

A 3D rendered sheet metal bracket, primarily orange and yellow, is shown against a light gray grid background. The bracket has a complex shape with a curved top and a vertical section. The background features a faint grid and a large, light gray circular arc.

AUTODESK[®] INVENTOR[®]

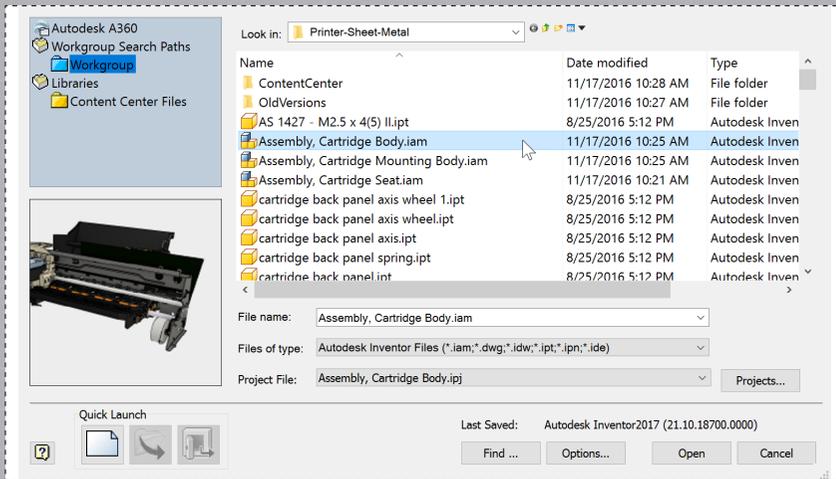
Trial Projects

Sheet Metal Design

Design a sheet metal bracket

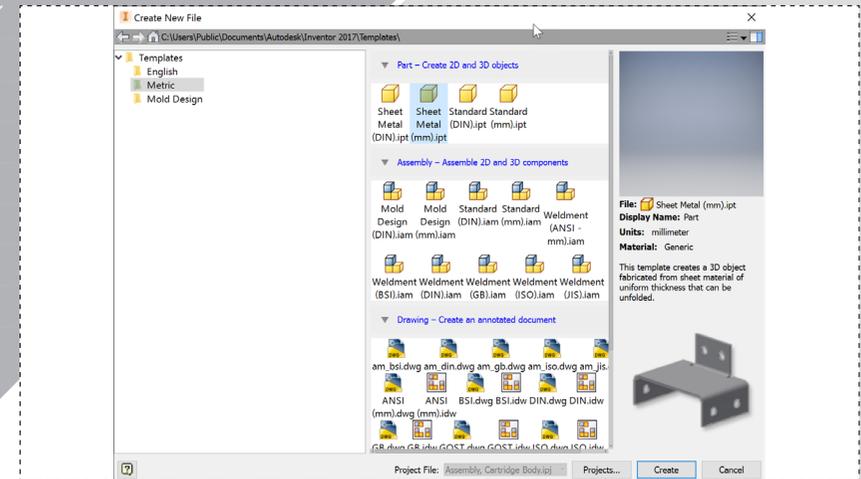
PART 1: CREATE BASE GEOMETRY

1.



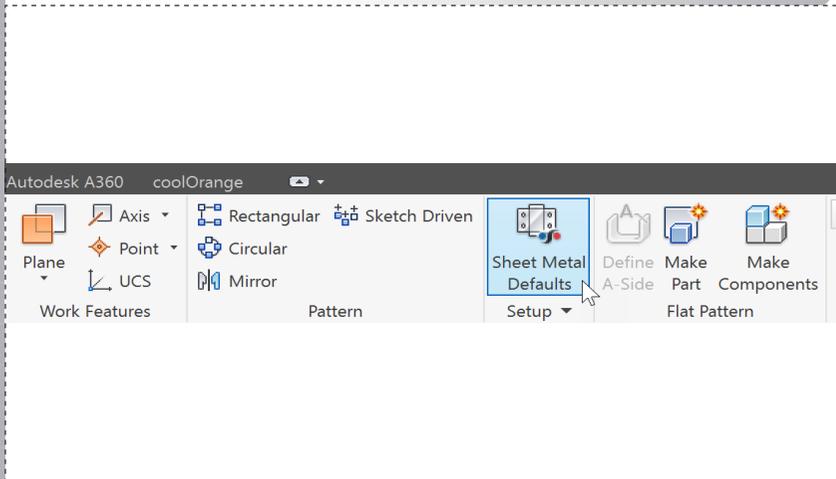
In Inventor, click the 'Projects' icon in the ribbon. Navigate to where you saved the files and select **Assembly, Cartridge Body.ipj**. Then open the file **Assembly, Cartridge Body.iam**.

2.



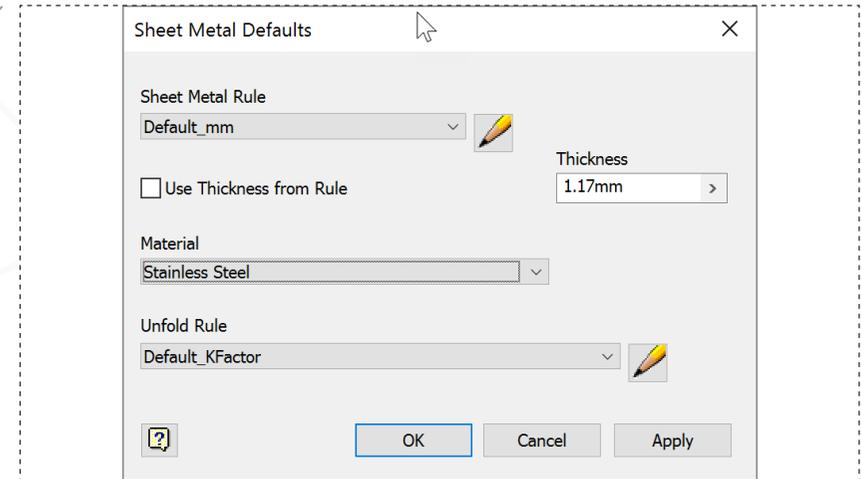
Select 'File' and choose 'New' from the ribbon menu. In the dialog, select the 'Metric' folder and choose 'Sheet Metal (mm).ipt' as the template.

3.



In the ribbon, select 'Sheet Metal Defaults'.

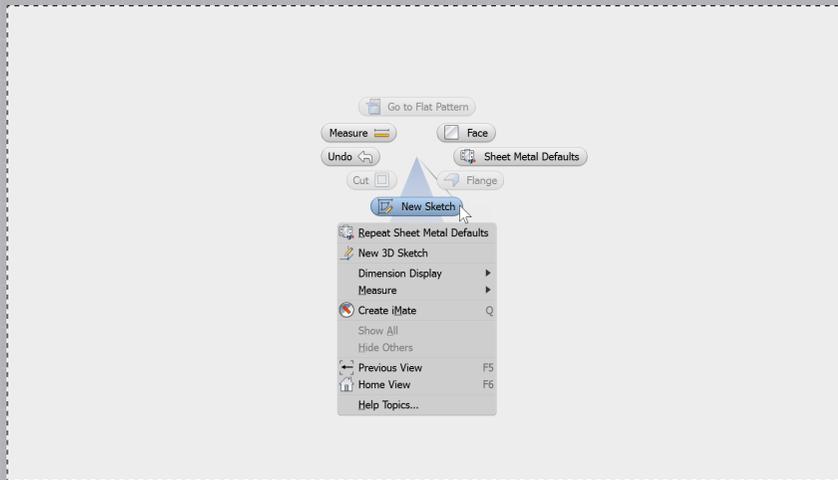
4.



In the dialog, uncheck 'Use Thickness from Rule' and enter 1.17mm as the thickness value. Change the material to 'Stainless Steel.' Select 'OK' to set the defaults.

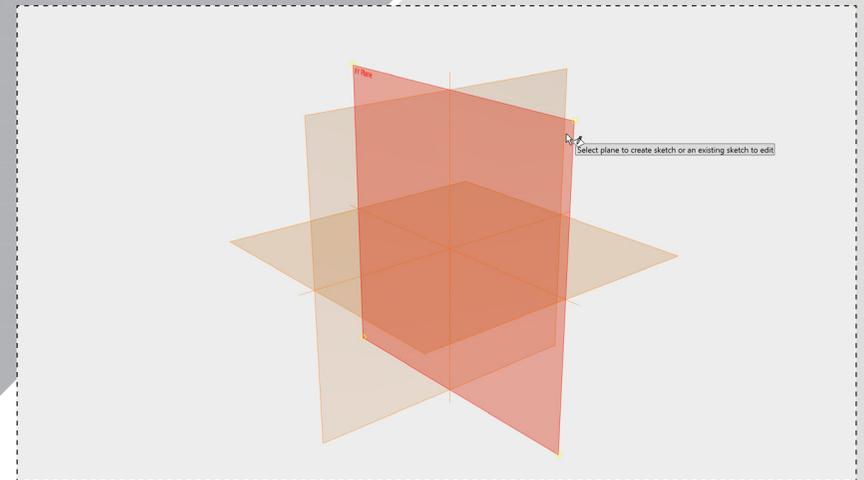
PART 1: CREATE BASE GEOMETRY

5.



