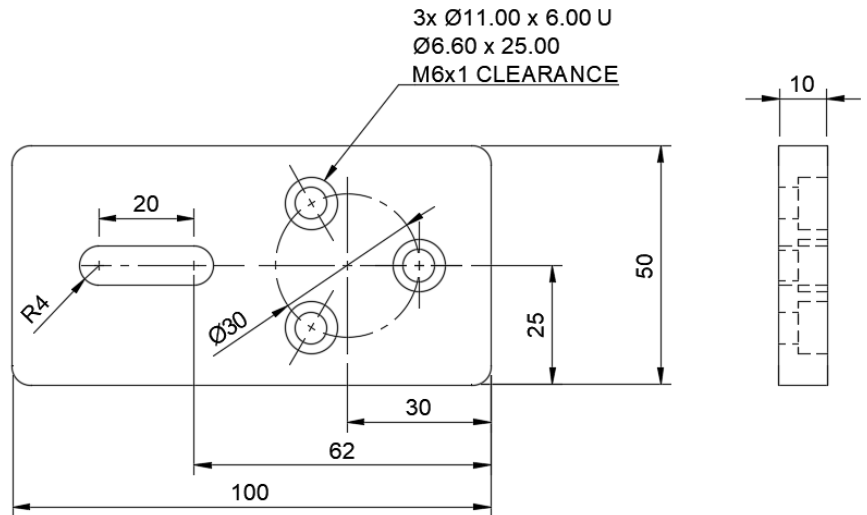
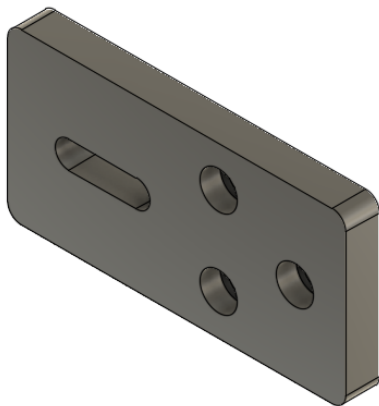


Part 1

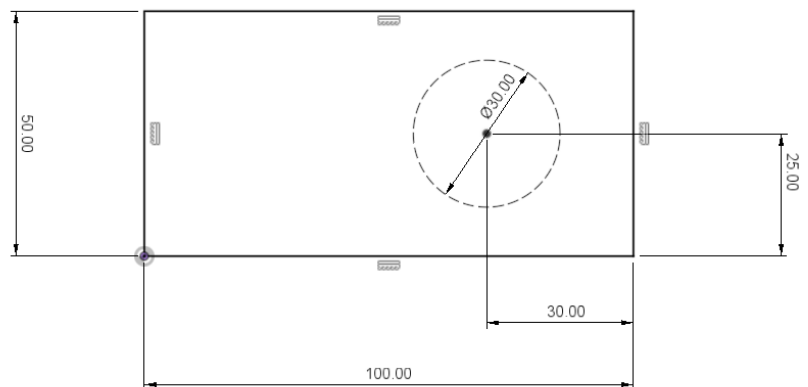


Step 1 –

New component, new sketch (on any plane)

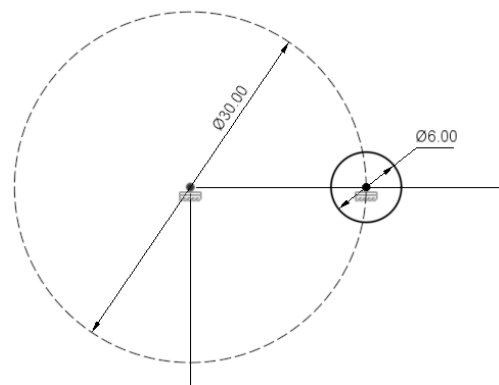
Step 2 –

R for rectangle, draw the outer shape. Dimension it 50x100. Add a centre-diameter circle (C) with a diameter of 30 mm. Make it a construction line. Position it from the bottom right corner as per the drawing.



Step 3 –

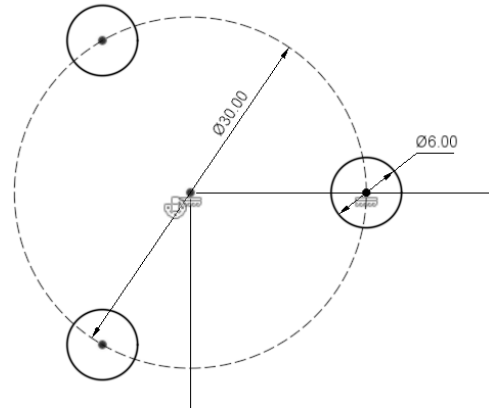
C for circle again, draw a normal 6 mm circle on the perimeter of the construction circle. Using the Horizontal/Vertical constraint tool make sure it is horizontal with the centre of the construction circle.



