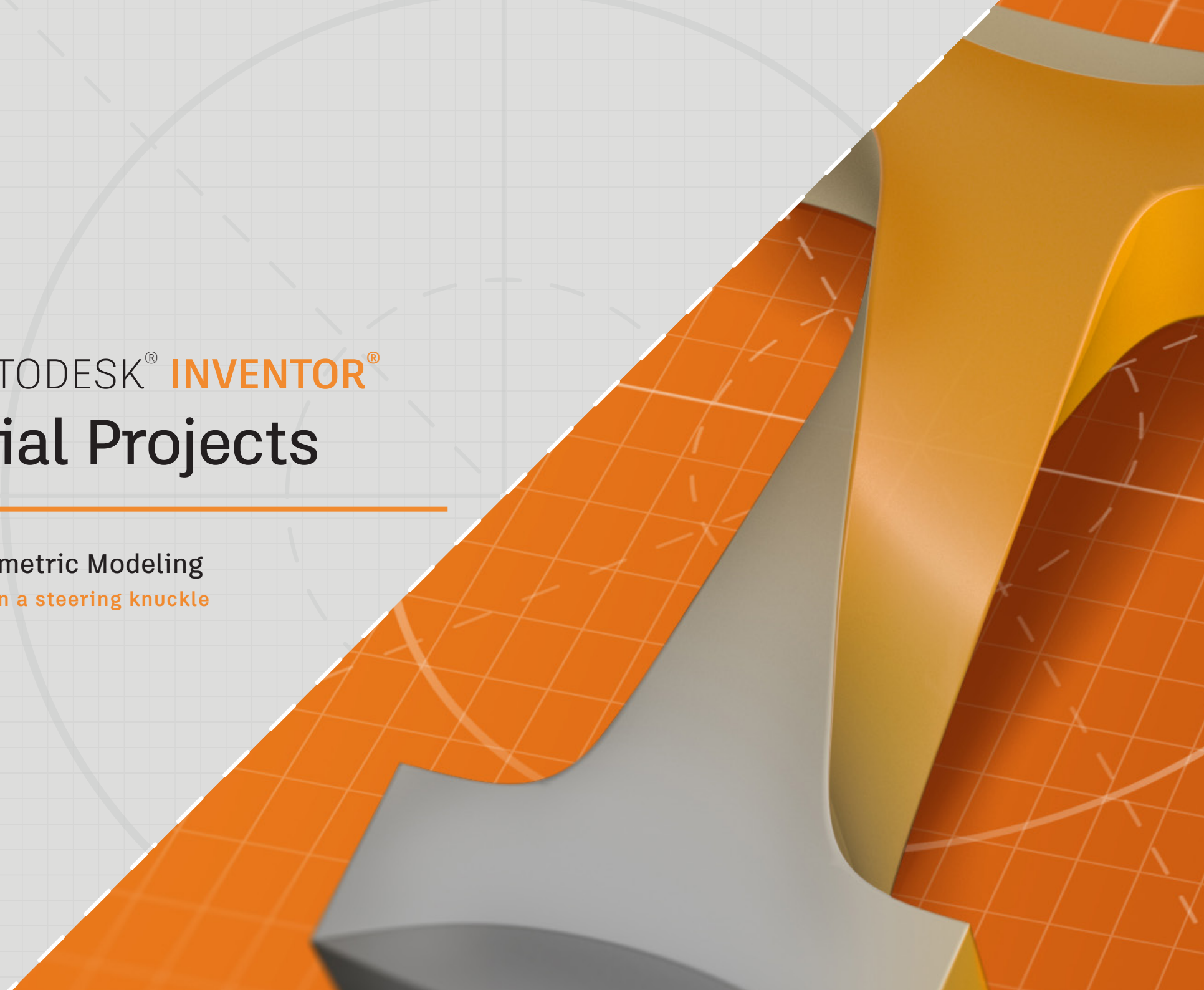
AUTODESK® INVENTOR®

Trial Projects

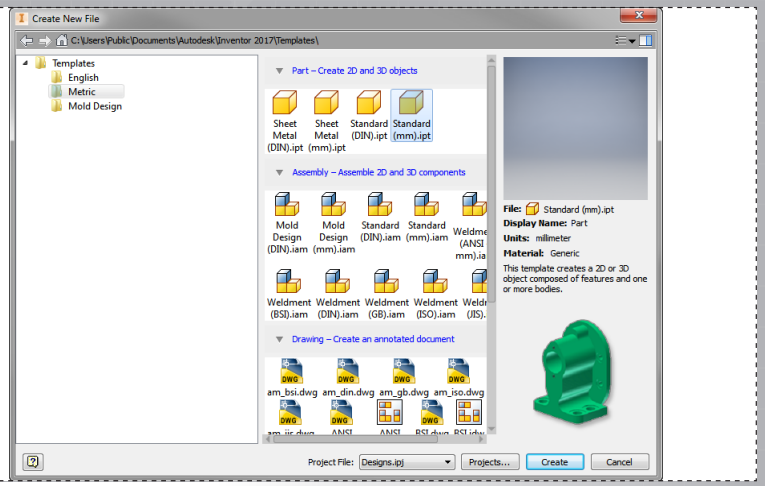
Parametric Modeling

Design a steering knuckle



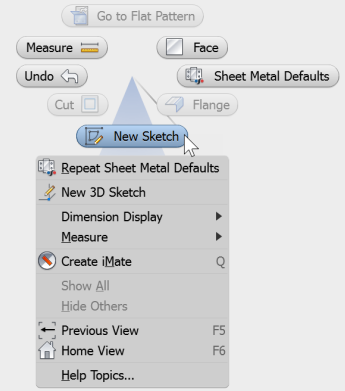
PART 1: CREATE THE BASE MODEL

1.



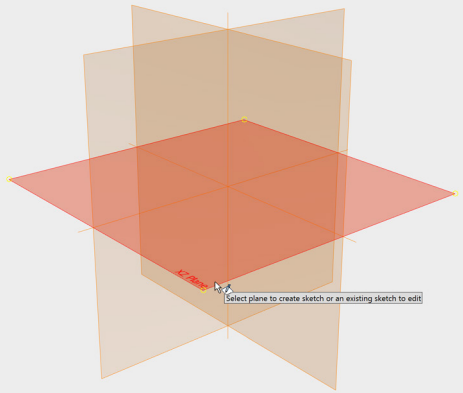
In Inventor, click the ‘Projects’ icon in the ribbon. Select the project name ‘Default’. Select ‘New’ in the ribbon and then choose the ‘Metric’ folder, ‘Standard (mm).ipt’ template.

2.



Right click in the modeling window to open the marking menu and select ‘New Sketch’.

3.



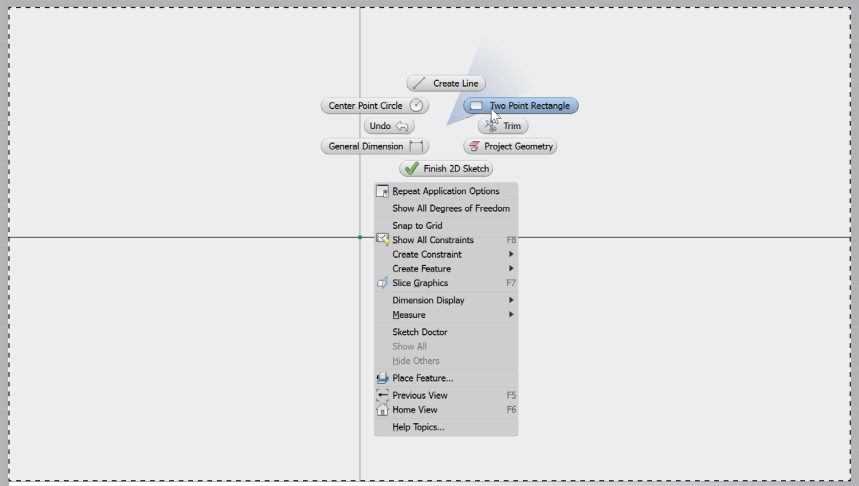
Select the ‘X-Z’ plane as the sketch plane.

4.

Using the ViewCube in the top right corner, use the rotation arrow to rotate the view counter-clockwise. The word ‘Top’ should be horizontally displayed.

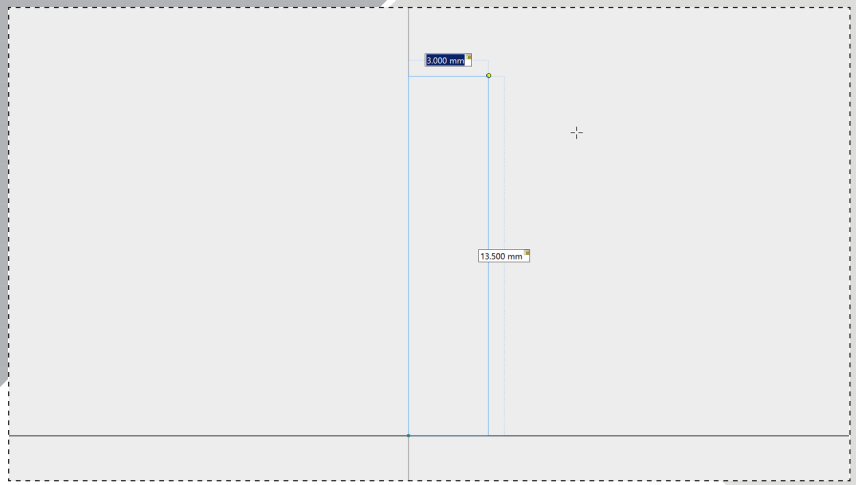
PART 1: CREATE THE BASE MODEL

5.