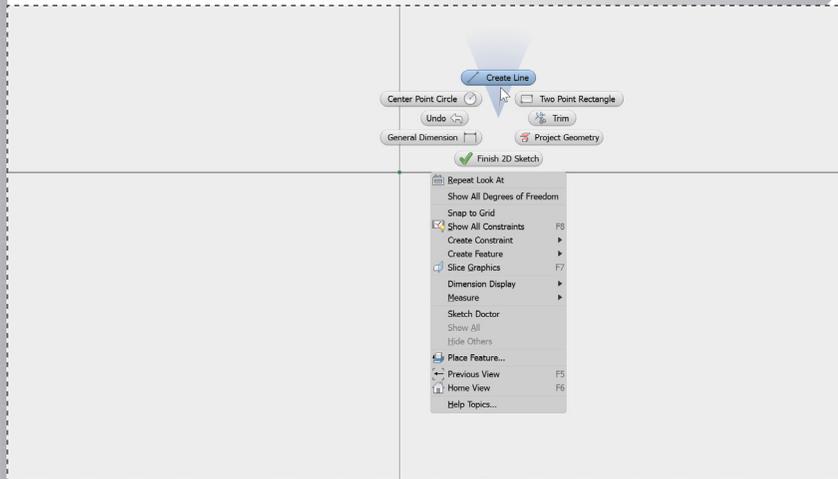
Right click in the model window to open the marking menu. Select 'New Sketch'.

6.



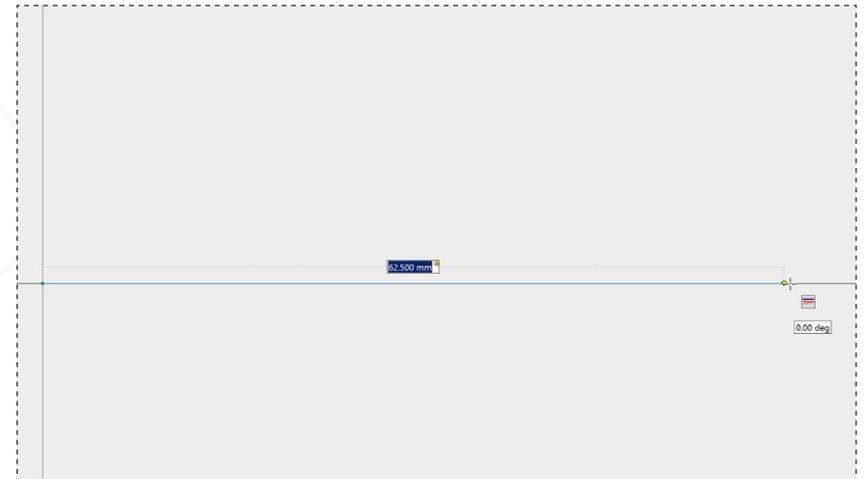
Select the 'X-Y' plane as the sketch plane.

7.



In the sketch view, right click to open the marking menu. Select 'Create Line'.

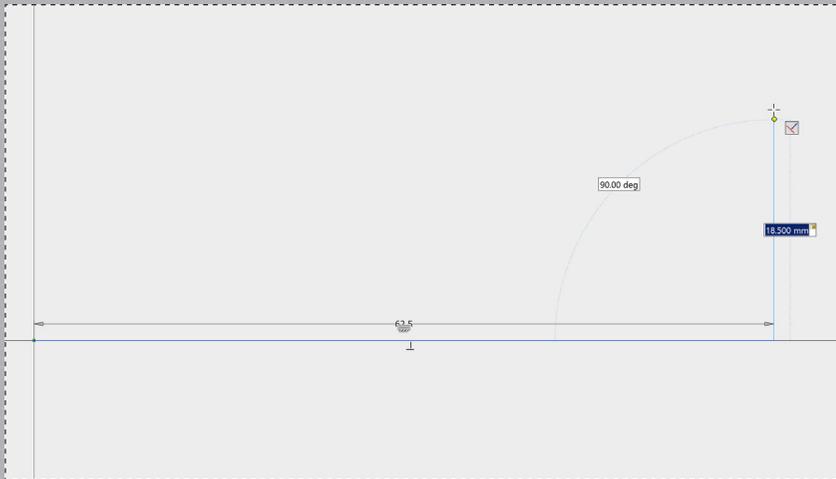
8.



Click the origin point to start the line and then move your cursor to the right. Before you click the end point, type 62.5mm into the dynamic dimensioning box.

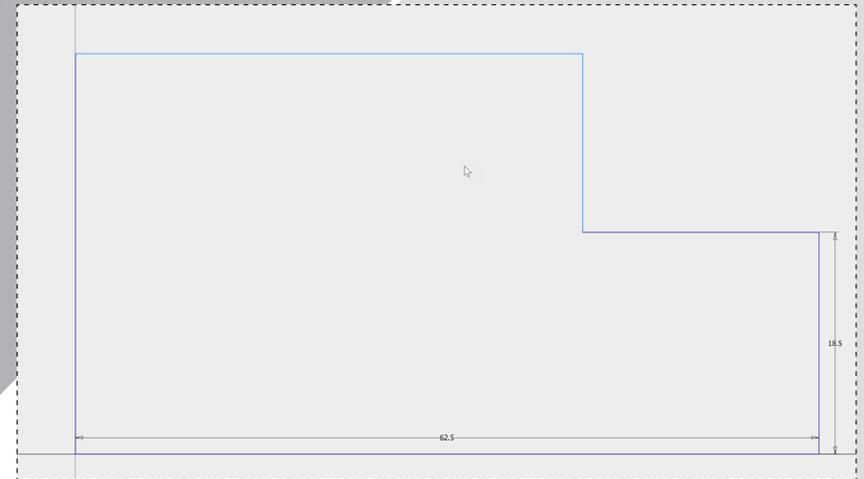
PART 1: CREATE BASE GEOMETRY

9.



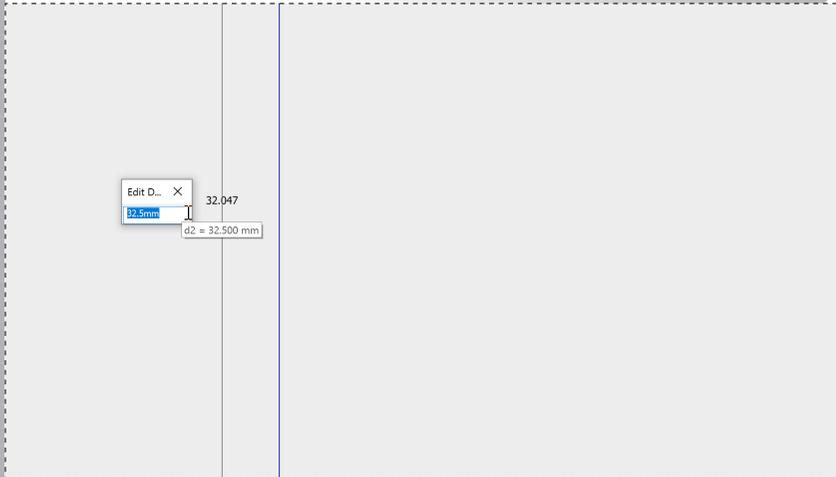
Click to set the end point of the line then move your cursor up. Type 18.5mm into the dynamic dimensioning box. Click to create the vertical line.

10.



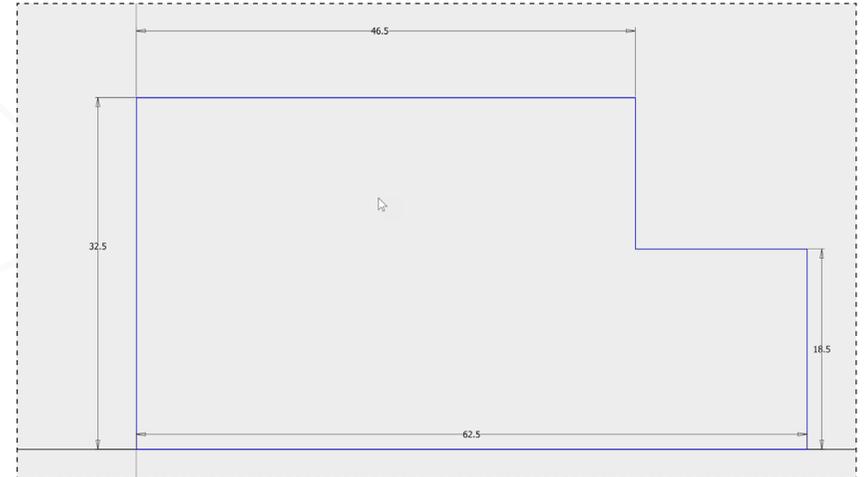
Continue to draw lines (without adding dimensions) to create the rough shape as shown. Ending on the origin point will complete the line segment, then right click and select 'OK'.

11.



Right click and select 'General Dimension'. Select the left vertical line segment and click to place the dimension. Type in 32.5mm and touch 'Enter' on your keyboard to finish.

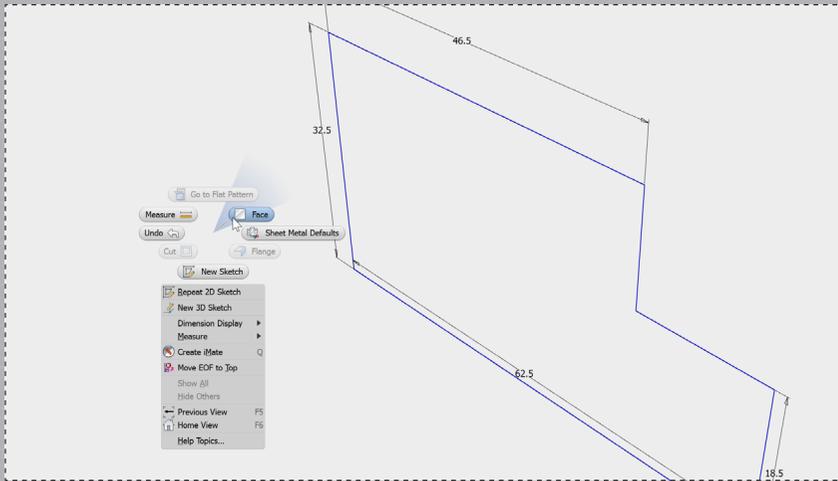
12.