Step 4 –

Create > Circular Pattern

Select the smaller 6mm circle as the object you wish to pattern, the centre of the construction geometry as the axis/centrepoint. Three copies.

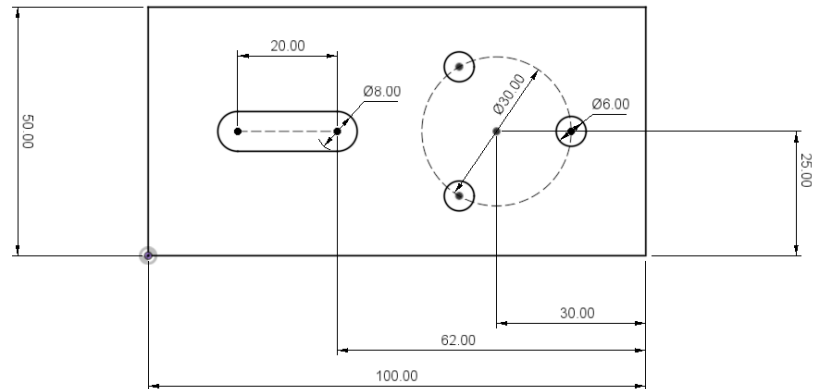


Step 5 –

Draw a Centre-To-Centre slot, 8 mm slot width, 20 mm between centres. Position it as per the drawing using the dimension tool.

Make the centre of the right hand slot Horizontally constrained to the centre of the construction circle.

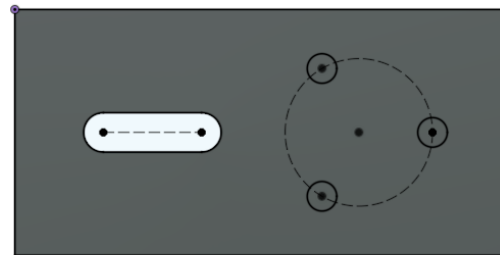
Finish the sketch.



Step 6 –

Extrude the entire body, including the holes but not the slot, to a depth of 10 mm.

Go into the browser tree, turn on the sketch visibility.



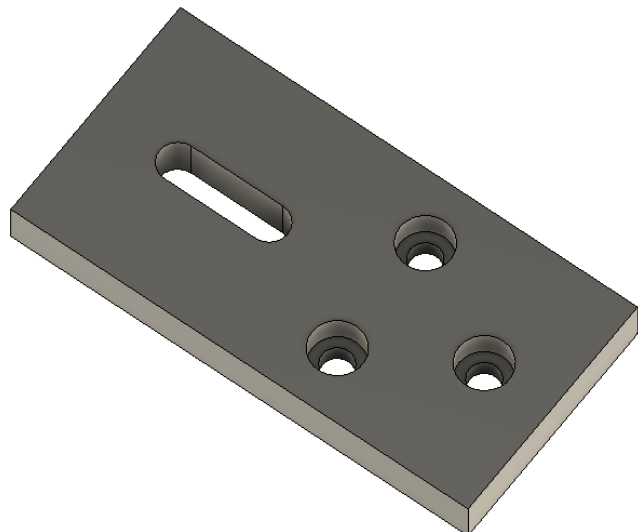
Step 7 –

Create > Hole (H)

