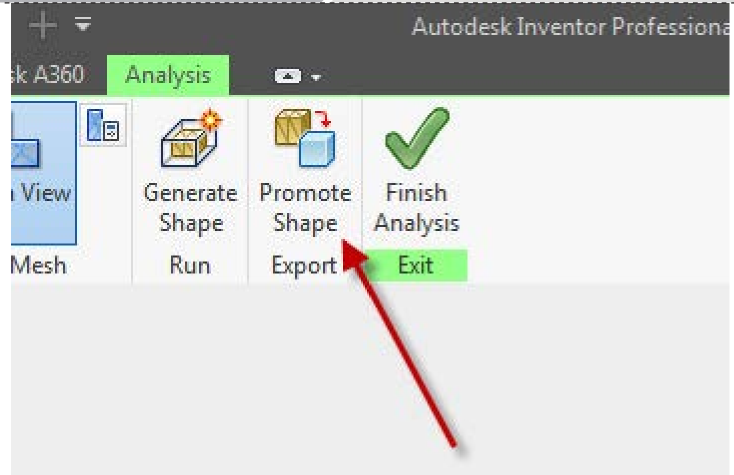
PART 2: SHAPE GENERATOR

28.



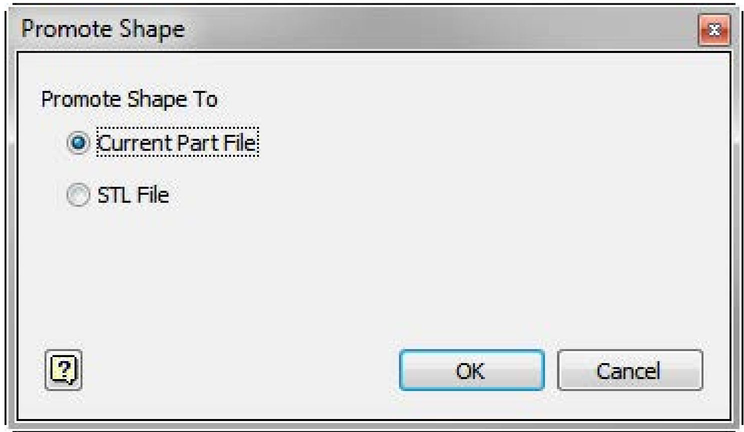
Open the shape generated part you created in the previous project.

29.



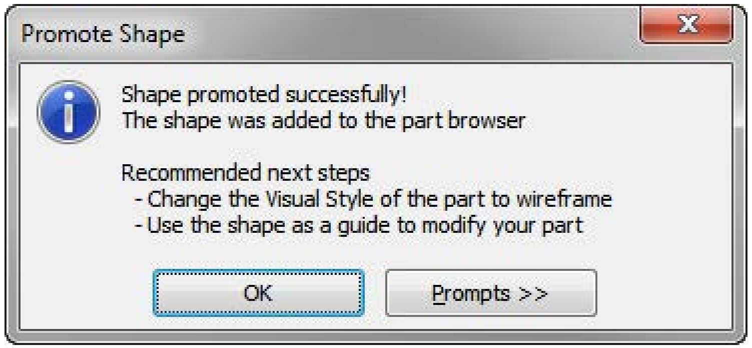
Next you will promote this part into your part file. Select ‘Promote Shape’ from the ‘Export’ panel.

30.



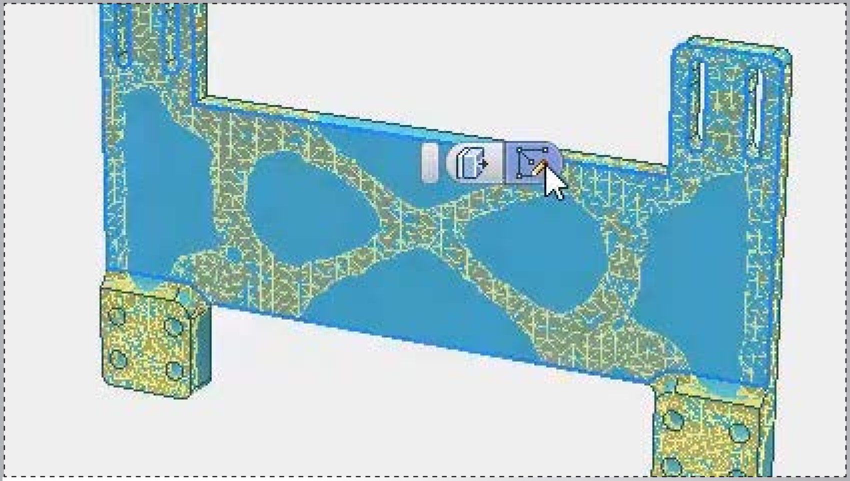
The ‘Promote Shape’ dialog appears, select ‘Current Part File’ and select ‘OK’.

31.



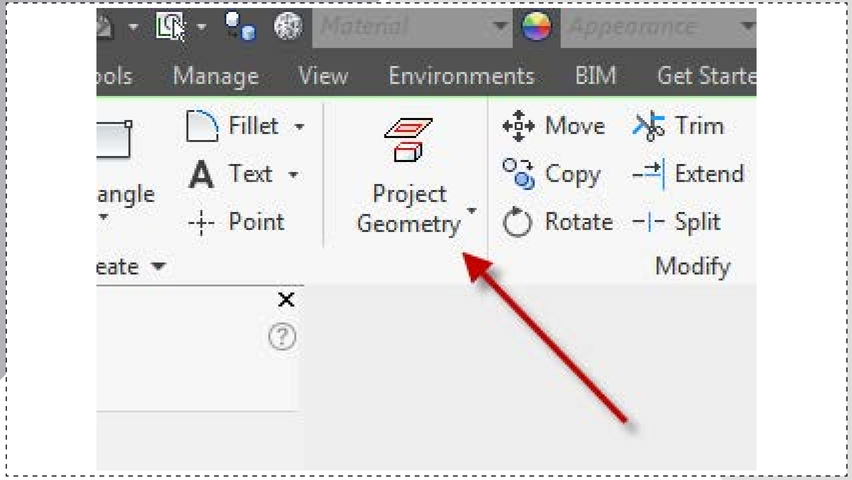
You will receive a prompt explaining to you that the shape has been successfully promoted select ‘OK’.