Repeat the process to add a 46.5mm dimension to the top horizontal line segment. The sketch is now fully constrained so click the green check mark in the ribbon to 'Finish Sketch'.

PART 1: CREATE BASE GEOMETRY

13.



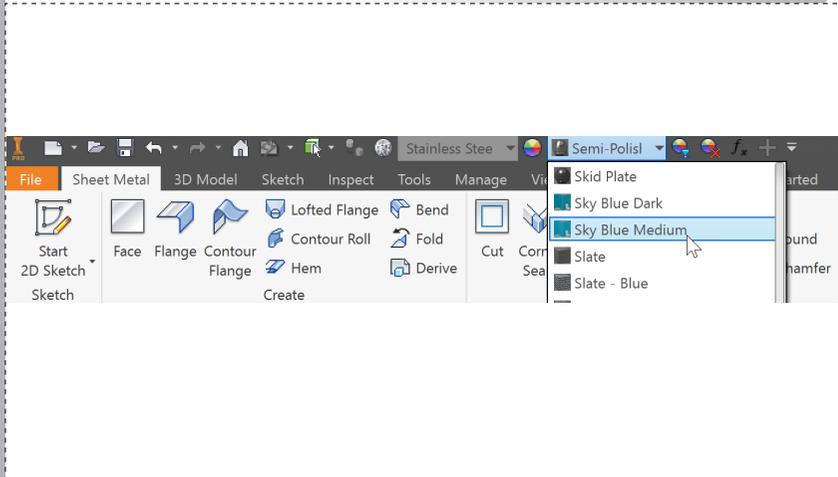
Right click and select the 'Face' command from the marking menu.

14.



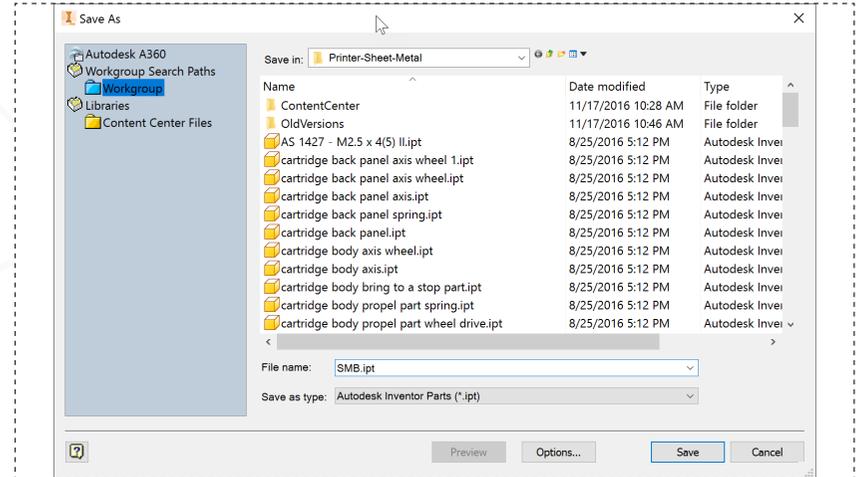
The outline will auto-select since it is the only profile available. Click 'OK' with the default settings to create the base 3D geometry.

15.



In the top toolbar, click the 'Appearance' drop down menu. Select 'Sky Blue Medium' to help distinguish this part.
Note: If this color is not available, change the library at the bottom of the drop-down menu to 'Autodesk Appearance Library'.

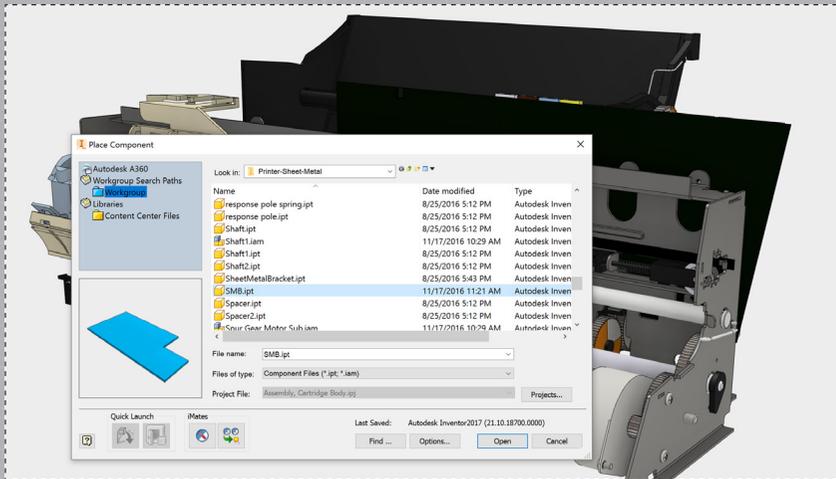
16.



Save the part as **SMB.ipt** and close the part window.

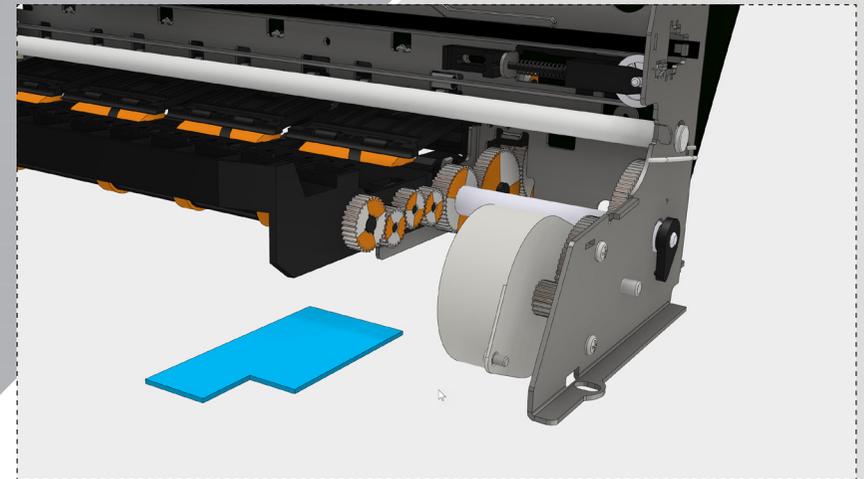
PART 1: CREATE BASE GEOMETRY

17.



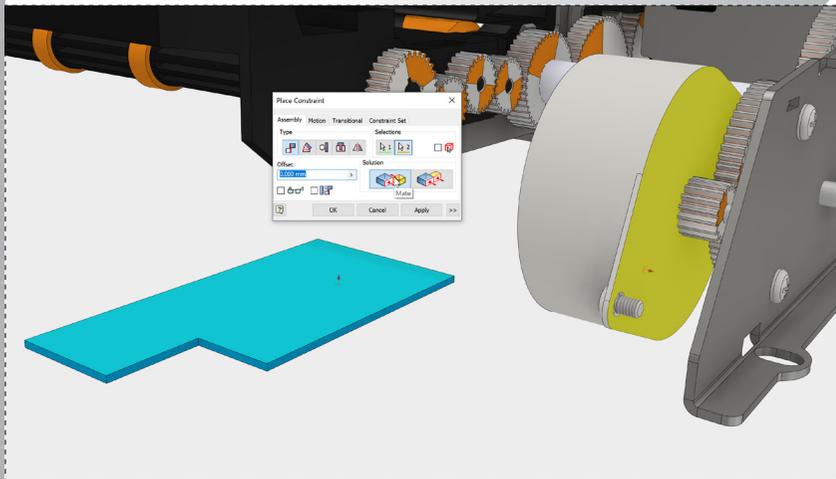
Inside the **Assembly, Cartridge Body.iam** assembly, right click and select 'Place Component'. Select the **SMB.ipt** file you just saved and select 'Open'.

18.



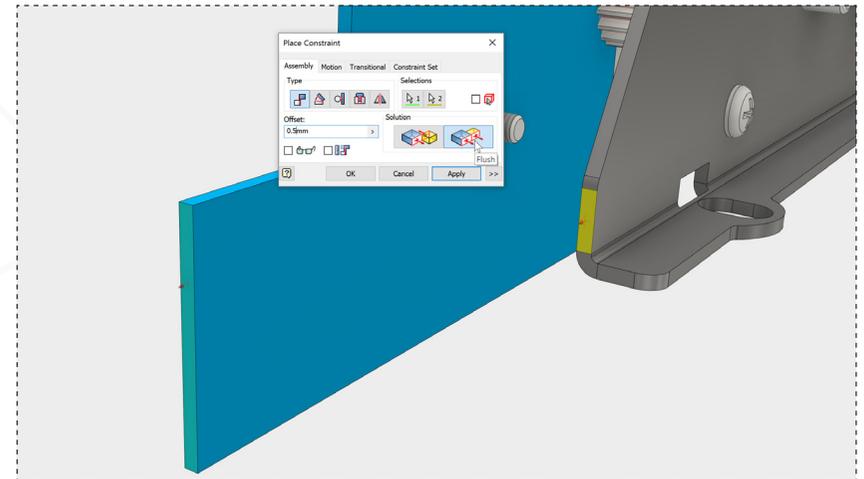
Click to place the part roughly as shown. Right click and select 'OK' to finish placement.

19.



Right click and select 'Constraint' from the marking menu. Select the top face of the bracket and the inside face of the motor as shown. Hit 'Apply' to create the first constraint.