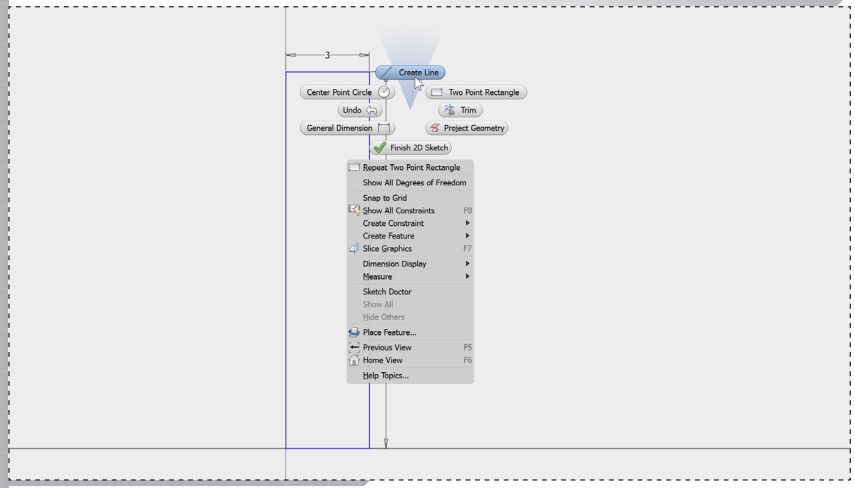
Right click to open the marking menu. Select 'Two-Point Rectangle'.

6.



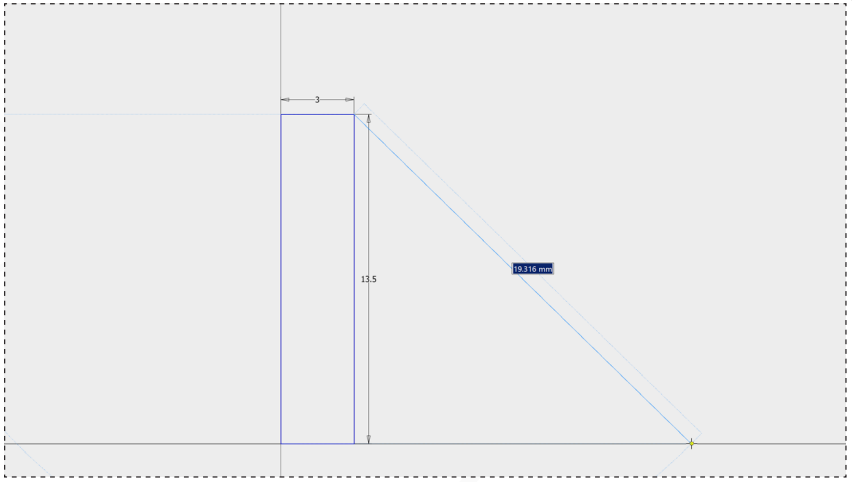
Click the origin point to start the rectangle and move your cursor to the upper right. In the dynamic dimensioning boxes, enter 3mm and 13.5mm as shown. Click to create the rectangle.

7.



Right click and select 'OK' to close the rectangle command. Using the marking menu, start the 'Line' command.

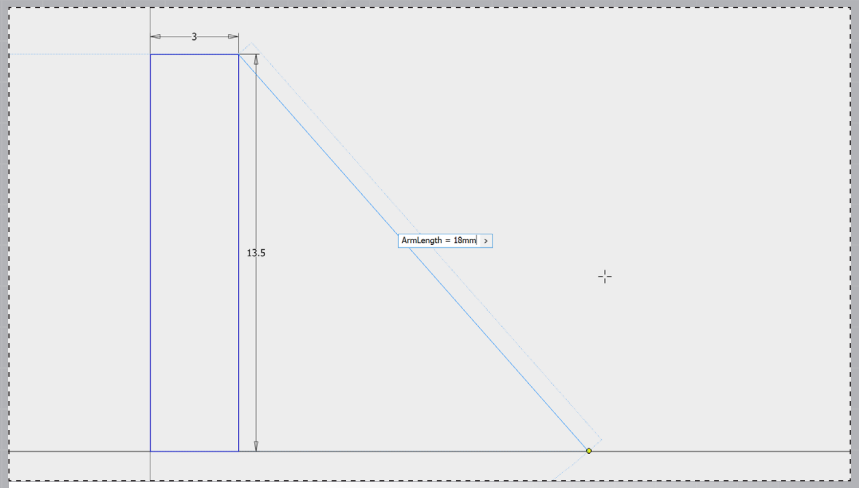
8.



Start the line by clicking the top right corner of the rectangle. Move your cursor down and to the right (do not click the 2nd point of the line yet).

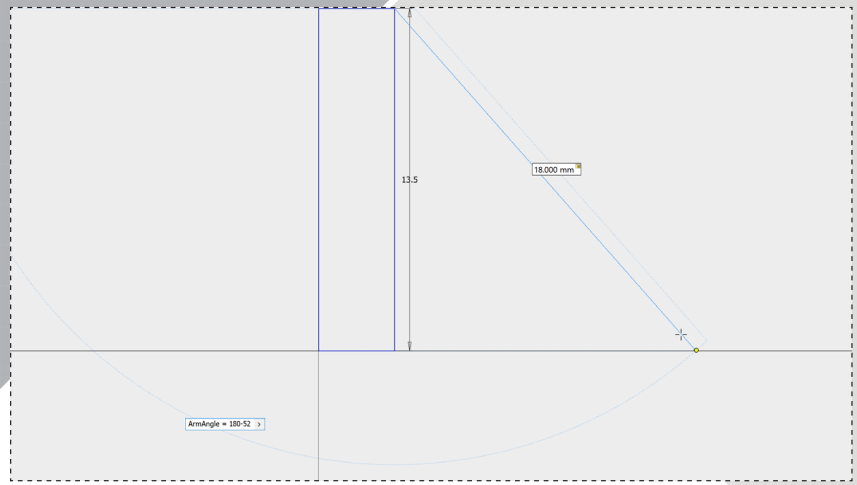
PART 1: CREATE THE BASE MODEL

9.



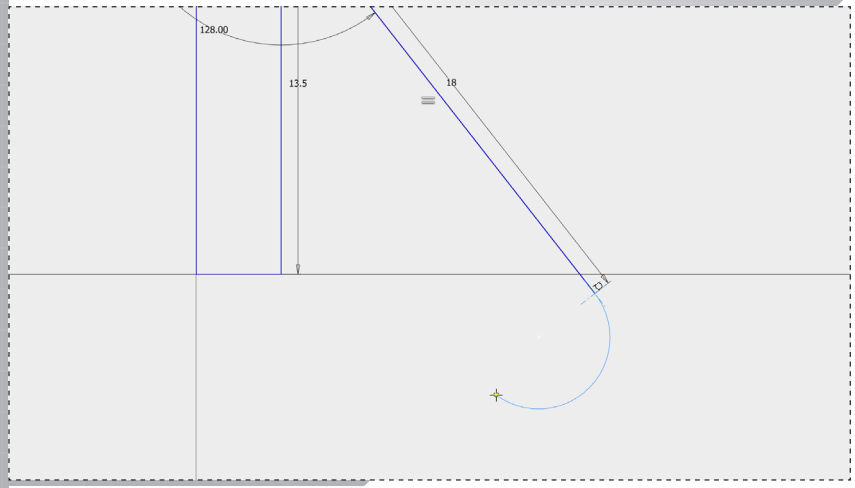
In the dynamic dimension box as shown, type in 'ArmLength = 18mm'. This creates a parameter named 'ArmLength' that can be referenced in other dimensions.

10.



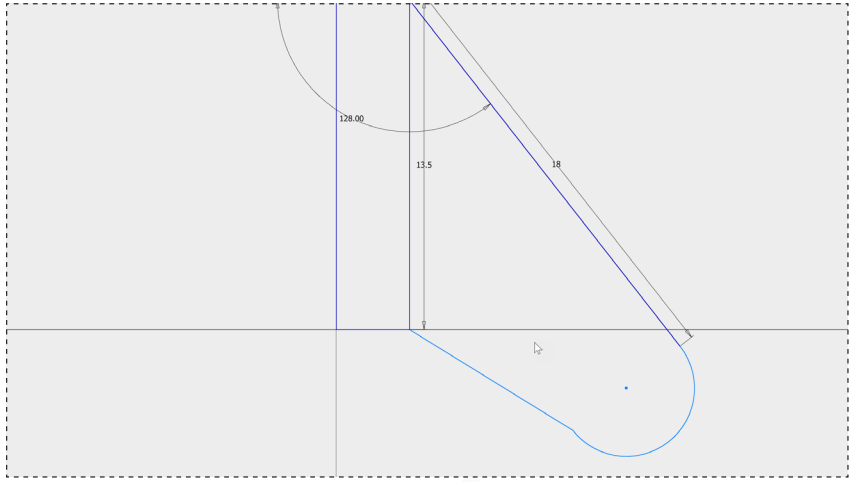
For the angular dimension, type in 'ArmAngle = 180-52'. This will calculate the value for the 'ArmAngle' parameter as 128-degrees. Click to complete the line.

11.



Click and hold your left mouse button down on the end point of the previous line to create an arc. Drag your cursor to create the rough shape as shown.

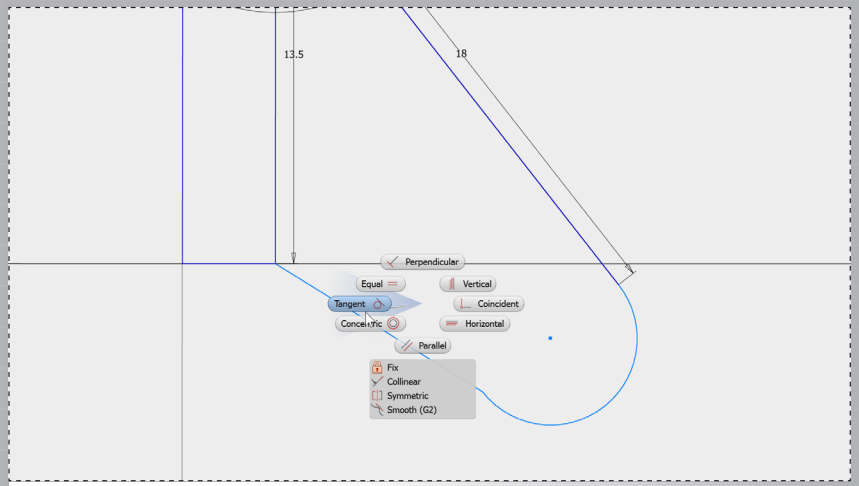
12.