32.



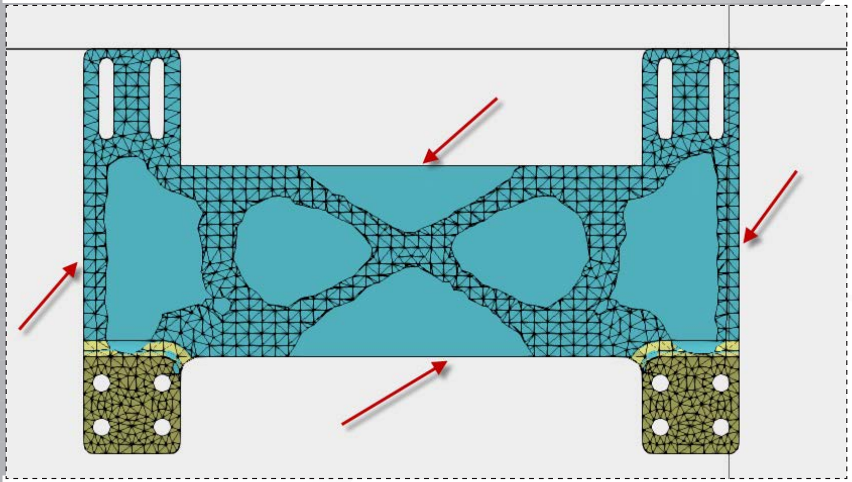
Next go to the 'Sketch tab' and start a new 2D sketch on the front face of the part.

33.



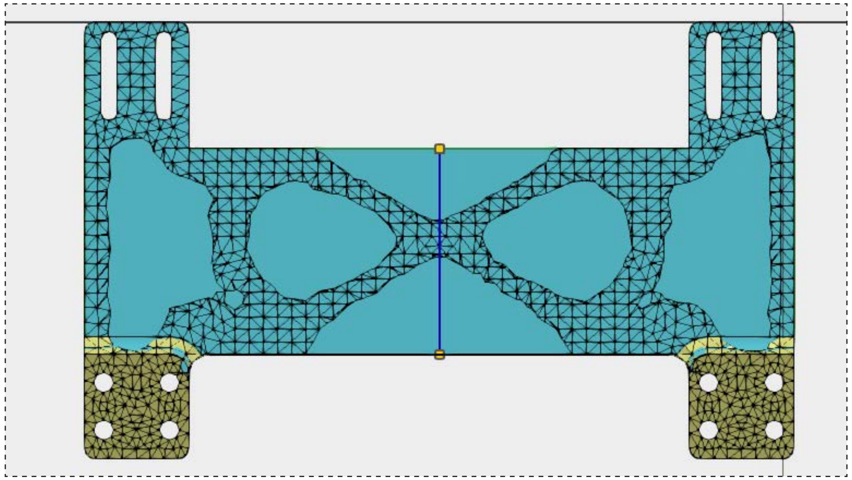
In the 'Sketch' tab select 'Project Geometry' and project the edges existing part into the sketch.

34.



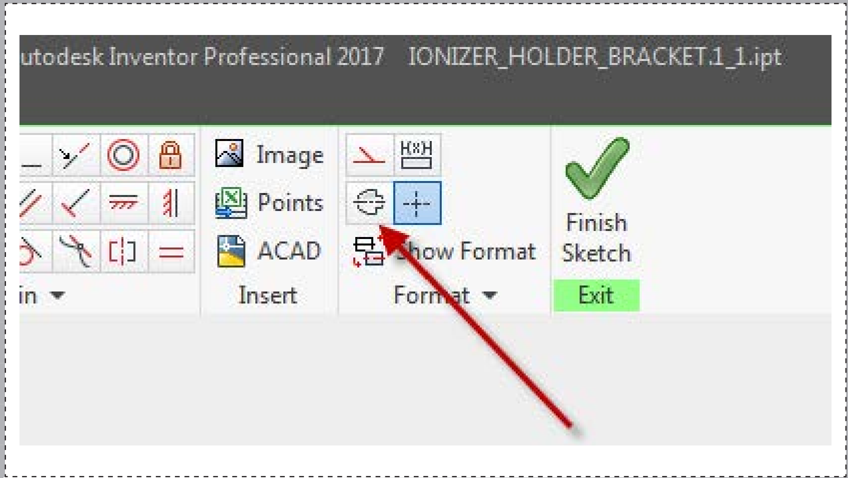
Select 4 edges.

35.



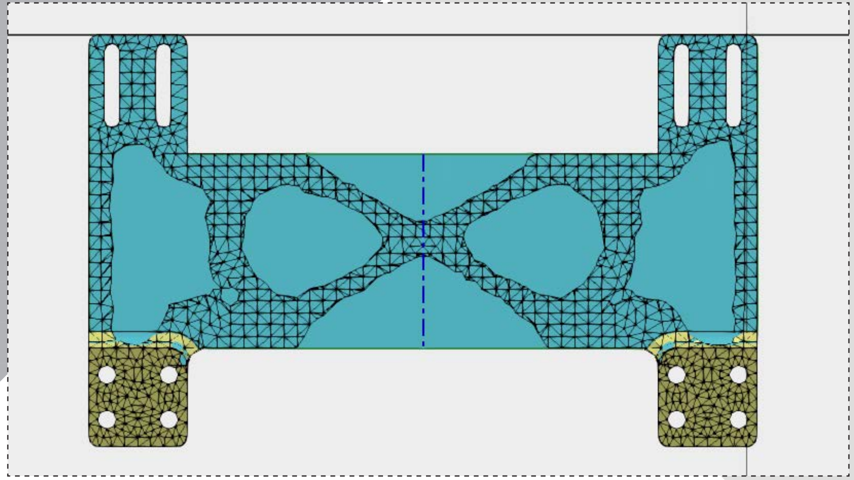
You will next select the 'Line' command and place a line between the midpoint of the upper projected line and the midpoint of the lower projected line.

36.