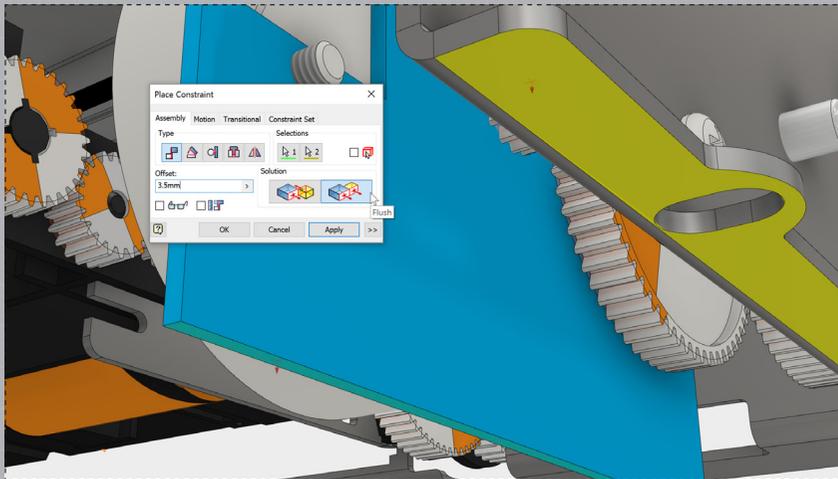
20.



Select the two faces of the parts as shown. Type 0.5mm as the offset distance and in the 'Solution' section of the dialog select 'Flush'. Hit 'Apply' to create the constraint.

PART 1: CREATE BASE GEOMETRY

21.

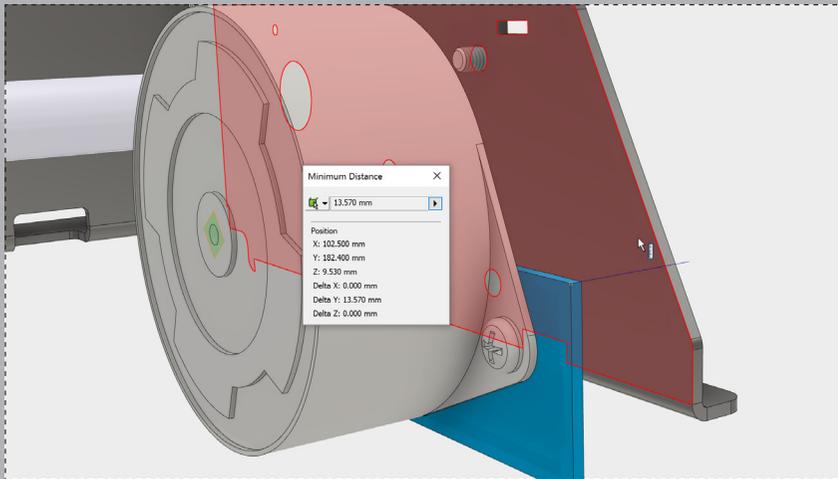


Rotate the model to view the bottom edge of the part. Select the two faces shown and use enter 3.5mm for the offset, continuing to use the 'Flush' constraint. Hit 'OK'.

Note: You may need to enter -3.5mm as the offset depending on the order you click the faces. The bracket should be higher than the yellow face shown.

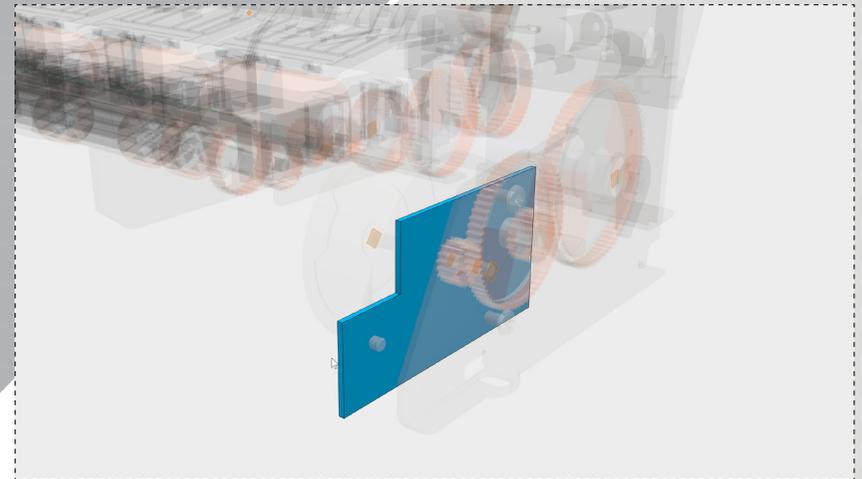
PART 2: ADDING FLANGES

1.



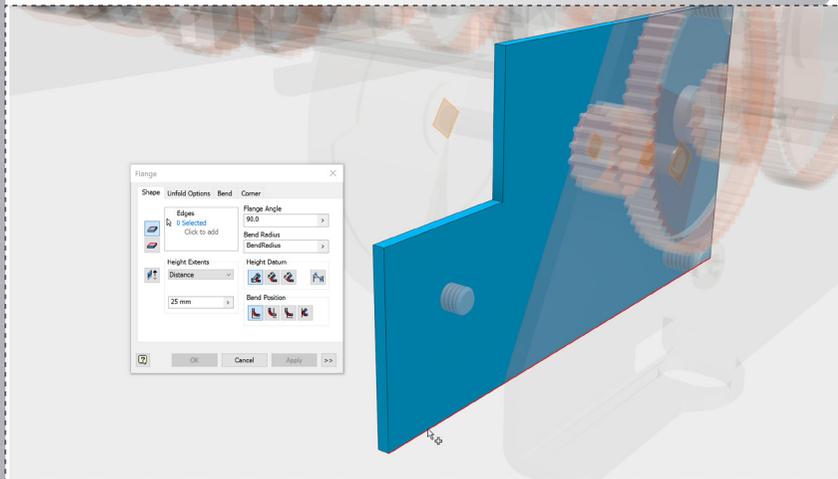
Right click and select 'Measure' from the marking menu. To determine the length of the flanges, measure from the outside edge of the bracket to inside edge of the part as shown.

2.



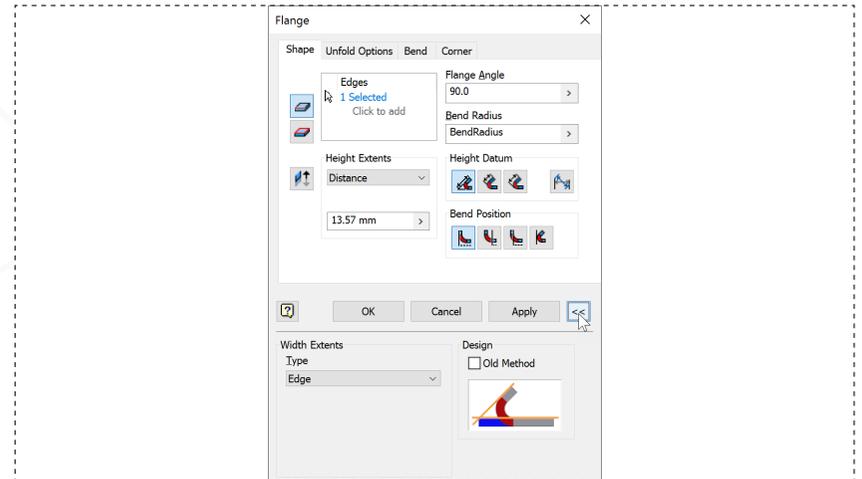
Double click the bracket to make the model editable.

3.



Right click and select 'Flange' from the marking menu. Select the edge as shown.

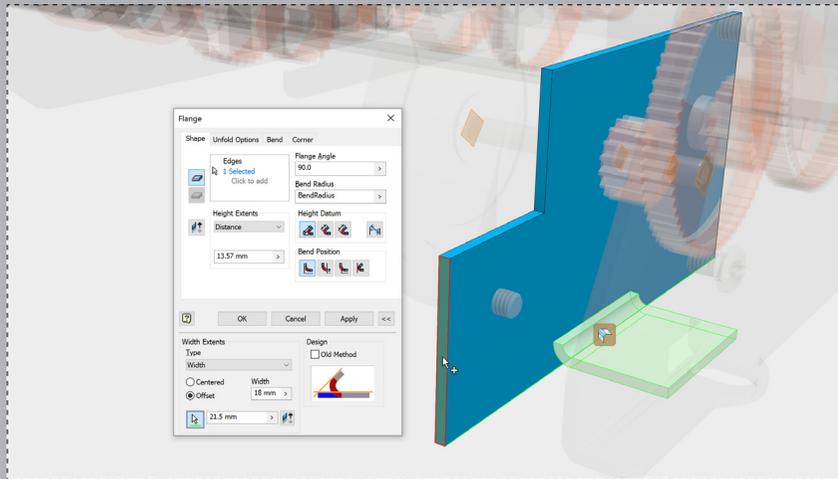
4.



Enter the measured distance, 13.57mm, in the 'Height Extents' field and then expand the dialog by clicking the double arrows next to the 'Apply' button.