Click the bottom right corner of the rectangle to complete the line segment. Right click and select 'OK' to close the line command.

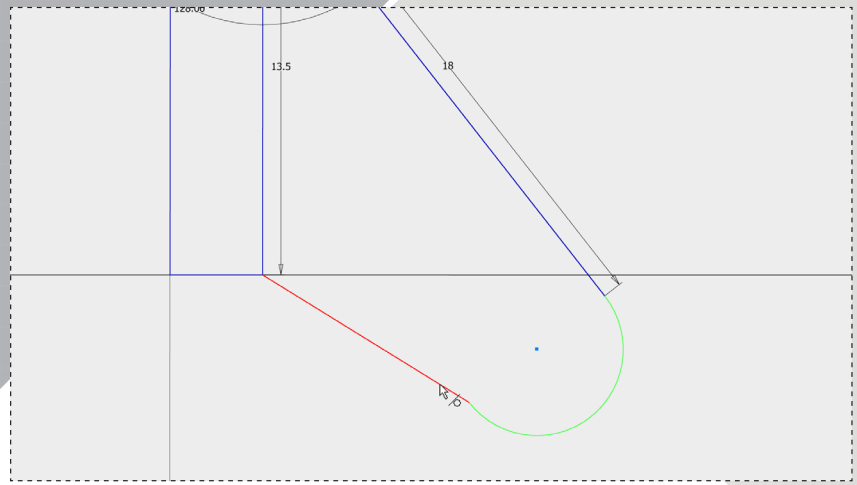
PART 1: CREATE THE BASE MODEL

13.



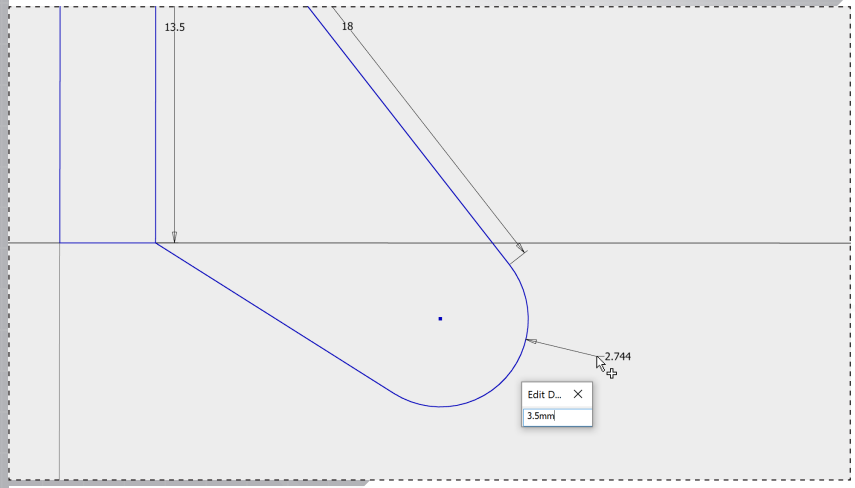
Hold ‘Ctrl’ on your keyboard and right click to bring up the constraint marking menu. Select ‘Tangent’.

14.



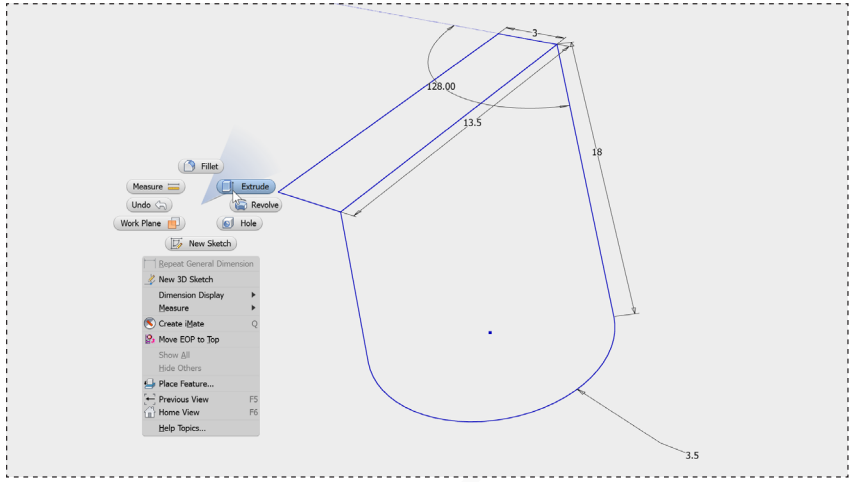
Select the arc and line segment as shown to add a tangent constraint.

15.



Right click and select ‘General Dimension’ in the marking menu. Select the arc and click to place the dimension. In the dimension box, type in 3.5mm and hit ‘Enter’.

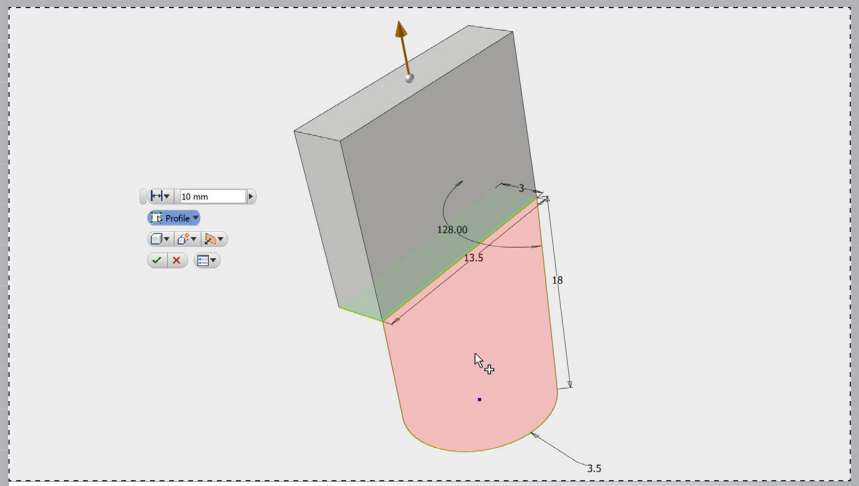
16.



Right click and select ‘Finish 2D Sketch’ from the marking menu. From the marking menu, start the ‘Extrude’ command.

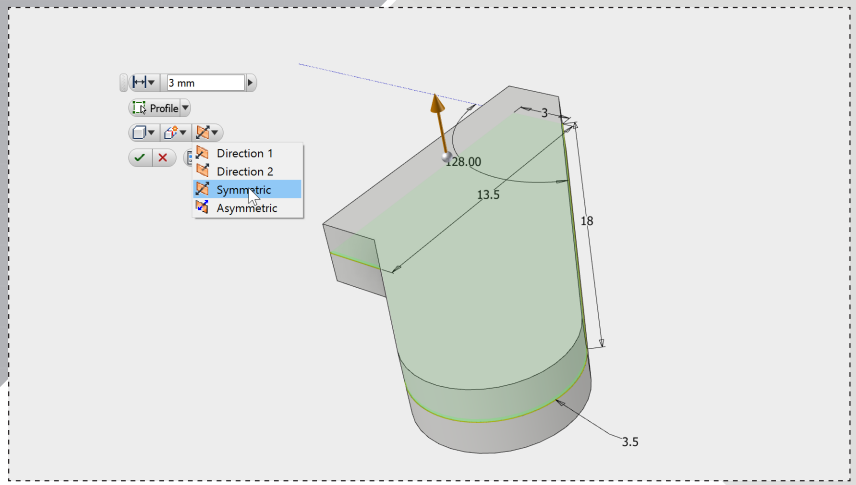
PART 1: CREATE THE BASE MODEL

17.



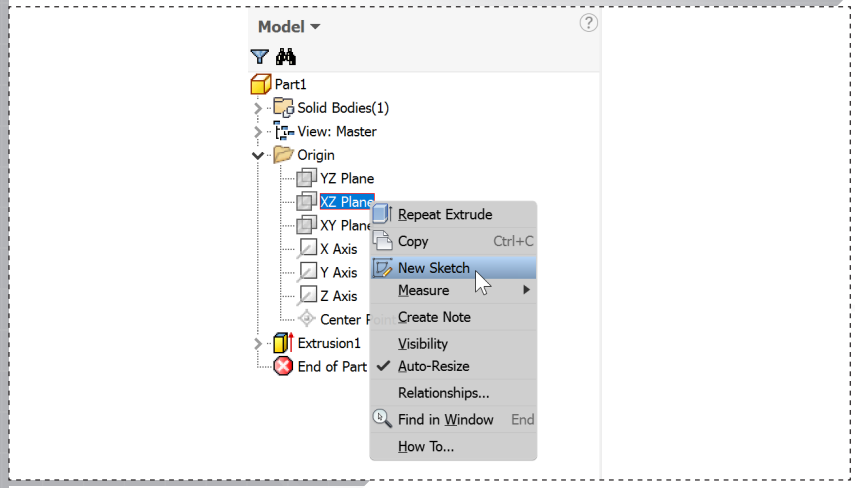
Select both closed profiles in the sketch.

18.