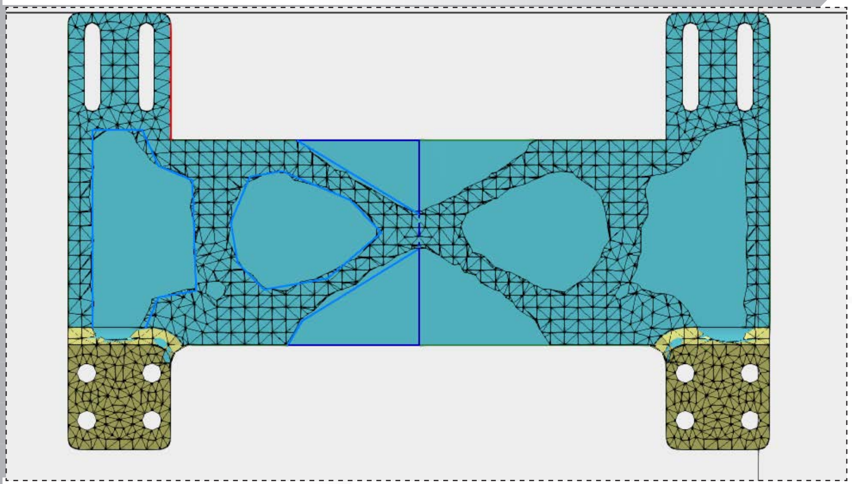
Select the line and in the 'Format' panel select 'Centerline' and change the line to a center line, as shown in the images below.

37.



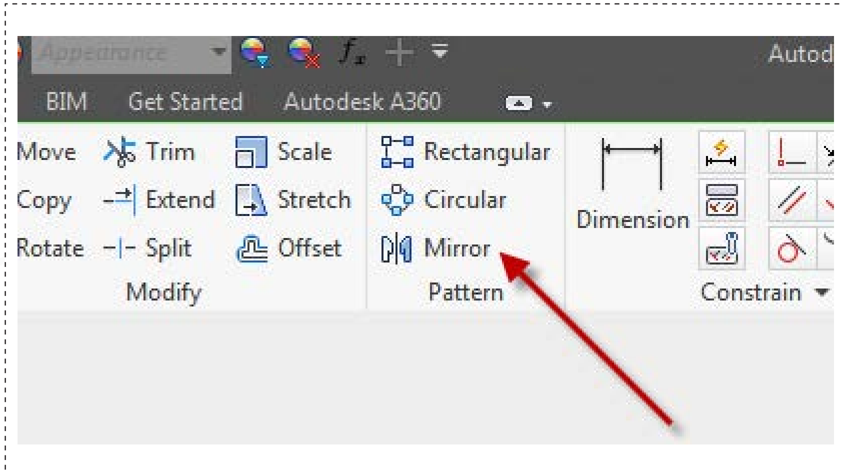
Line changes to centerline.

38.



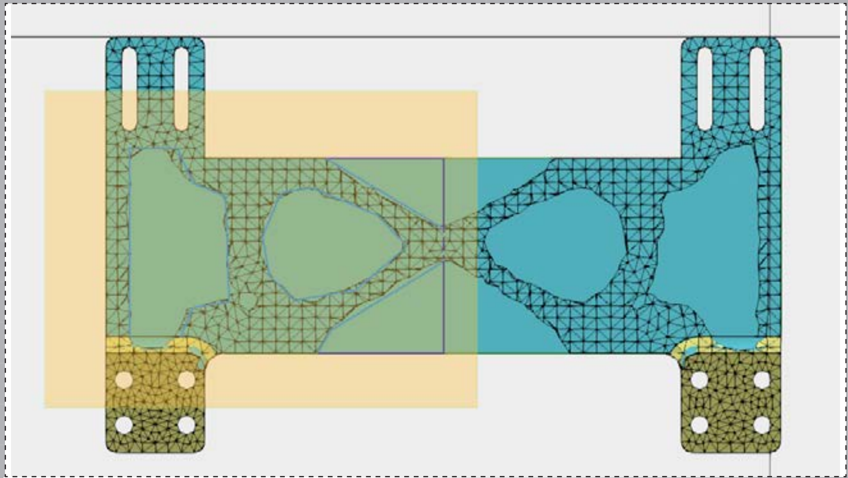
Using the mesh as a guide use the 'Line' tool to draw in line segments to cut away material from the existing design. Begin by using the 'Line 'tool in the 'Create' panel and follow the images above.

39.



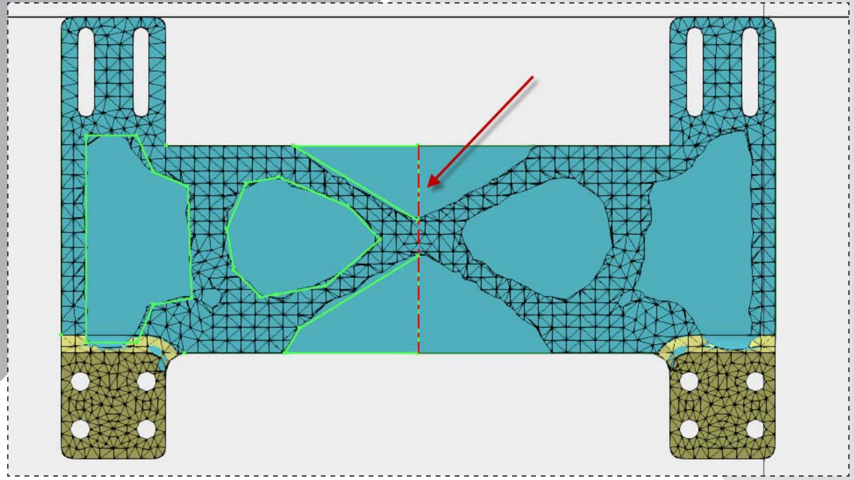
Next select the 'Mirror' command in the 'Pattern' panel.

40.



The ‘Mirror’ dialog box appears, click on ‘Select’ and expand a box over the line segment drawing you just created.

41.