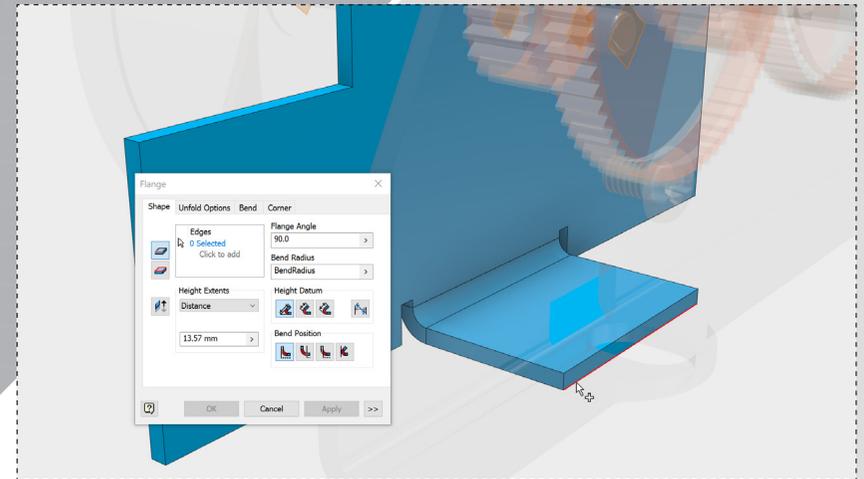
PART 2: ADDING FLANGES

5.



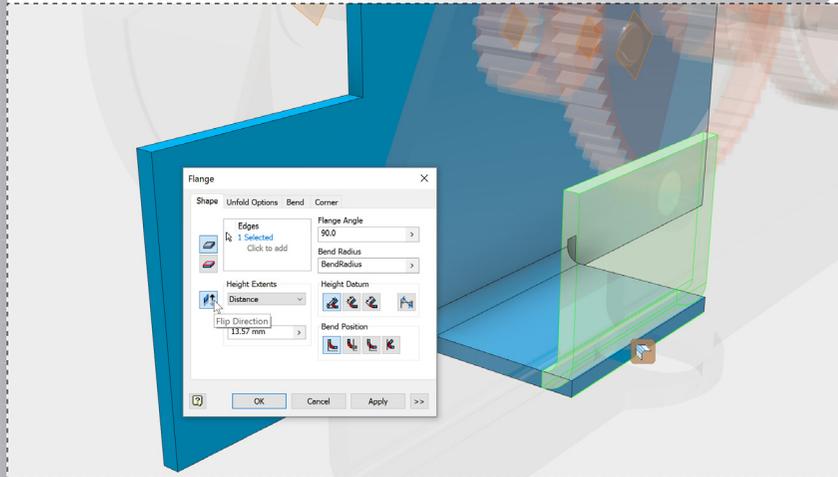
Change the 'Width Extents' type to 'Width' and enter 18mm. Select the 'Offset' button, enter 21.5mm in the field, and then click the arrow to select the face of the part as shown.

6.



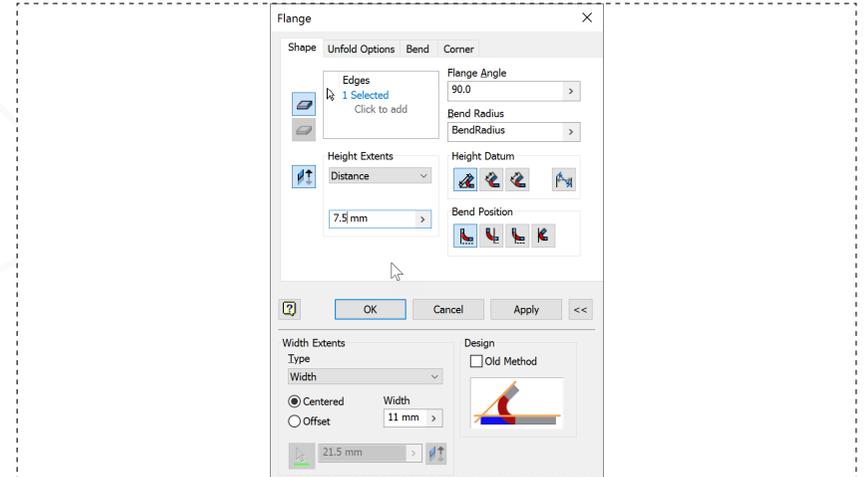
Click 'Apply' on the dialog to create the first flange. To start the second flange, select the edge as shown.

7.



Use the 'Flip Direction' button in the dialog (if necessary) to make sure the new flange is in the correct orientation.

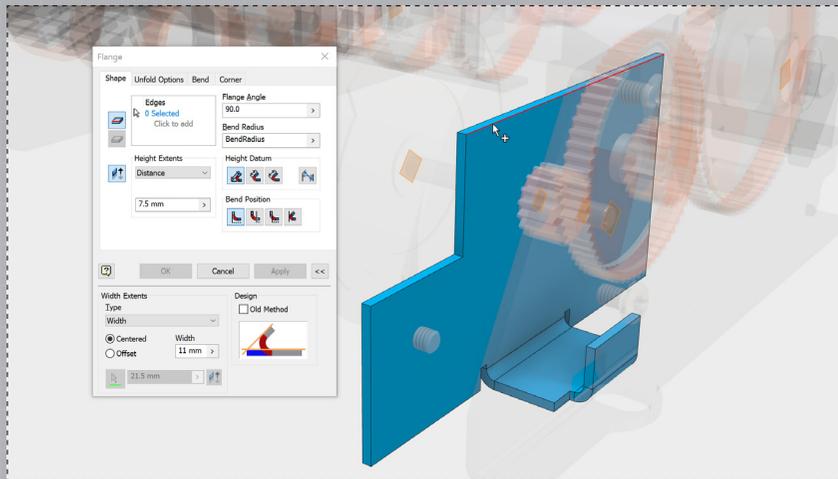
8.



Configure the flange using the values and options for the 'Height Extents' and 'Width Extents' as shown. Click 'Apply' to create the flange.

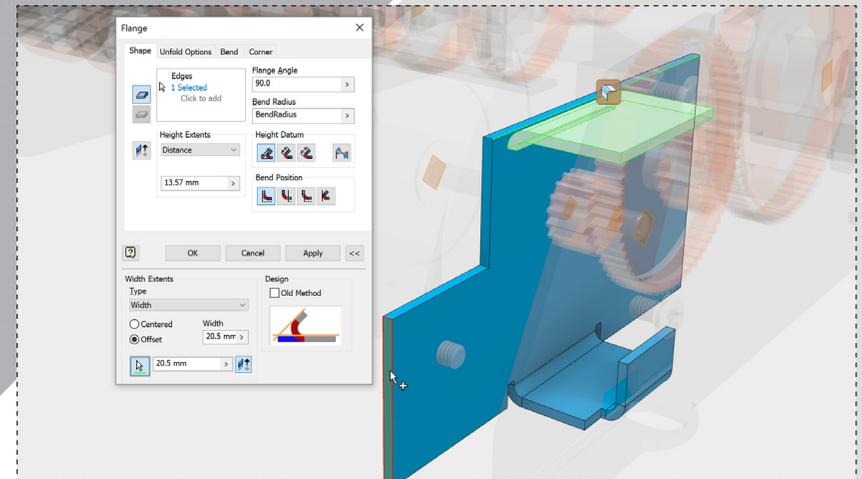
PART 2: ADDING FLANGES

9.



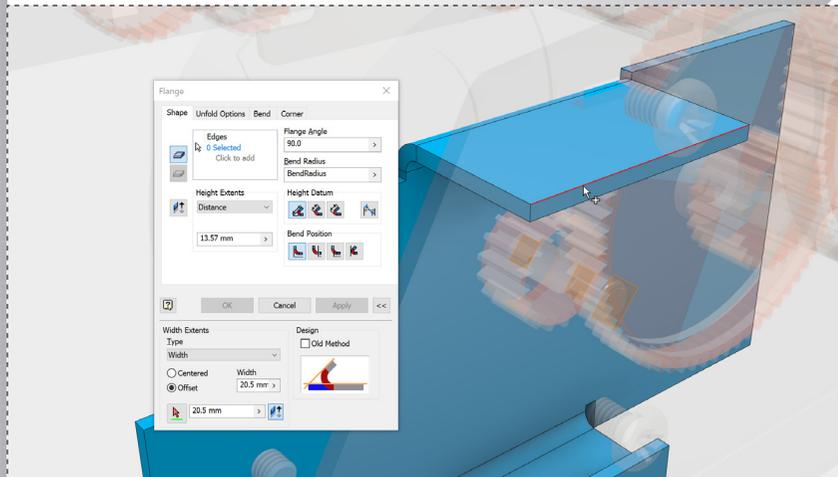
To start the next flange, click the edge as shown (flip if necessary).

10.



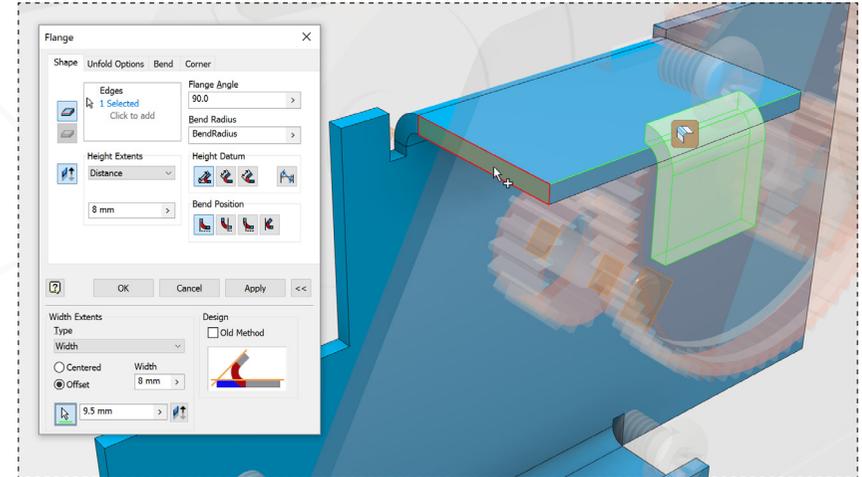
Again use 13.57mm for the 'Height Extents' and change the 'Width Extents' as shown. Click the arrow to select the face of the bracket as shown for the offset (flip if necessary).