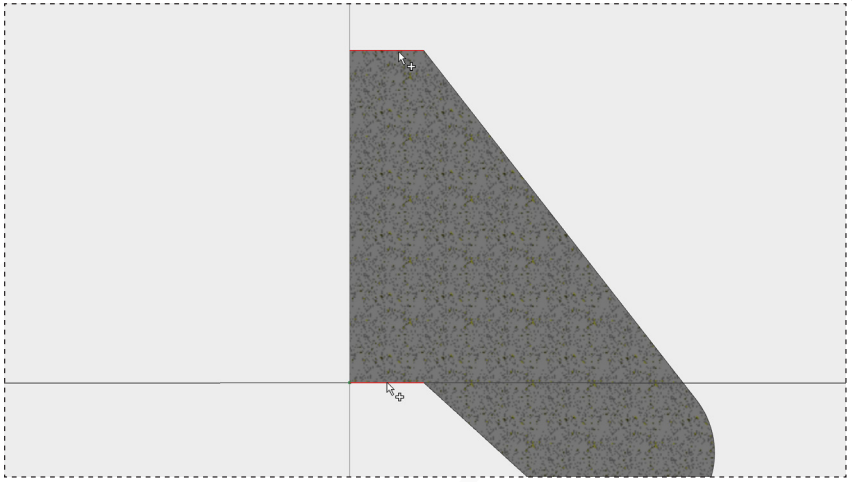
As shown, change the extrusion type to 'Symmetric' and enter in a value of 3mm for the extrusion depth. Click the green check mark to finish the extrude.

19.



In the model browser on the left side of the screen, expand the 'Origin' folder. Right click on the XZ Plane and select 'New Sketch'.

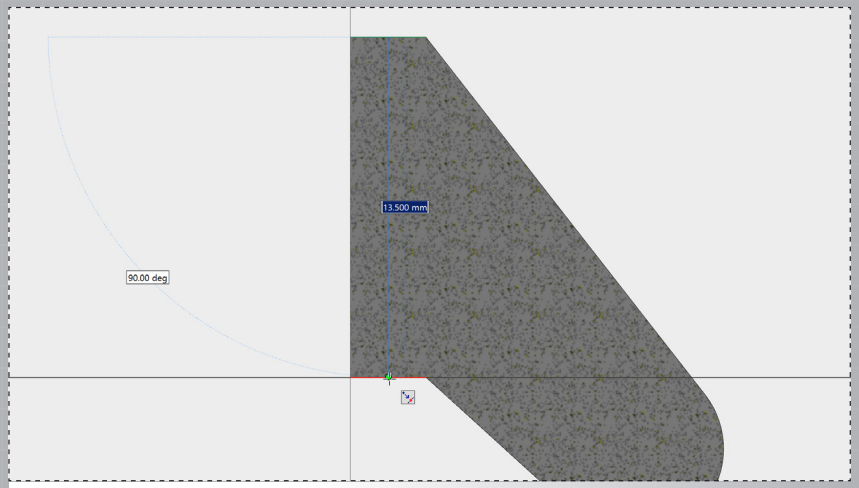
20.



In the sketch view, press F7 on your keyboard to slice the view. Start the 'Project Geometry' command from the marking menu. Click on the two edges as shown.

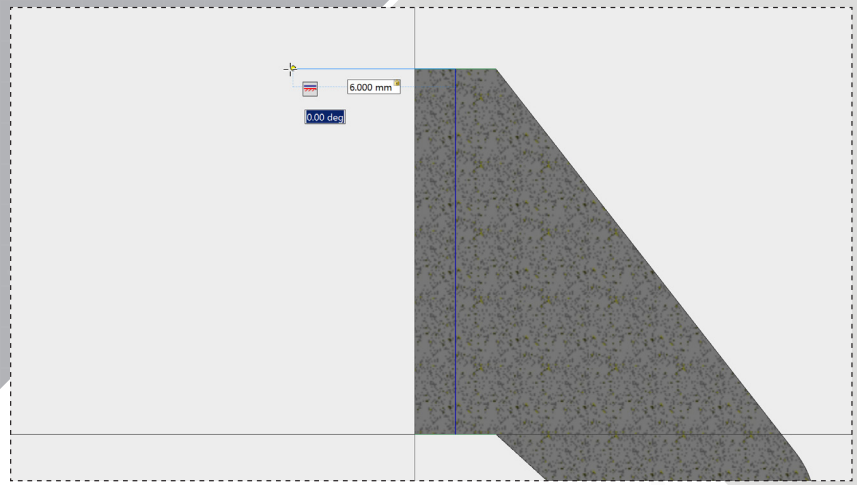
PART 1: CREATE THE BASE MODEL

21.



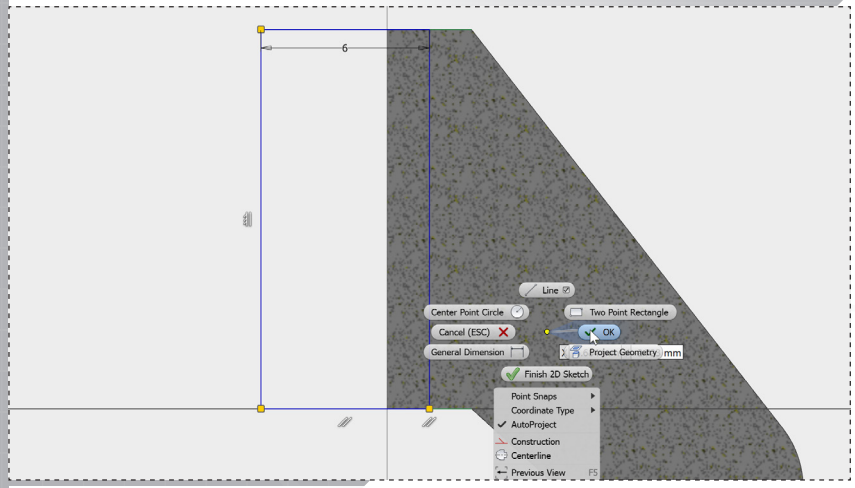
Start the ‘Create Line’ command from the marking menu. Create a vertical line between the midpoints of the two projected lines. A green dot will appear, indicating the center point.

22.



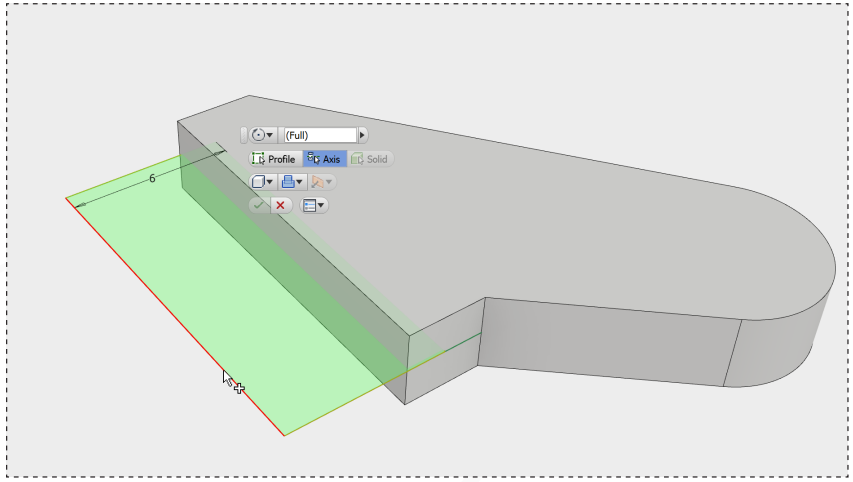
Start the next line segment from the same location, and move your cursor to the left. Type 6mm into the dynamic dimension box and click to create a horizontal line.

23.



Draw additional lines as shown. When complete, right click and select ‘OK’ to finish the line command. Right click and select ‘Finish 2D Sketch’.

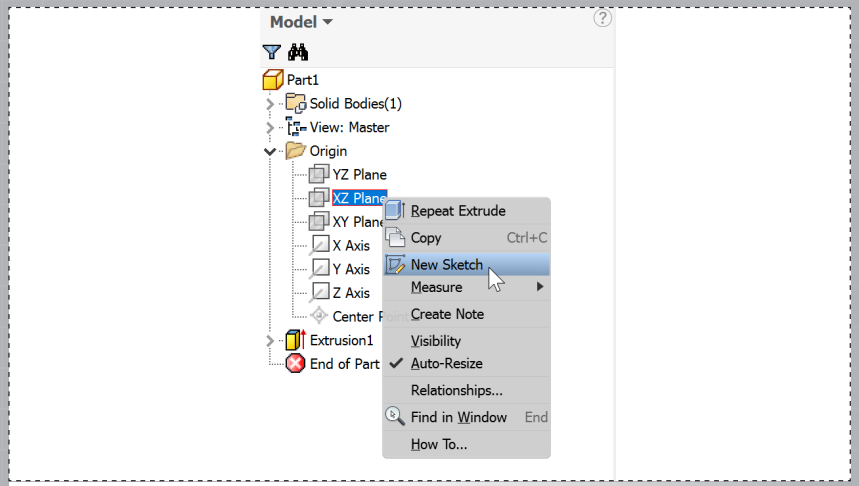
24.



Start the ‘Revolve’ command from the marking menu. The profile will auto-select. Choose the left edge as the axis as shown and click the green check box to complete the revolve.

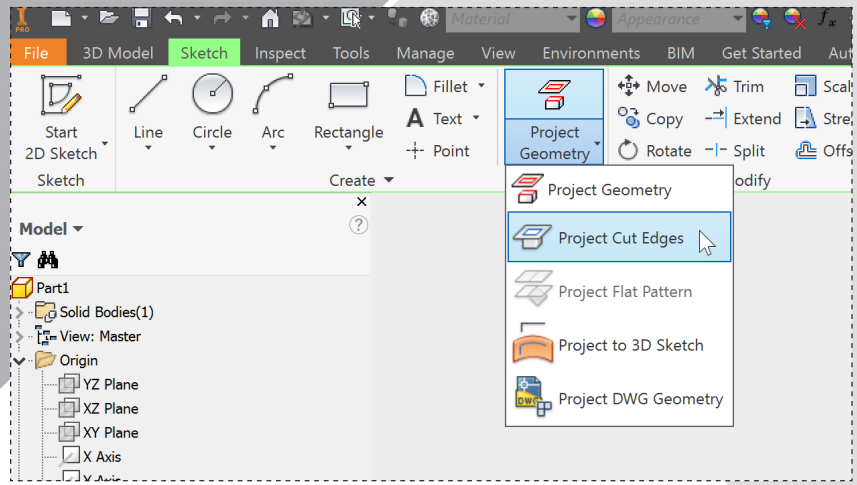
PART 1: CREATE THE BASE MODEL

25.