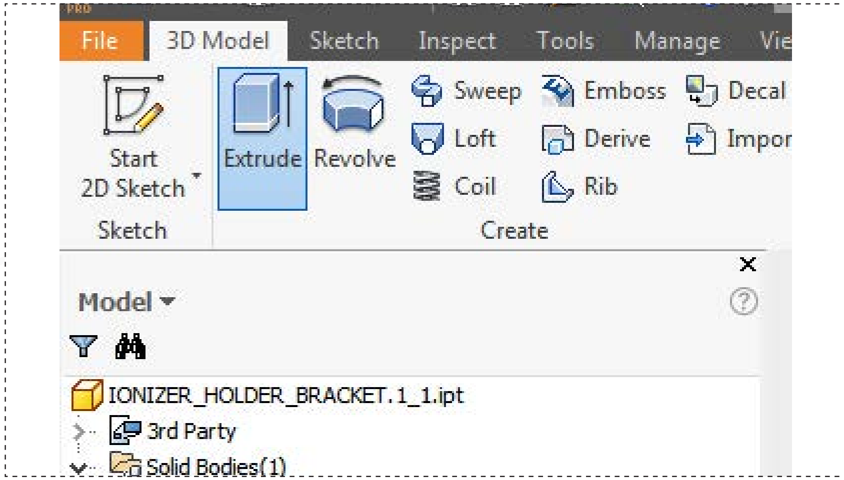


Next select the centerline created earlier and click ‘Mirror Line’ in the ‘Mirror’ dialog box. Then select ‘Apply’ to mirror the lines you just created.

42.

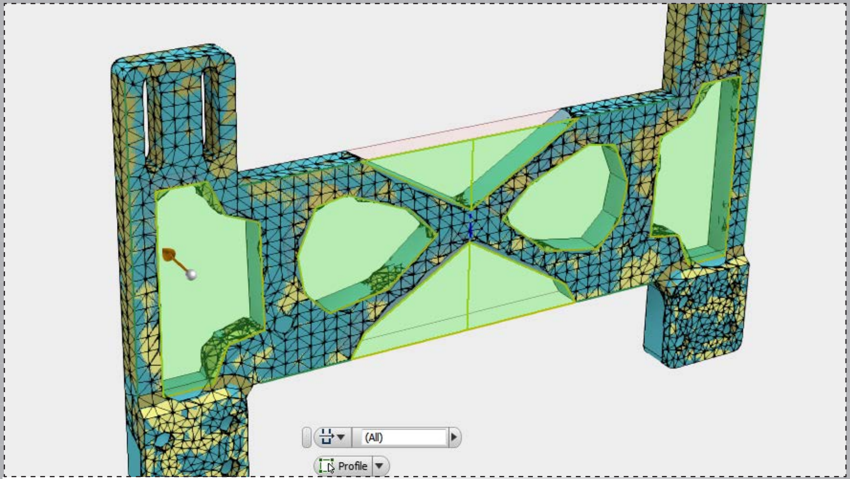
Next select ‘Finish Sketch’ in the ‘Exit’ panel to exit out of the sketch.

43.



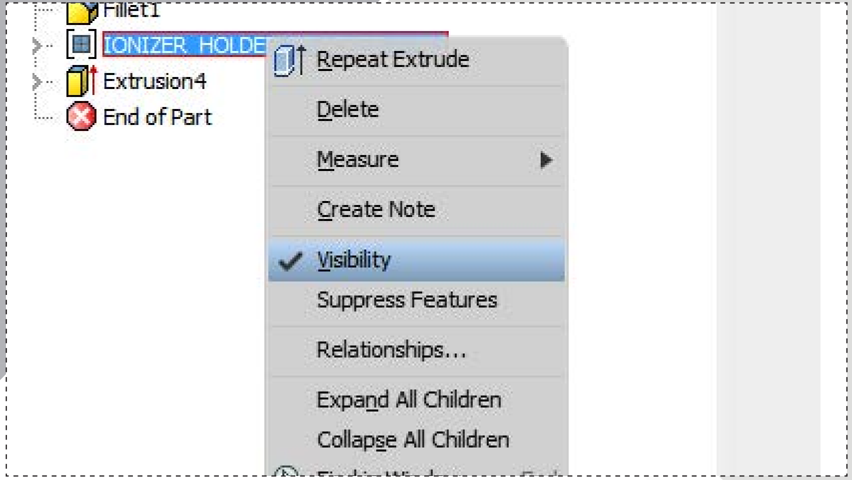
Next click the ‘3D Model’ tab and select the ‘Extrude’ command in the ‘Create’ panel.

44.



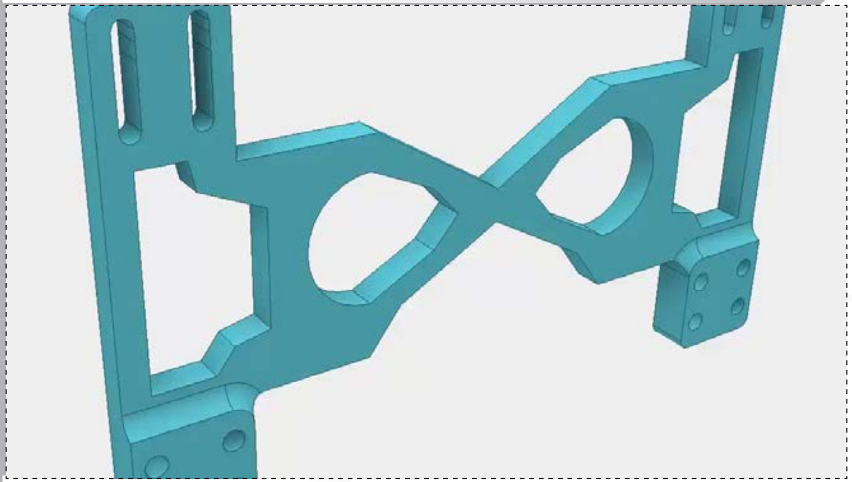
Then select all the closed profiles on the part as shown in the image below. Select 'Cut' from the drop down menu. Then select 'Through All'. Then select 'OK'.

45.