11.



Select 'Apply' in the dialog to create the flange. Then select the edge as shown (flip if necessary).

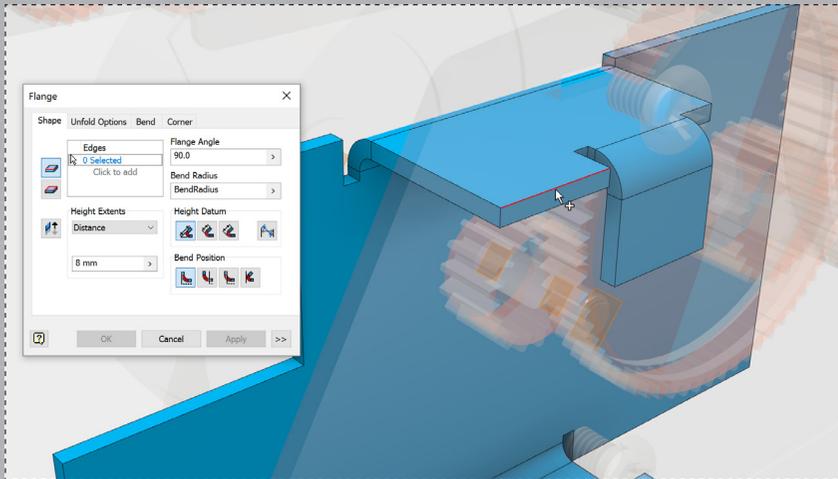
12.



Configure the flange using the values and options for the 'Height Extents' and 'Width Extents' as shown. Select the face shown for the offset reference. Click 'OK'.

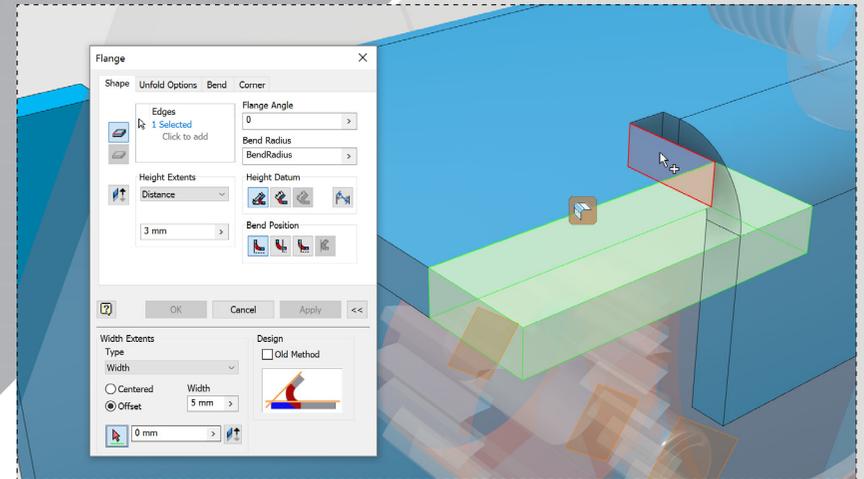
PART 2: ADDING FLANGES

13.



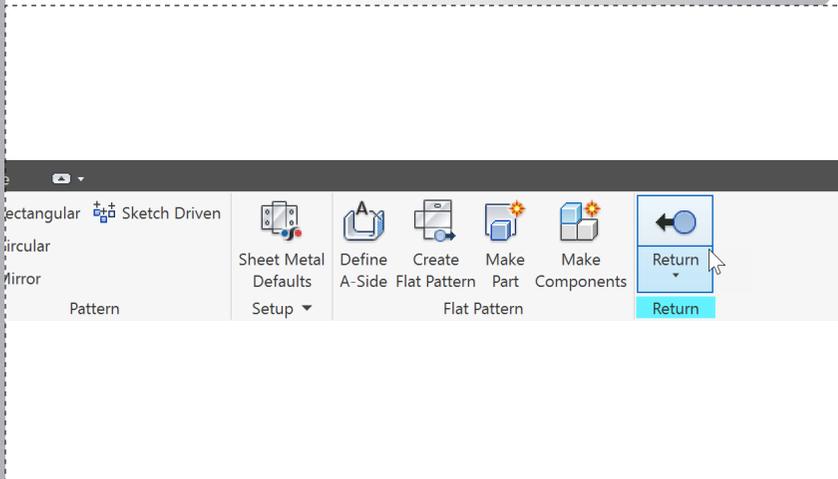
For the final flange, restart the 'Flange' command from the marking menu and select the edge as shown.

14.



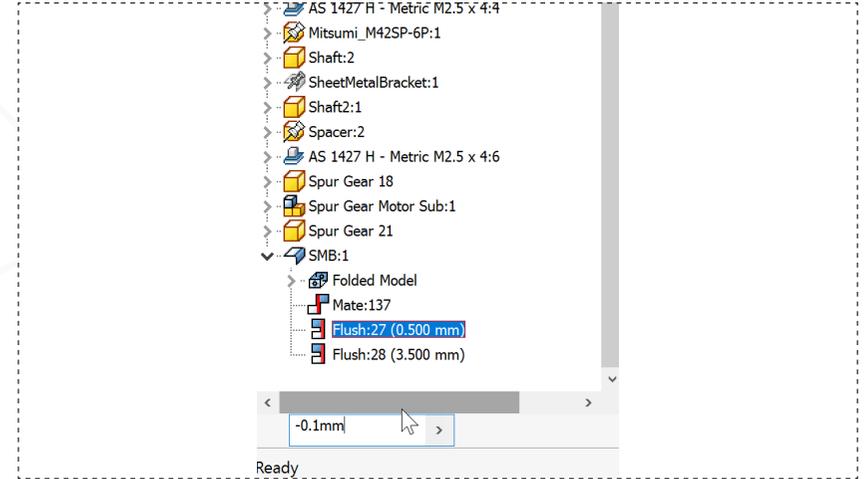
Change the 'Flange Angle' to 0 and configure the 'Height Extents' and 'Width Extents' as shown. Select the red highlighted face for the offset reference (flip if necessary).

15.



Hit 'OK' to complete the flange. Then select 'Return' in the ribbon menu to exit the editing environment and reactivate the entire assembly.

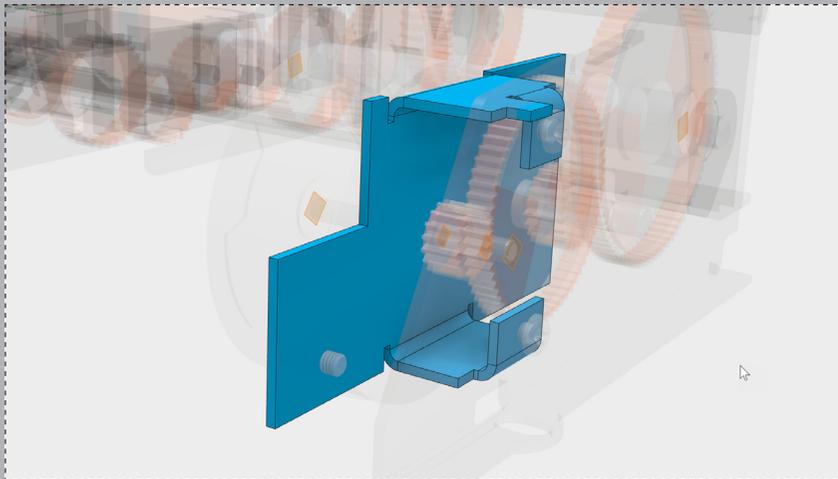
16.



In the model browser, click to edit the 'Flush:27 (0.500 mm)' constraint to be -0.100 mm so the locator flange is aligned correctly in the assembly.

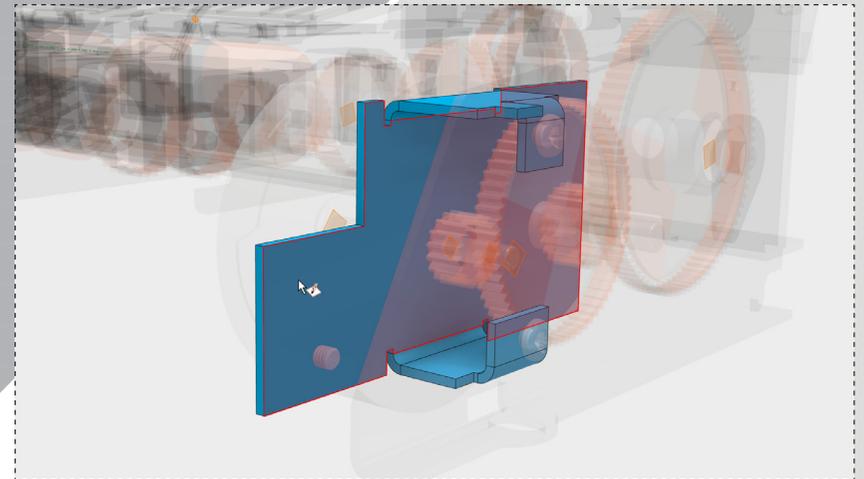
PART 3: CUTOUTS AND THREADED HOLES

1.