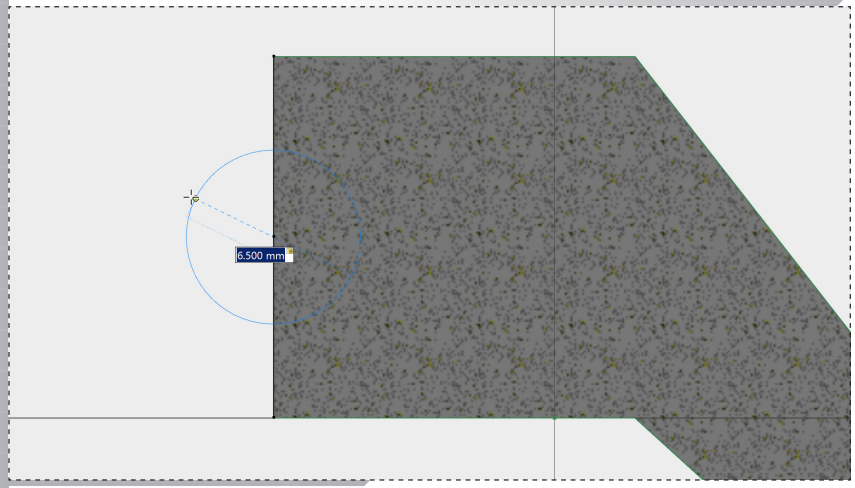
Create a new sketch by right clicking on the XZ plane from the model browser and selecting ‘New Sketch’.

26.



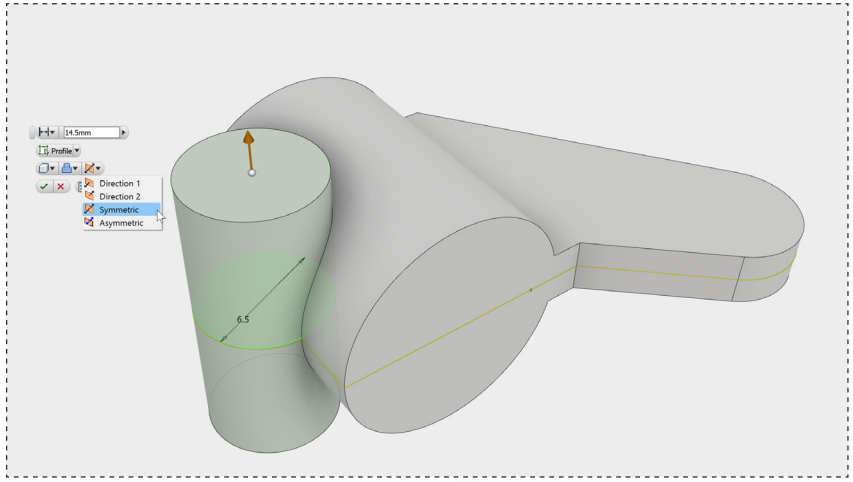
Press F7 on your keyboard to slice the view and, in the ribbon menu, select ‘Project Cut Edges’.

27.



Start the ‘Center Point Circle’ from the marking menu. Click the center point *(look for green dot)* of the left edge as shown and enter 6.5mm as the diameter. Hit ‘Enter’ to complete the circle.

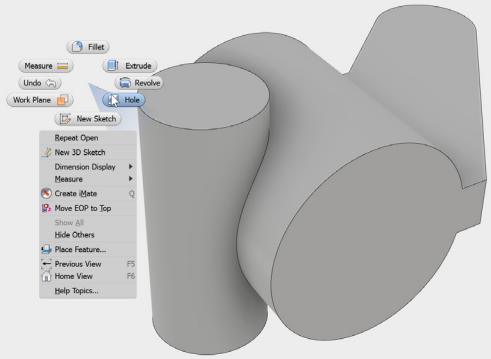
28.



Right click and select ‘Finish 2D Sketch’. Start the ‘Extrude’ command from the marking menu. Create a 14.5mm symmetric extrude of the profile.

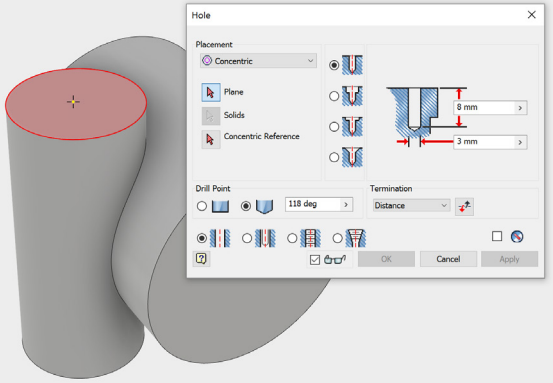
PART 2: ADD PLACED FEATURES

1.



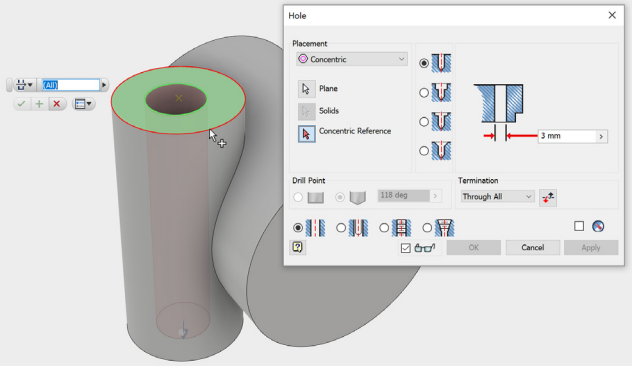
Right click in the model window to open the marking menu. Select the ‘Hole’ command.

2.



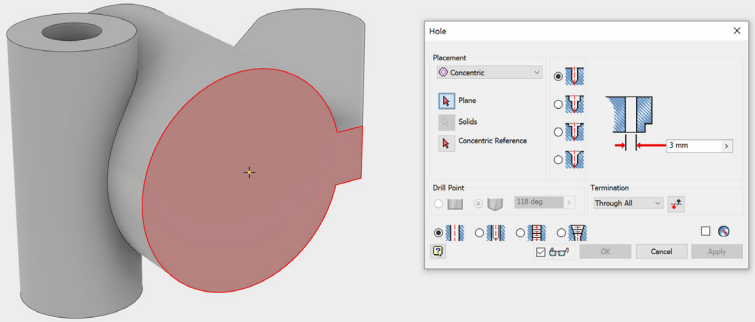
Change the placement option to ‘Concentric’ and click to select the face as shown.

3.



Select the outer edge of the cylinder as the ‘Concentric Reference’. Set the hole diameter to 3mm, and change the termination type to ‘Through All’. Click ‘Apply’ or the plus sign.

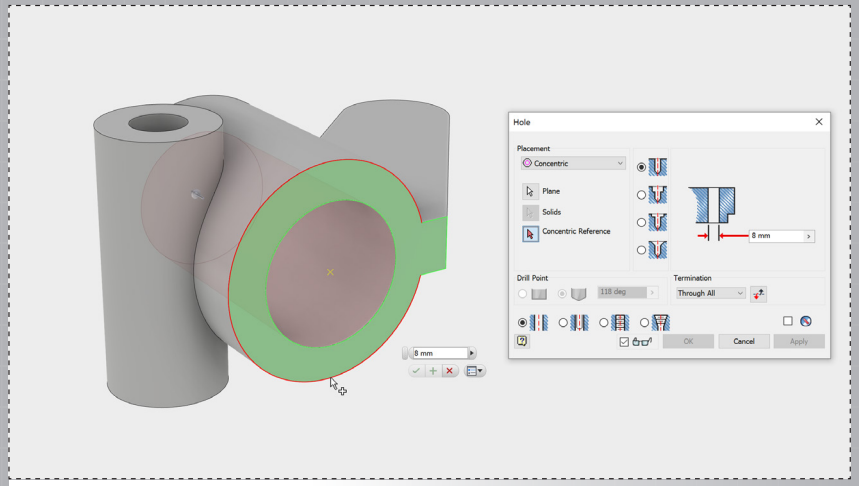
4.



Click the face shown to place another hole.

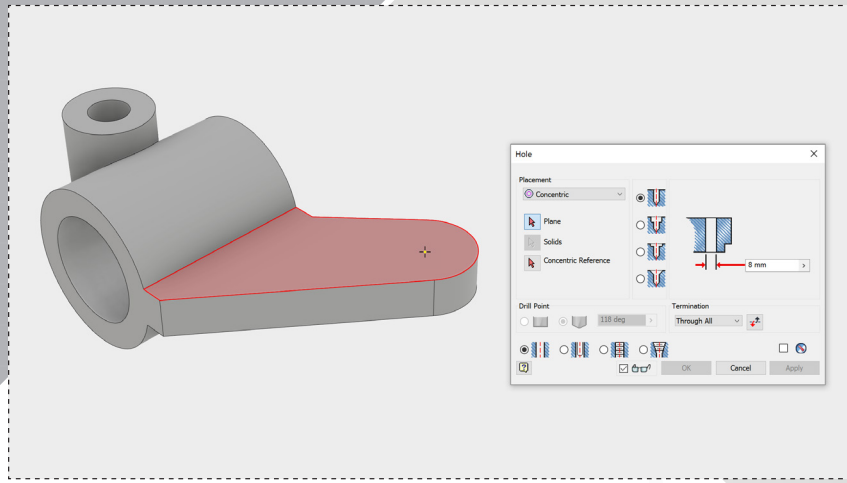
PART 2: ADD PLACED FEATURES

5.