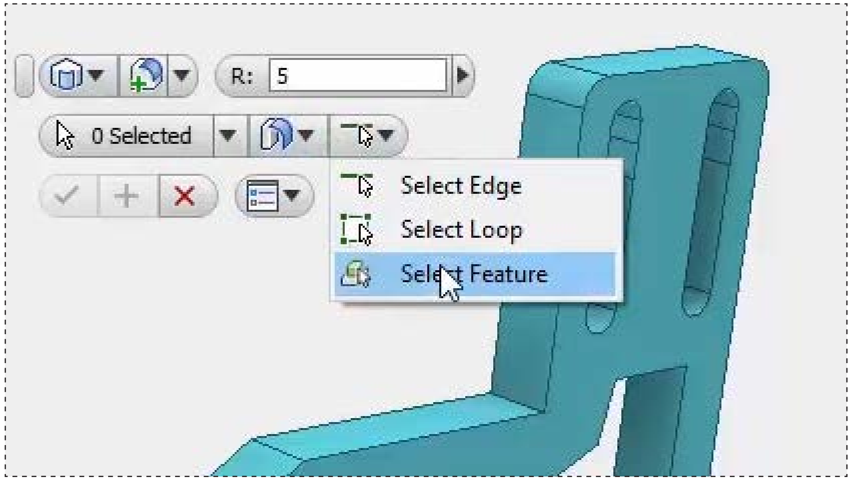
Go to the browser click on the mesh item and then right click to select 'Visibility' to turn the mesh off.

46.



Your model should look similar to the image above.

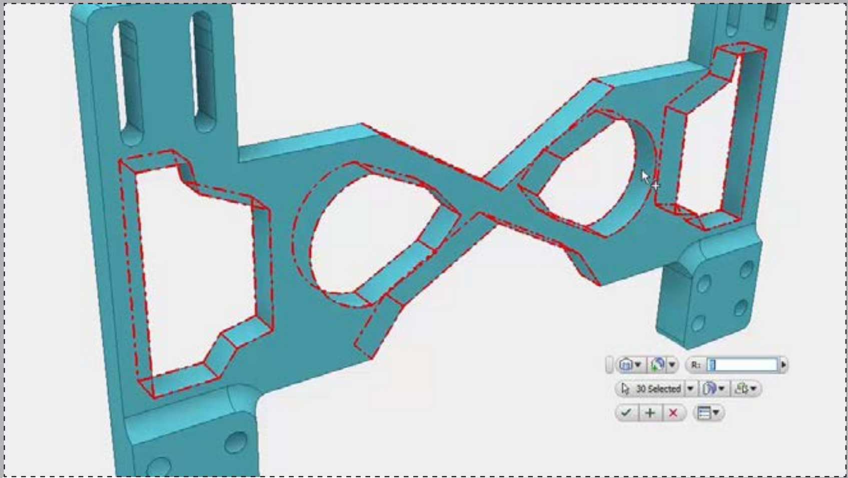
47.



Next select the 'Fillet' button in the 'Modify' panel. The 'Fillet' toolbar appears, click 'Select Feature'.

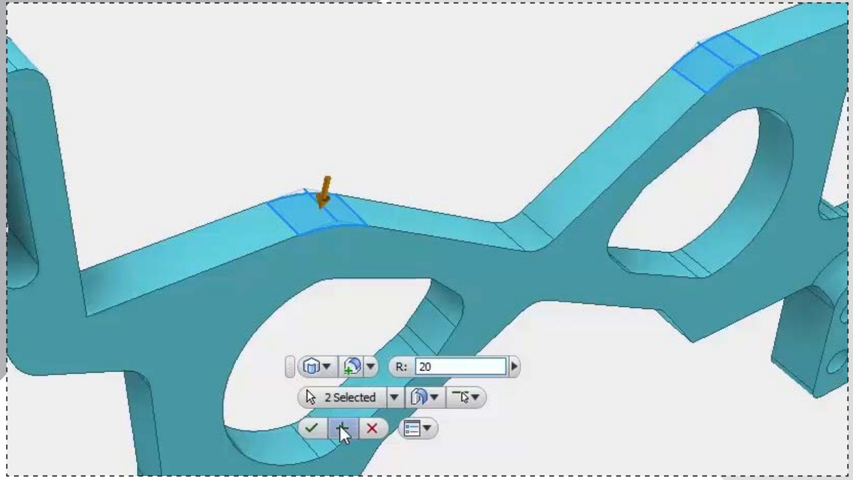
PART 2: SHAPE GENERATOR

48.



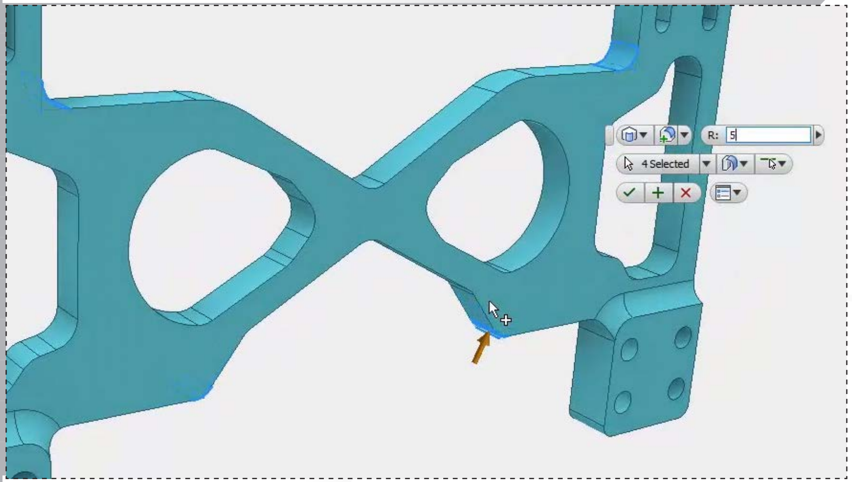
Then select the extrude feature previously created and add a 5 mm radius to any corner in that feature. Select 'OK'.

49.



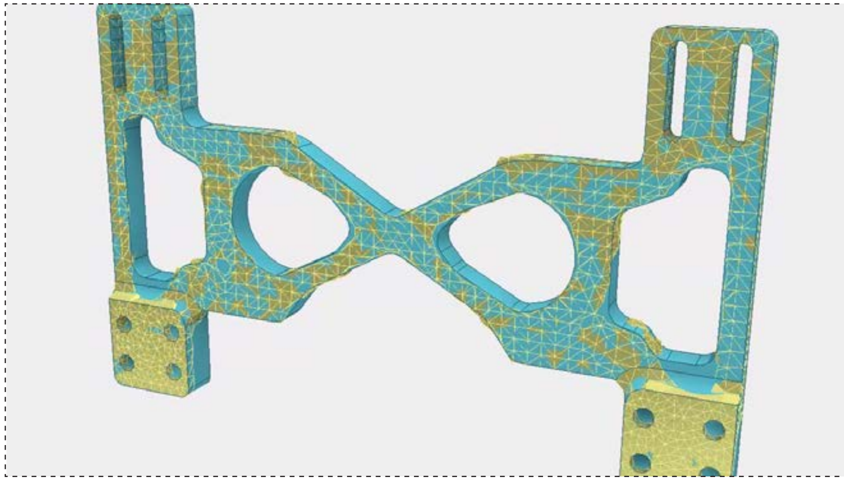
Create two more fillets on the upper corners of the part, make them 20 mm radius, as shown in the image above.