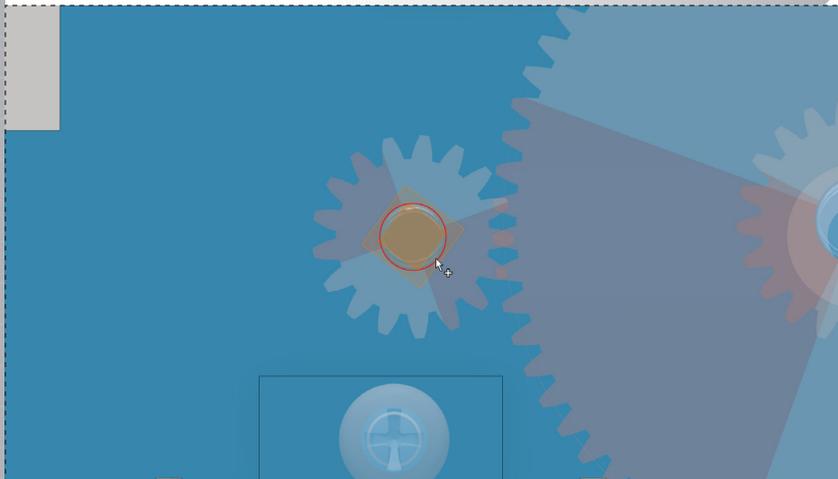
Double click the flange to re-enter the editing environment.

2.



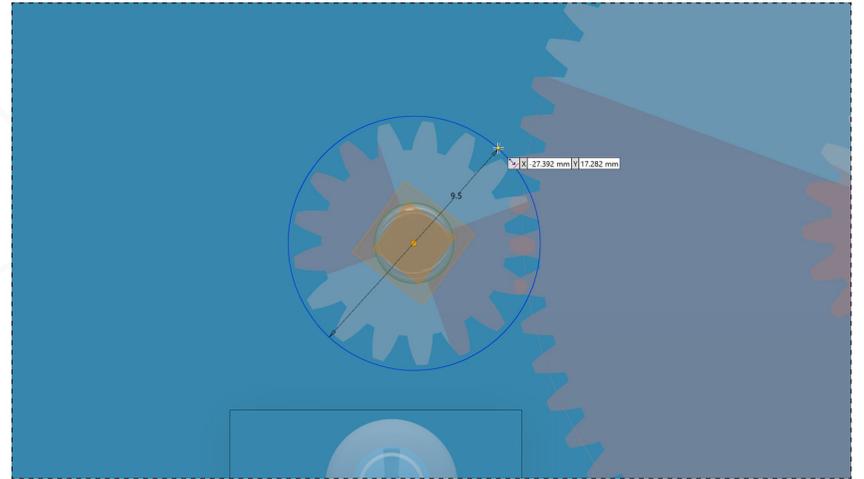
Right click and select 'New Sketch' from the marking menu. Select the face as shown highlighted as the sketch plane.

3.



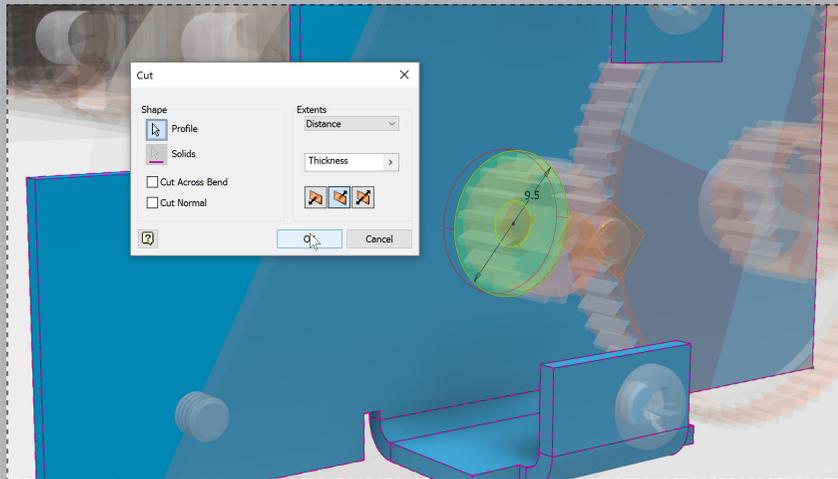
Right click and select 'Project Geometry' from the marking menu. Select the highlighted circular feature as shown. Right click again and select 'OK'.

4.



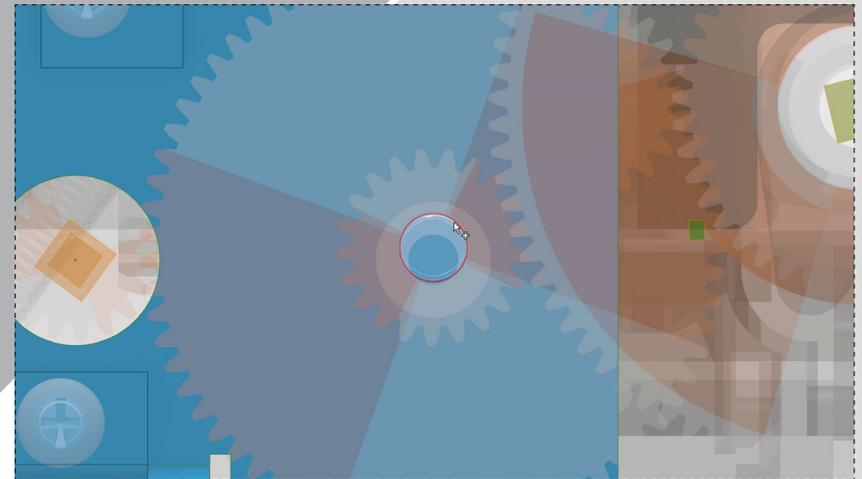
Start the 'Center Point Circle' from the marking menu. Select the center point of the projected geometry and enter 9.5mm in the dynamic dimension box and hit 'Enter'. Right click and 'OK'.

5.



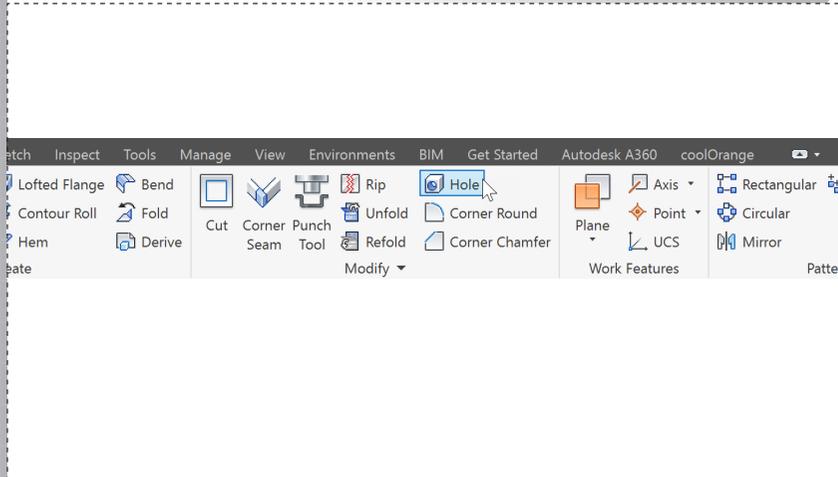
Click the green check in the ribbon to finish the sketch. Select 'Cut' from the marking menu and pick both circles in the sketch as the 'Profile' and select 'OK'.

6.



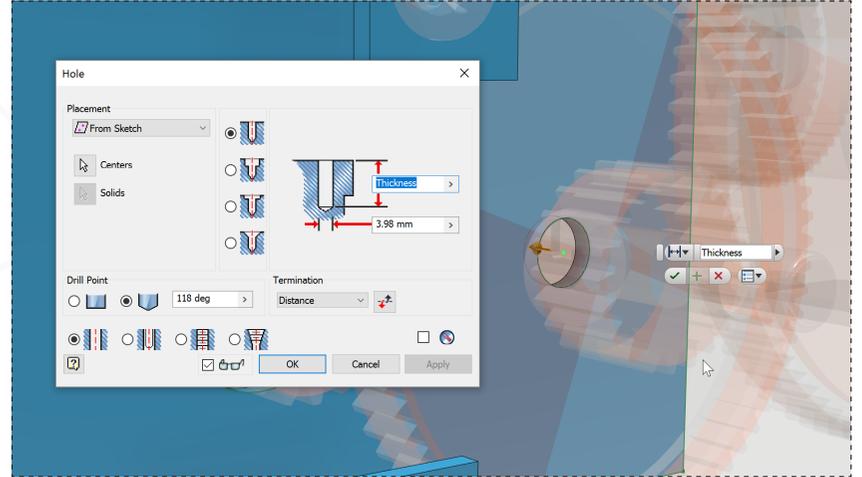
Create another new sketch on the same face as before. Start the 'Project Geometry' command from the marking menu and pick the circular feature shown. Right click and select 'OK'.

7.



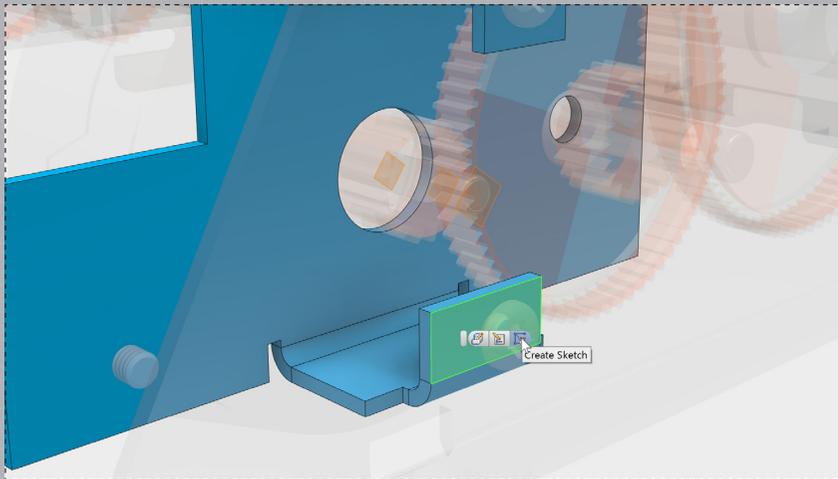
Right click and select 'Finish 2D Sketch'. In the ribbon, start the 'Hole' command.