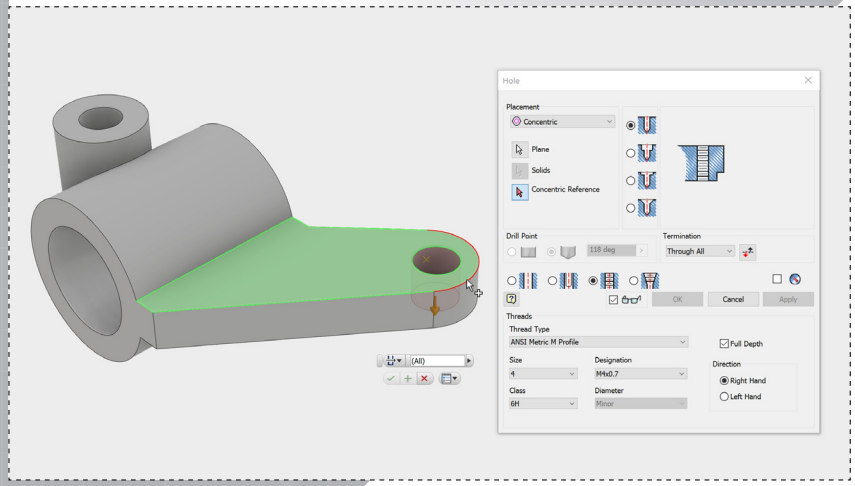
Click the outer edge of the cylinder as the 'Concentric Reference' as shown and change the diameter to 8mm. Click 'Apply' or the plus sign.

6.



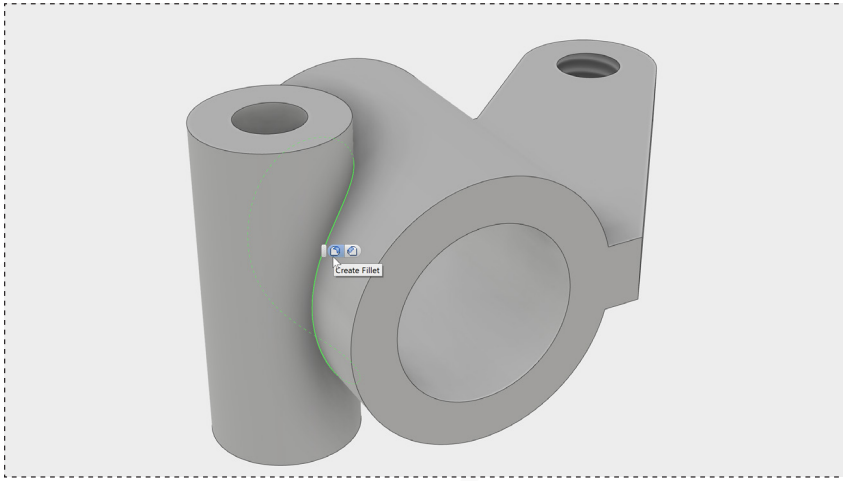
Select the face as shown for the final hole.

7.



Select the outer edge of the round feature as the 'Concentric Reference'. Click the radio button next to the threaded hole, and change the values as shown. Click 'OK' to finish.

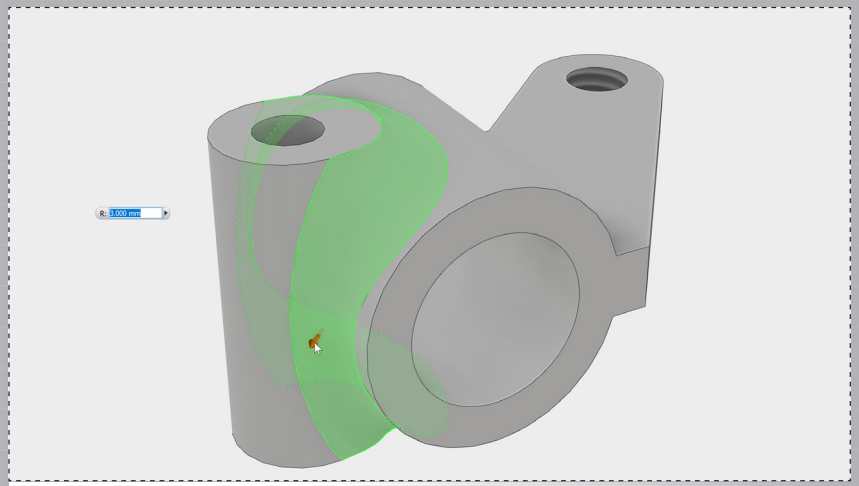
8.



Select the edge as highlighted in the image above and click 'Create Fillet' from the pop-up menu.

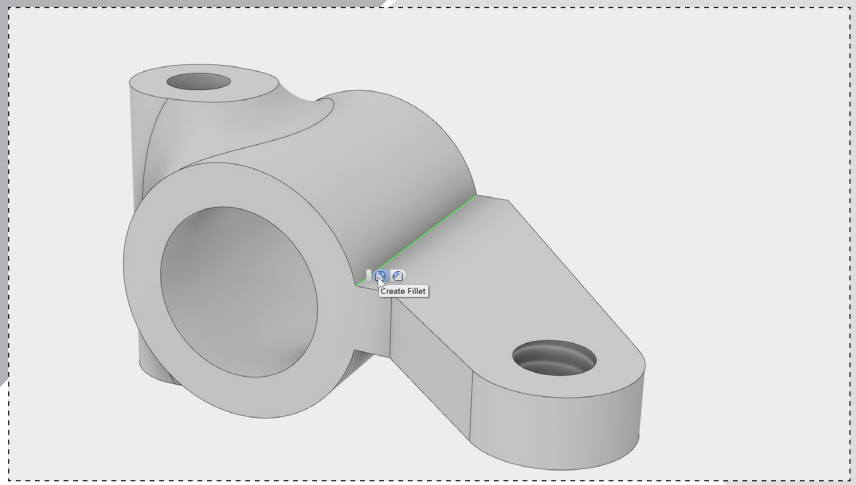
PART 2: ADD PLACED FEATURES

9.



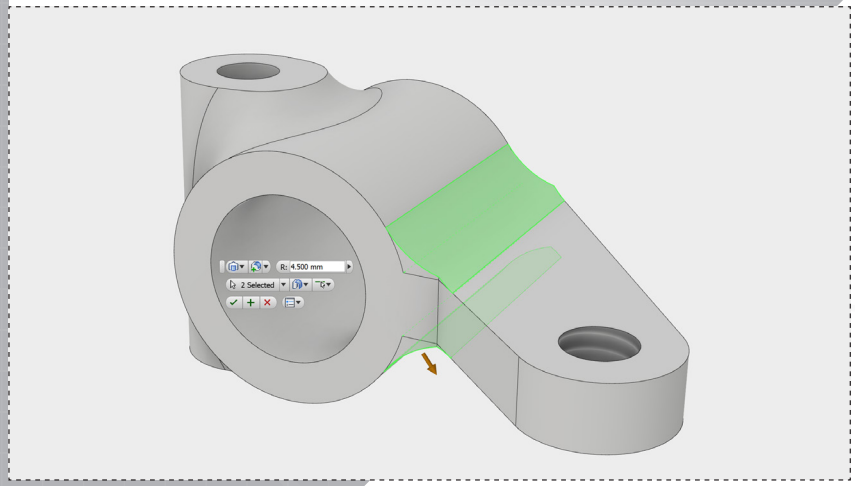
Drag the arrow to dynamically size the fillet until the dimension box shows a 3mm radius. Click the green check mark to complete the fillet.

10.



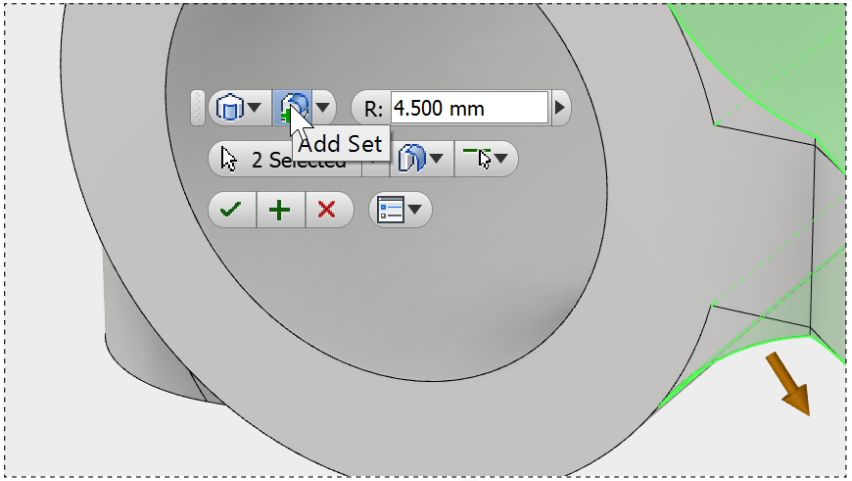
Select the edge as highlighted in the image above and select ‘Create Fillet’ from the pop-up menu.

11.



Add the edge underneath the flange by clicking on it. Use the arrow to size the fillet to 4.5mm (do not click the green check mark yet).

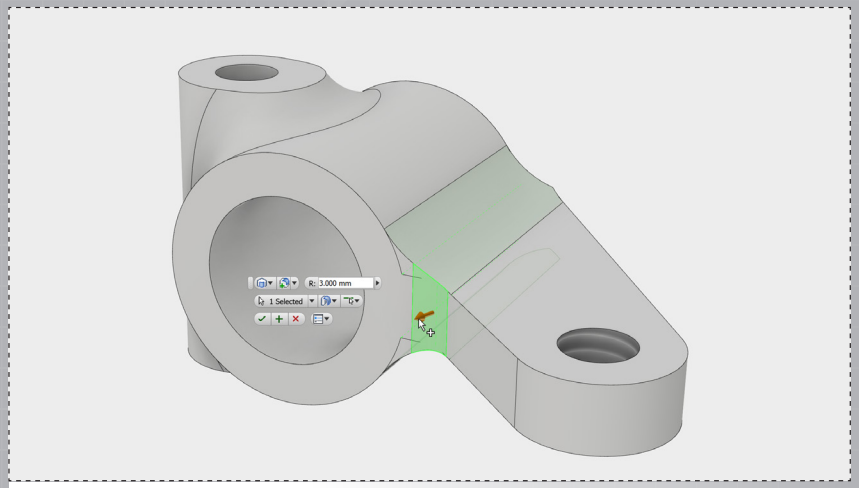
12.