50.



Next add another group of fillets, two at the top and two at the lower portion of the part as shown in the images above. Place a value of 5 mm radius for all four fillets. Select 'OK' to create the fillets.

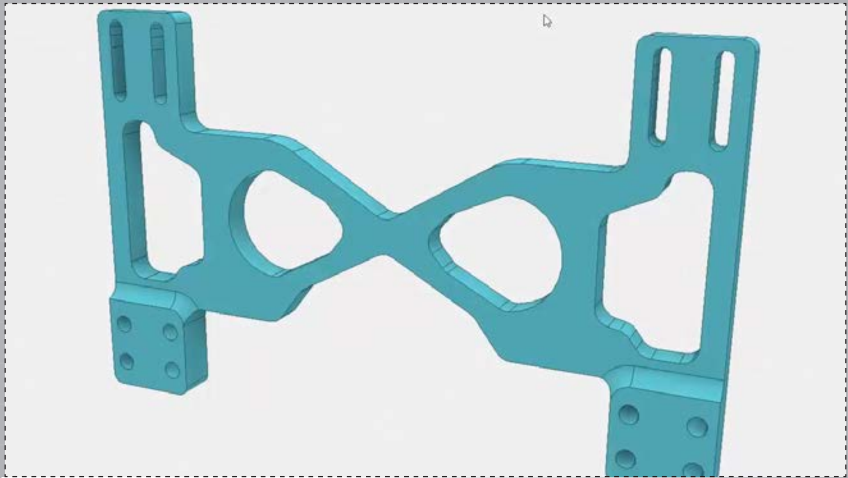
51.



Finally, go to the browser, click on the mesh item and then right click to select 'Visibility' to turn the mesh on. Make sure to save the part file before moving on to the next project.

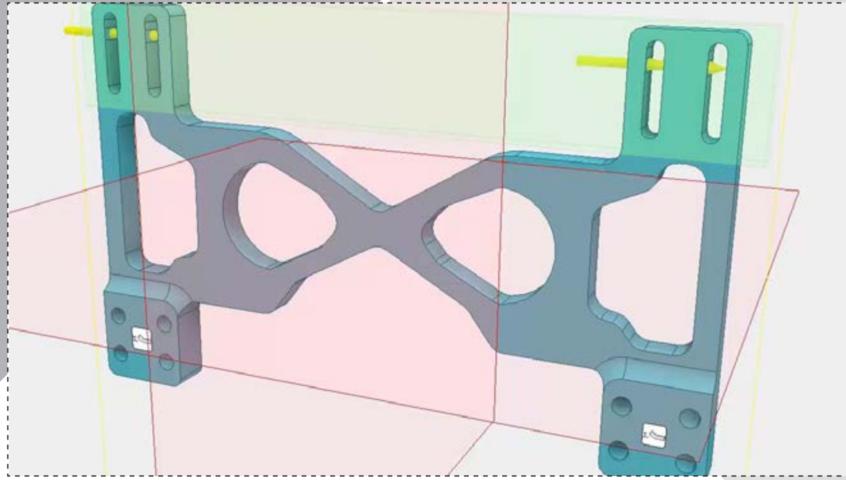
PART 3: SHAPE GENERATOR

52.



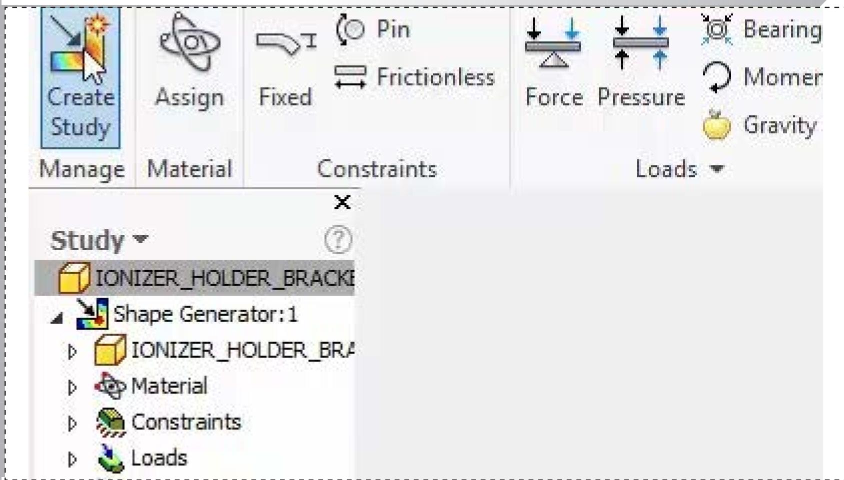
Open the part file from the previous project.

53.



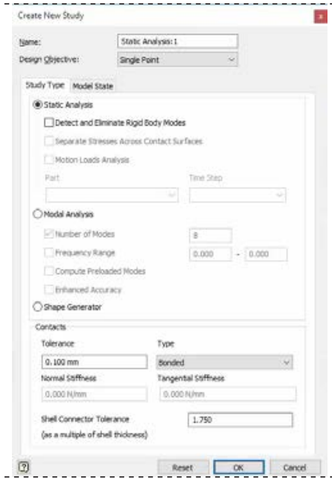
Select the ‘Stress Analysis ‘tool in the ‘Simulation’ panel under the ‘3D Model tab’.

54.



Next select the ‘Create Study’ in the ‘Manage’ panel under the ‘Analysis tab’.