8.



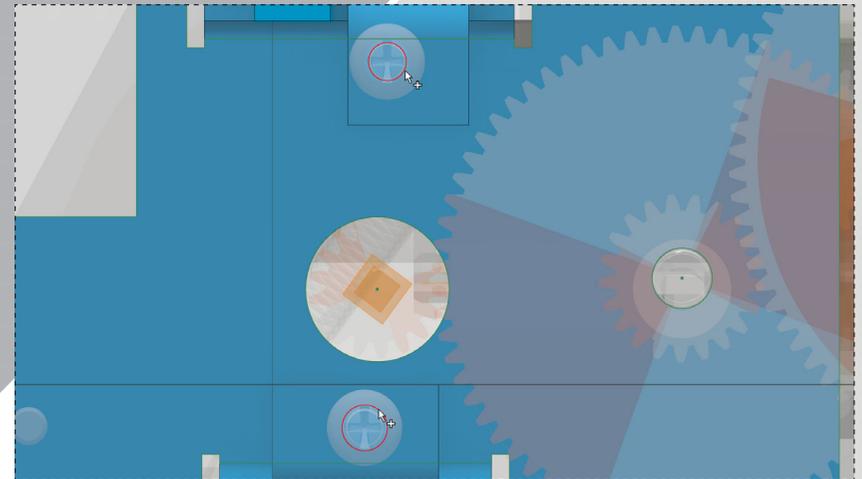
Pick the center point of the circle in the sketch and change the hole diameter in the dialog to 3.98mm. Select 'OK' to create the hole.

9.



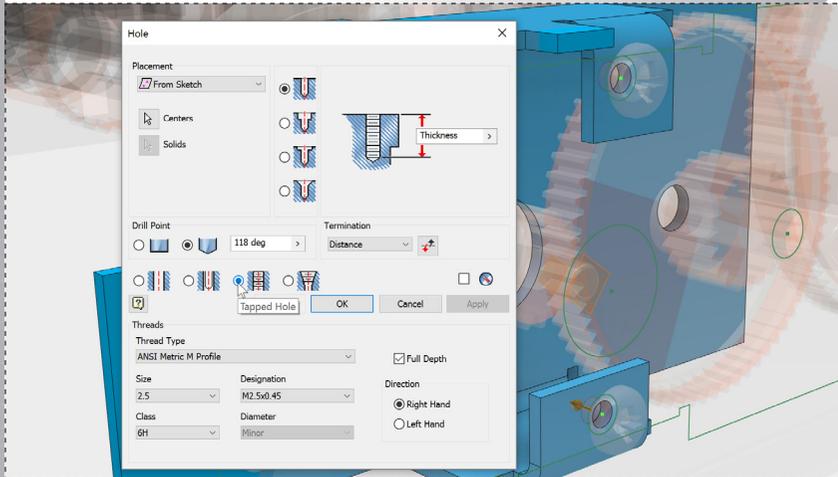
Click on the highlighted surface shown and then select 'Create Sketch' from the pop-up menu.

10.



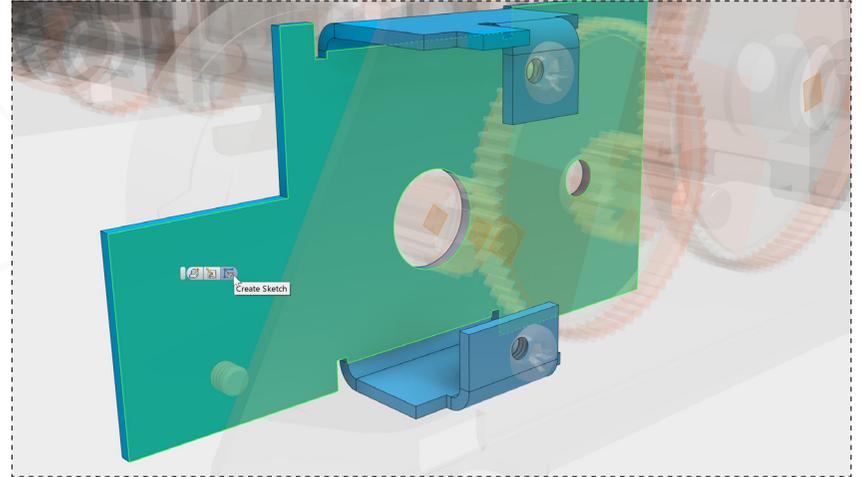
Start the 'Project Geometry' command from the marking menu. Select the two circular features highlighted above. Right click and select 'Finish 2D Sketch'.

11.



Restart the 'Hole' command from the ribbon. Select the center-points of the two circles in the sketch. Set the hole settings as shown. Click 'OK' to create the threaded holes.

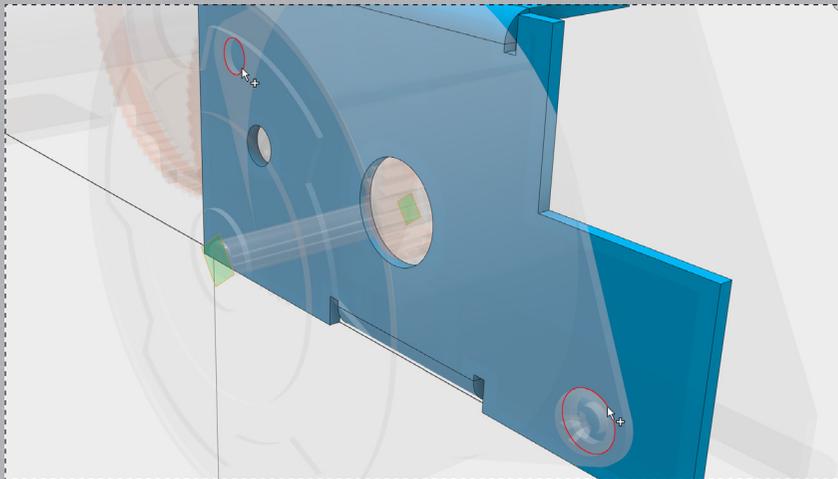
12.



Click on the highlighted surface shown and then select 'Create Sketch' from the pop-up menu.

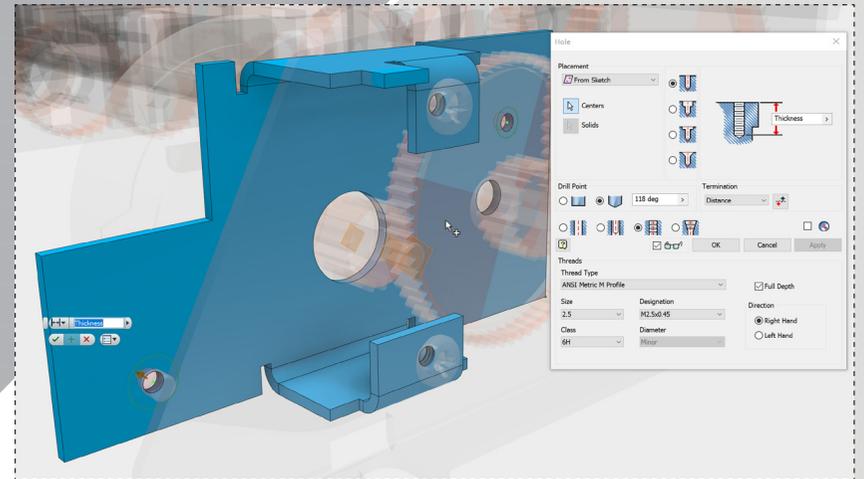
PART 3: CUTOUTS AND THREADED HOLES

13.



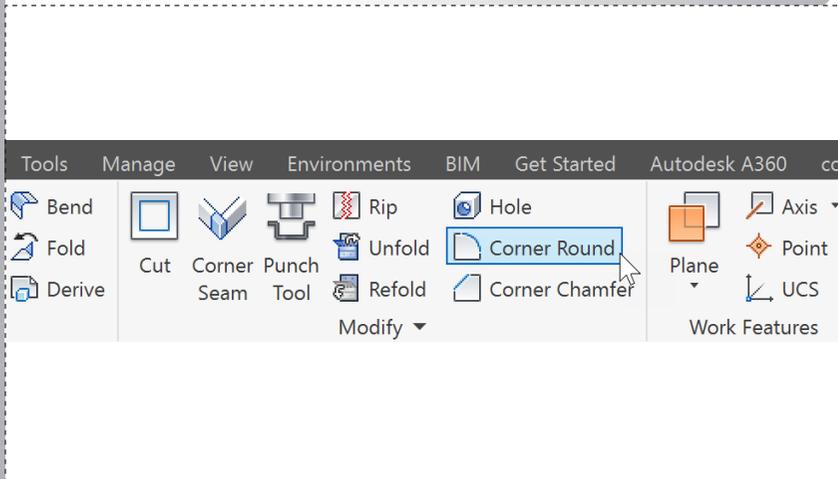
Rotate the model to view the back of the bracket. Start the 'Project Geometry' command and select the two circular features highlighted. Right click and select 'Finish 2D Sketch'.

14.



Restart the 'Hole' command from the ribbon. Select the center-points of the two circles in the sketch. Use the same threaded hole settings as before. Click 'OK' to create the threaded holes.

15.



Start the 'Corner Round' command from the ribbon.

16.



Select the edges as shown above and change the radius to 1mm. Click 'OK'. In the ribbon, select 'Return' to finish editing and save and close the files.



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