In the fly-over menu, select ‘Add Set’.

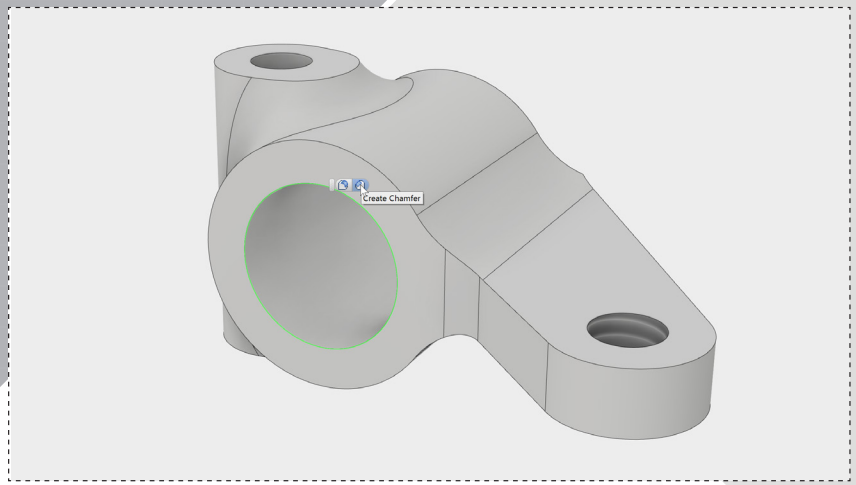
PART 2: ADD PLACED FEATURES

13.



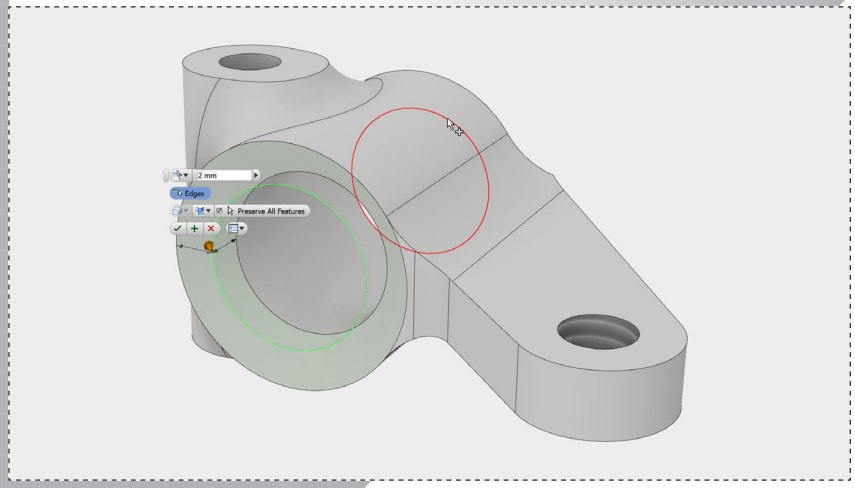
Select the edge shown and resize the fillet to 3mm. This will affect only this edge, the other fillets will remain 4.5mm. Click the green check mark to complete the fillets.

14.



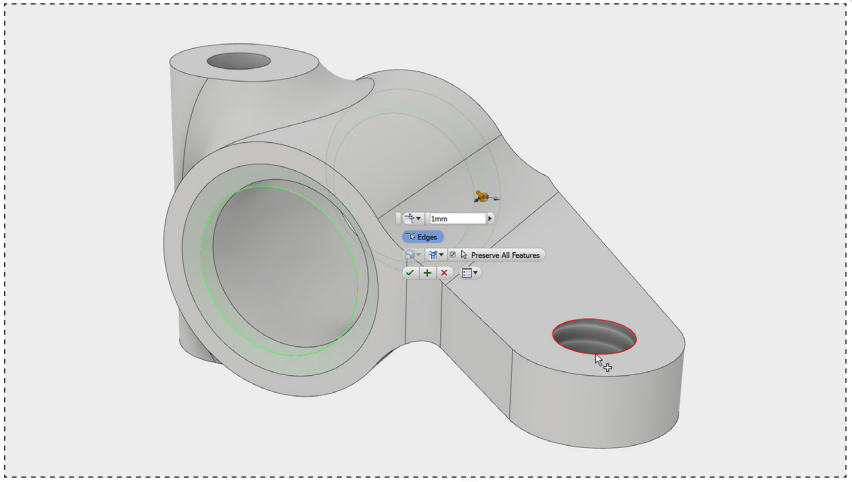
Select the edge of the hole as highlighted in the image above and select ‘Create Chamfer’ from the pop-up menu.

15.



Add the other edge of the hole as shown.

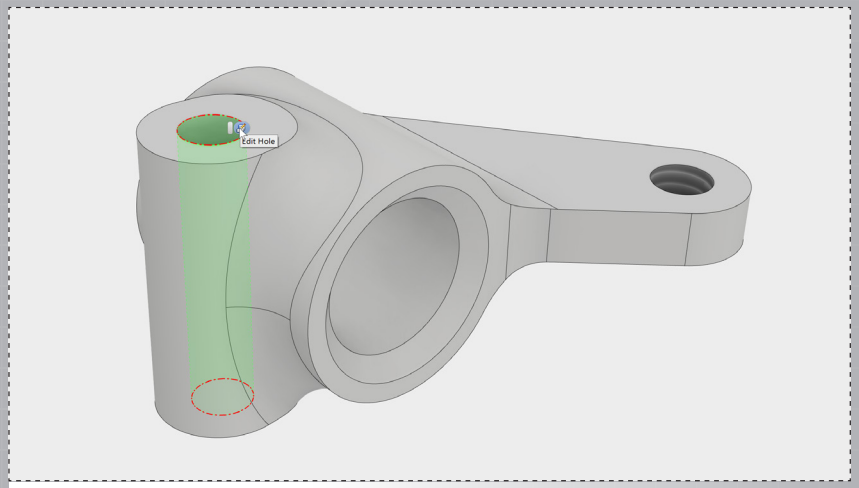
16.



Change the chamfer dimension to 1mm and select the green check mark to complete the chamfer.

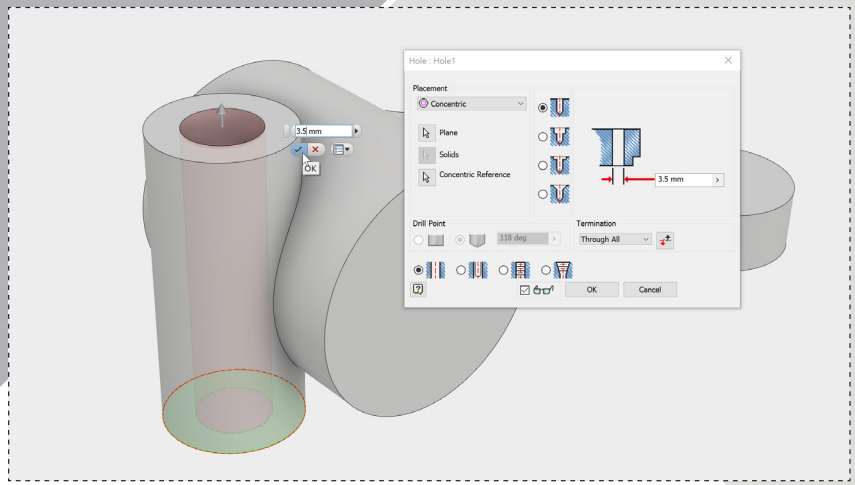
PART 3: EDITING THE MODEL

1.



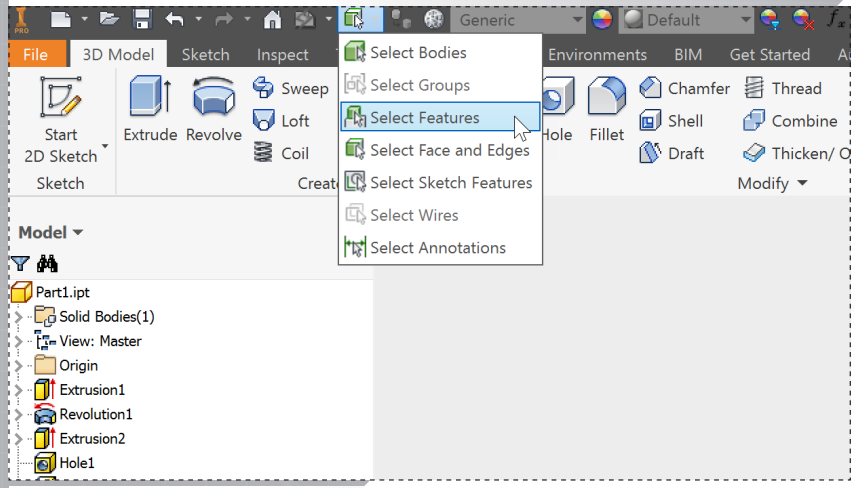
Select the hole surface as shown and click ‘Edit Hole’ from the pop-up menu.

2.



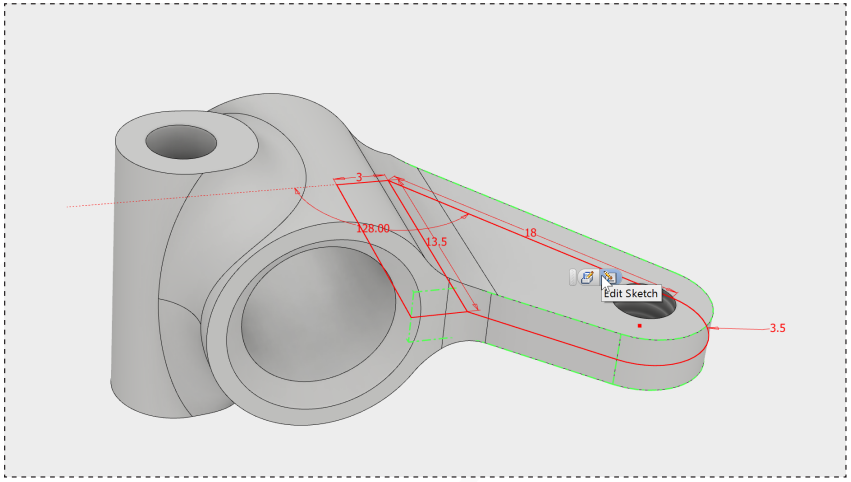
Change the diameter of the hole to 3.5mm and select the green check box to update the model.