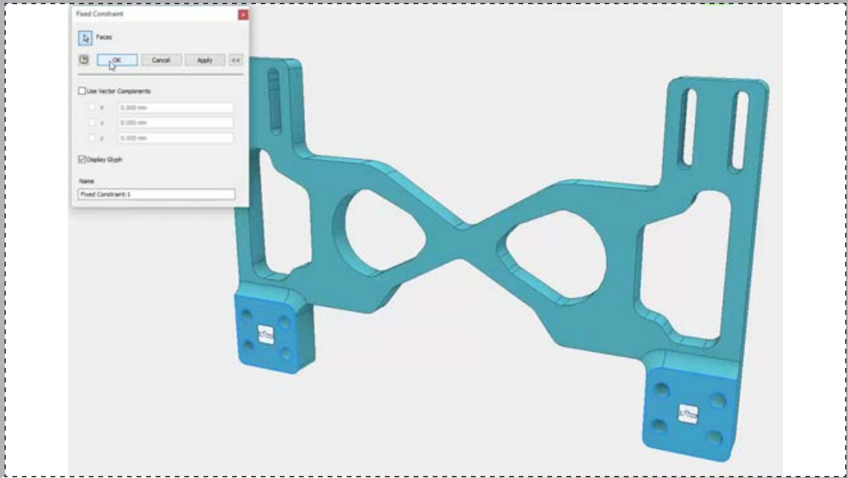
55.



The ‘Create New Study’ dialog box apppears, select ‘Static Analysis’ under the ‘Study Type’ tab. Select ‘OK’.

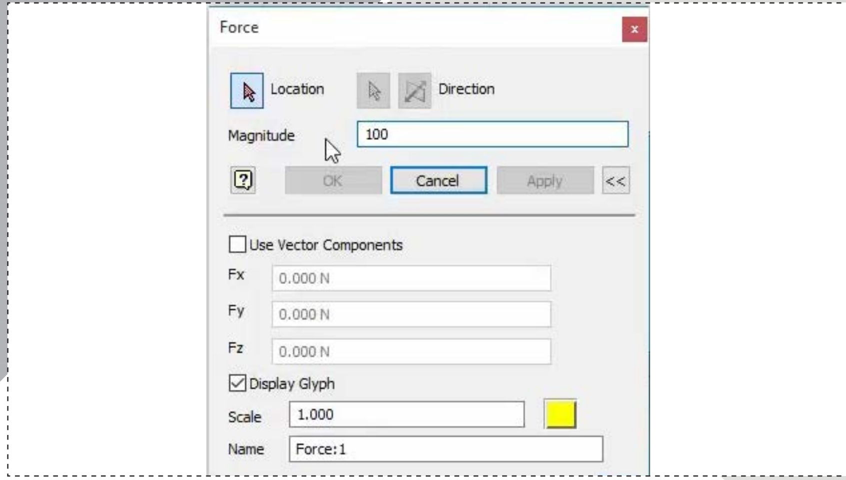
PART 3: SHAPE GENERATOR

56.



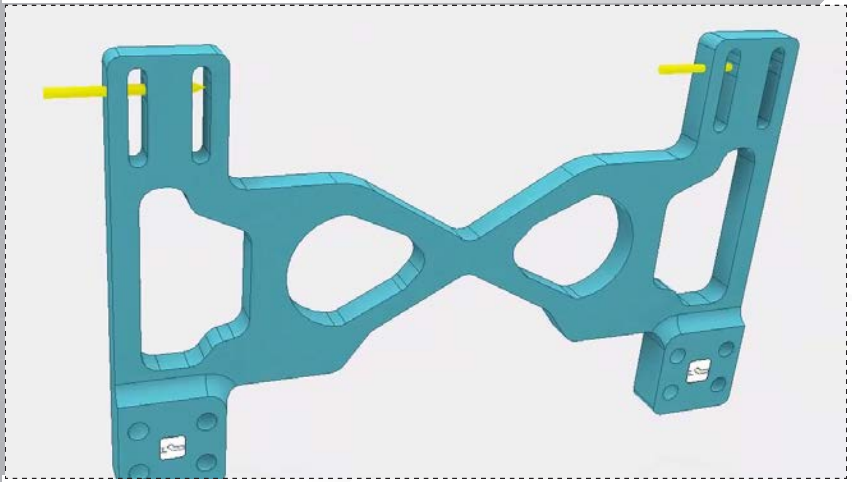
Next set your constraints by going to the 'Constraints' panel and select 'Fixed'. The 'Fixed Constraint' dialog appears. Set the fixed constraint to the two bottom mounting faces on the part.

57.



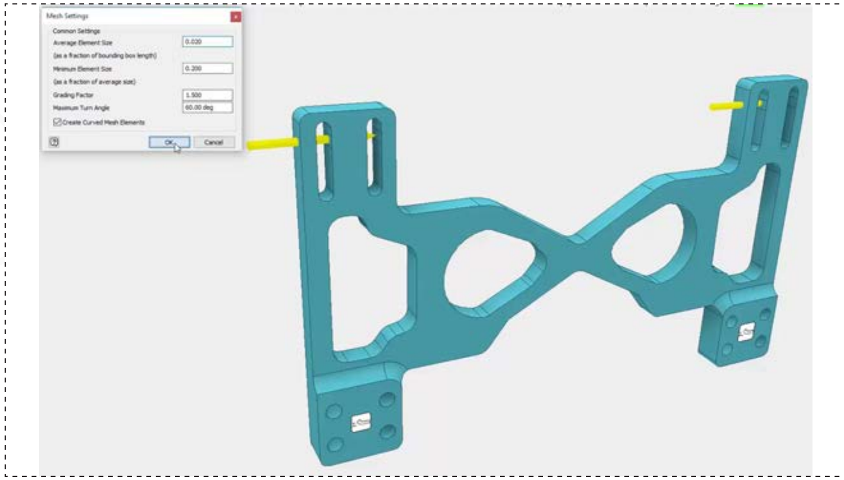
Next you will next need to apply a force. Go to the 'Force' tool in the 'Loads' panel and in the 'Force' dialog box place a value of 100 N in the 'Magnitude' box, as shown in the images above.

58.



Next select the inside face edge on all for all four mounting slots as shown in the image above. Select 'OK'.

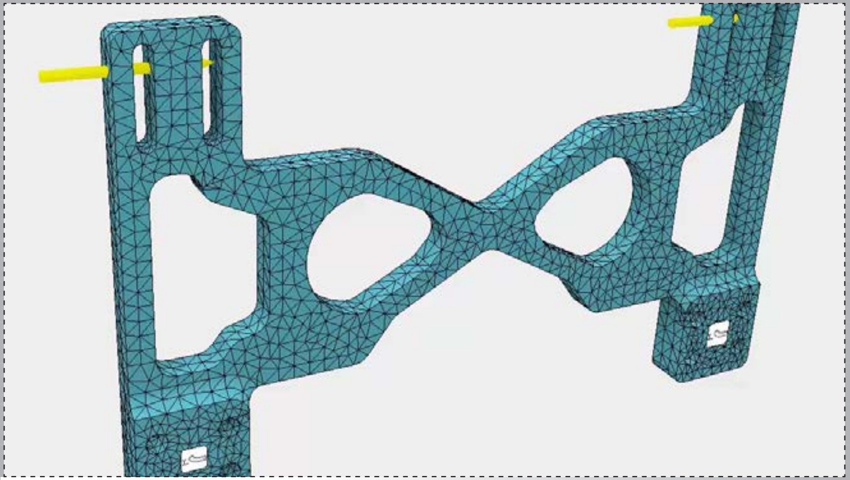
59.



Next select 'Mesh Settings' button in the 'Mesh' panel. The 'Mesh Settings' dialog appear. Under 'Common Settings' in the 'Average Element Size' box set the value to 0.02 and select 'OK".

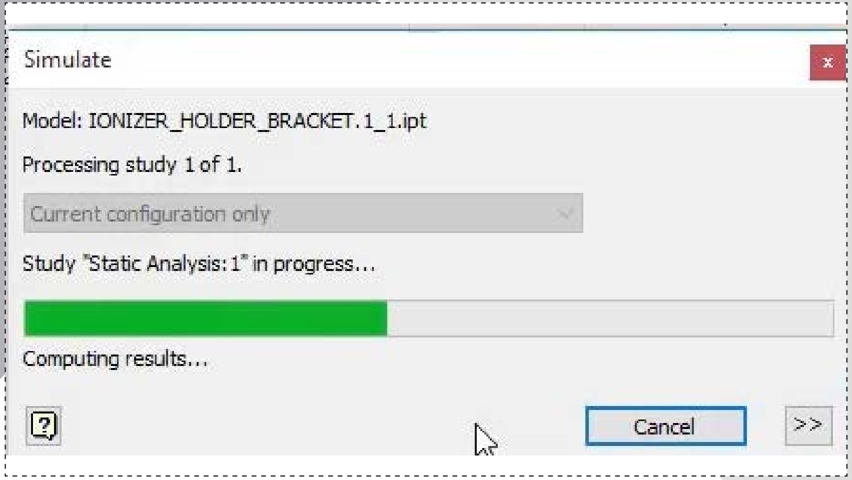
PART 3: SHAPE GENERATOR

60.



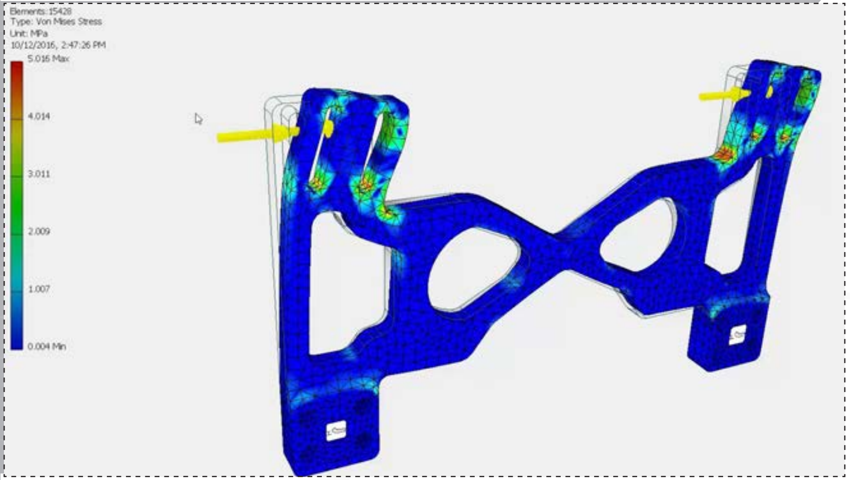
Select ‘Mesh View’ in the ‘Mesh’ panel. The part will begin to calculate. The final mesh appears after it’s calculation, as shown in the image above.

61.



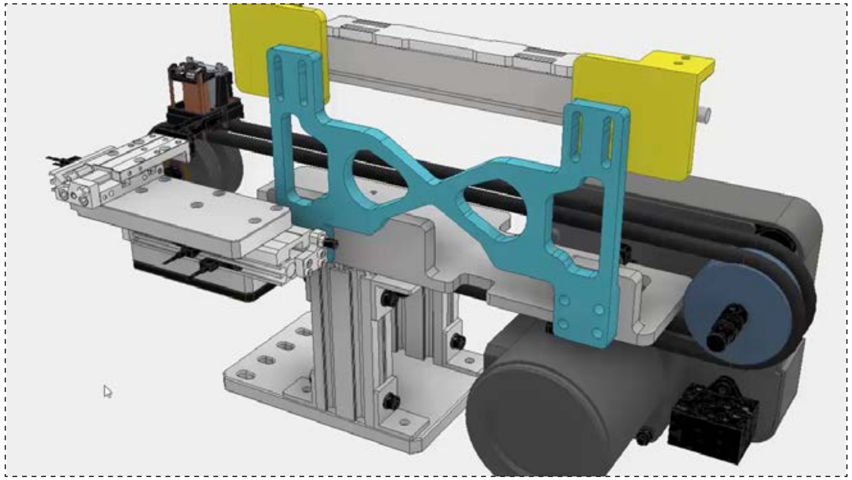
Finally select ‘Simulate’ in the ‘Solve’ panel to run the study. The ‘Simulate’ dialog box appears, select ‘Run’.

62.



Notice in the analysis that based on the shape generation that there is little-to-no stress at all.

63.



Next exit the analysis by selecting ‘Finish Analysis’ in the ‘Exit’ panel. Next go back to the ***Conveyer Assembly*** and notice that you now have a lighter part that is as stiff as the original intended design. Make sure to save all your files before closing.