3.



In the top toolbar, change the selection filter to ‘Select Features’.

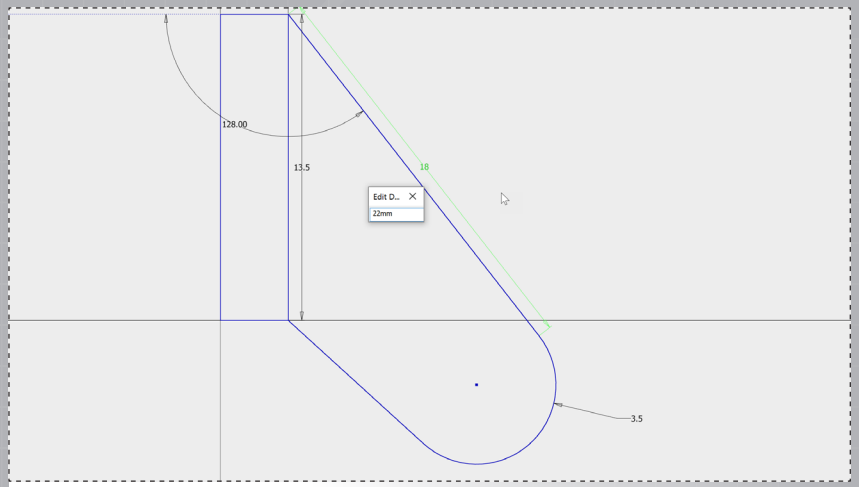
4.



Click anywhere on the flange and select ‘Edit Sketch’ from the pop-up menu.

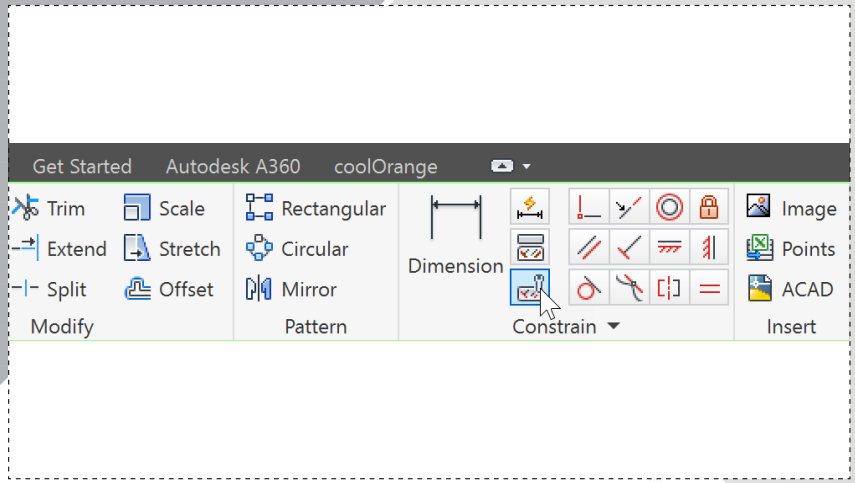
PART 3: EDITING THE MODEL

5.



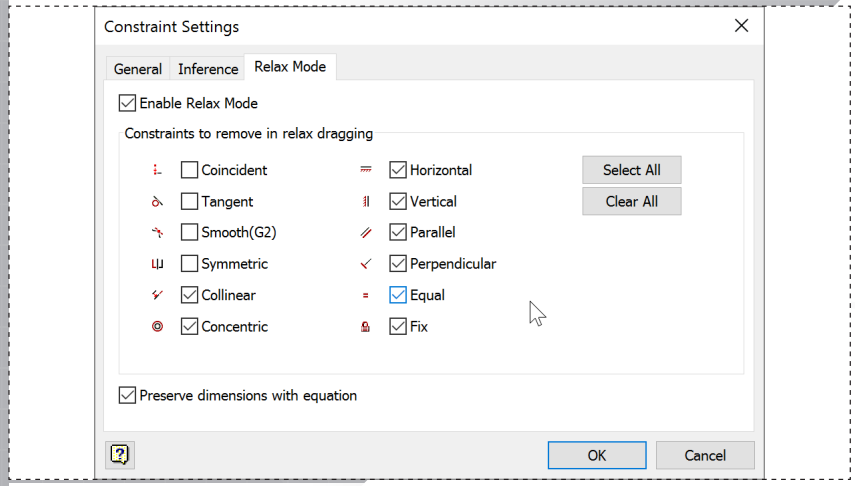
Double click the 18mm dimension and change the value to 22mm in the dimension box. Hit 'Enter'. Right click and select 'Finish 2D Sketch' to update the 3D model.

6.



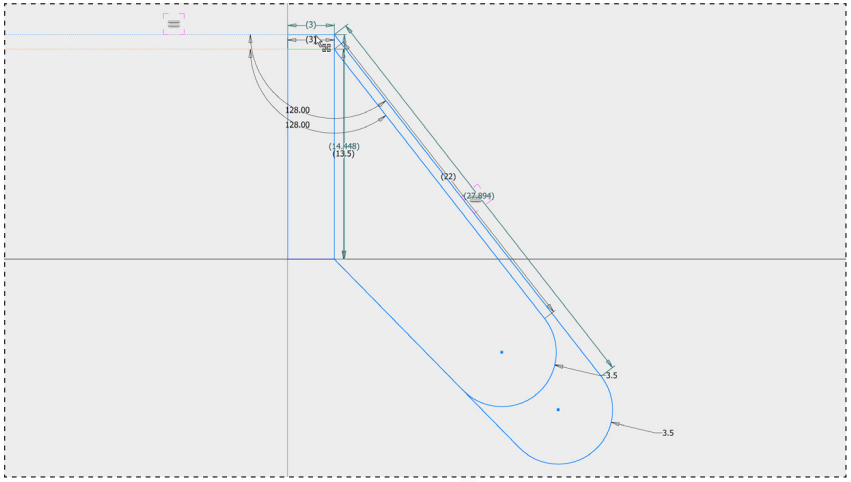
Repeat Step 4 to reopen the sketch. Open the 'Constraint Settings' dialog from the ribbon menu.

7.



In the 'Relax Mode' tab of the dialog, click the box to 'Enable Relax Mode'. This allows you to automatically remove constraints by dragging geometry in the sketch.

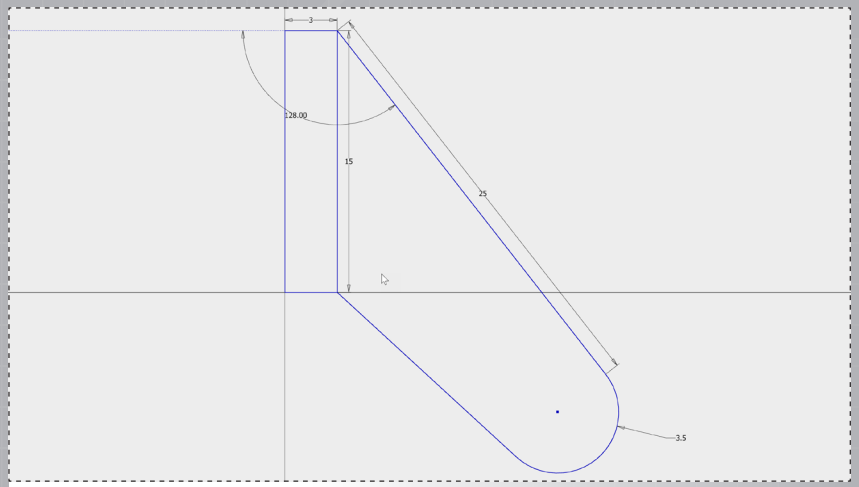
8.



Grab the top horizontal line and drag upwards to the approximate location shown. As you drag, you can see what dimensions and constaints are changing.

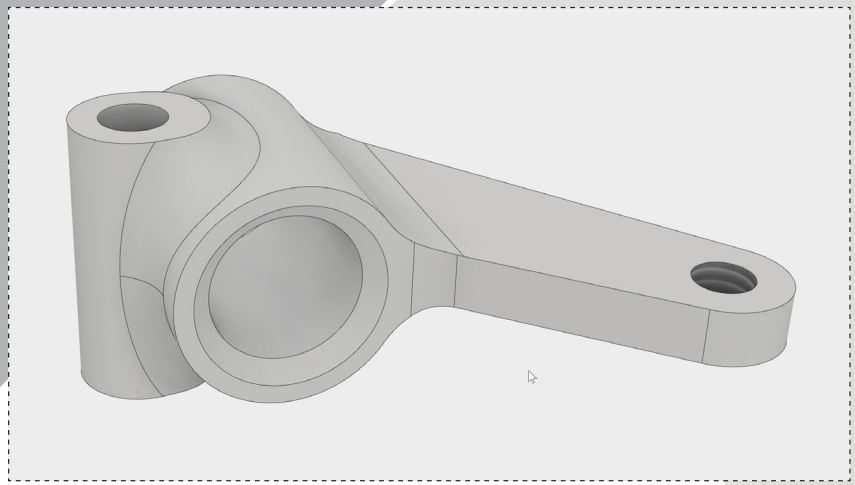
PART 3: EDITING THE MODEL

9.



Double click to change the vertical dimension to 15mm and the aligned dimension for the flange to 25mm.

10.



Right click and select ‘Finish 2D Sketch’ to update the 3D model. The steering knuckle is now complete, remember to save before you close the file.

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