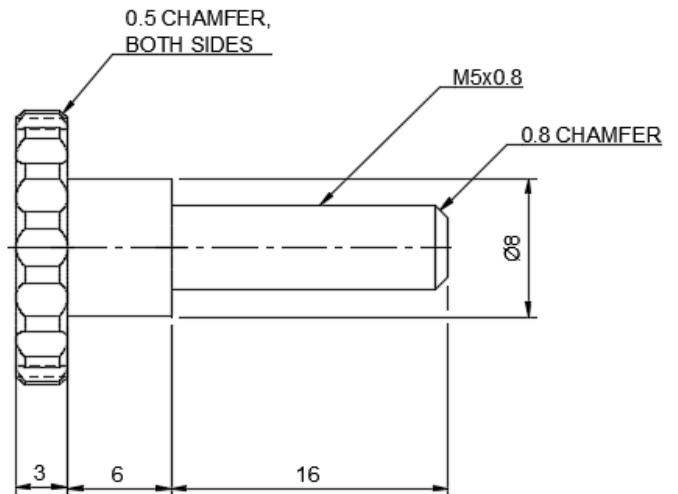
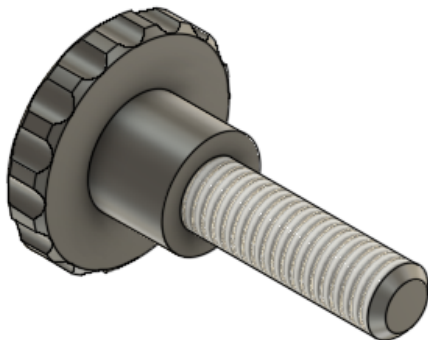
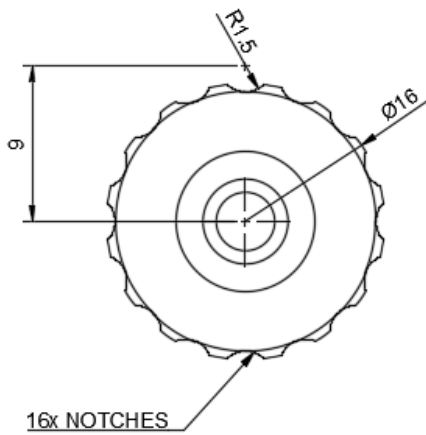
Select each of the three holes on the PCD pattern.

Extents – All (FLIP DIRECTION if needed)
Hole type – counterbore
Tap type – clearance
Fastener Type – Socket Head Cap Screw
Designation – M6
Fit – Normal

Turn off sketch visibility. Finished.



Part 2

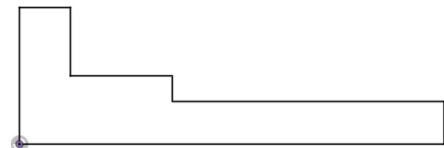


Step 1 –

New Component, New Sketch (on any plane)

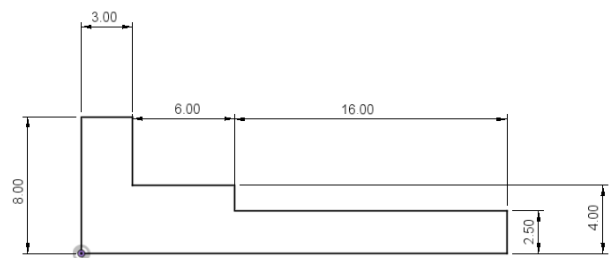
Step 2 –

Draw to top half of the rough profile using the line tool. Start anchored to the origin. Make sure you are keeping things square by letting it snap to perpendicular or using the Horizontal/Vertical constraint tool.



Step 3 –

Start adding dimensions from the drawing. Bear in mind that we are drawing half the profile so we can revolve it, so some dimensions will be halved (turned to the Radius from the Diameter). Finish sketch once you are done.



Step 4 –

Create menu, revolve, select the profile you just drew. The axis is the long centreline. Admire your 3D creation.



Step 5 –

Create a chamfer either side of the top flange, 0.5mm per the drawing.

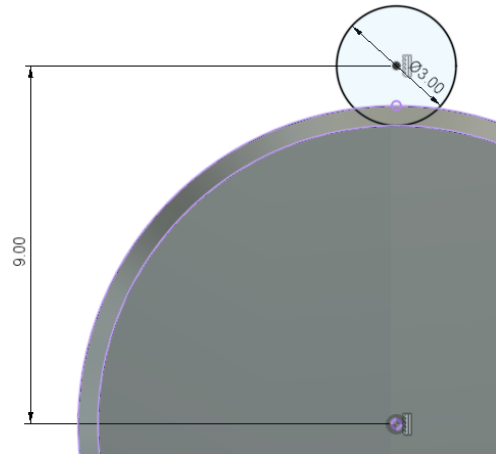


Step 6 –

Create a new sketch, on the large flat surface (the “top” of our thumbscrew) to the left hand side of our part.

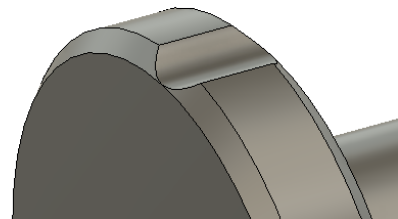
Draw a circle, vertically above the origin point in the centre. Dimension this circle 3mm diameter, constrain it vertically above the origin using Horizontal/Vertical constraint tool. Position it 9mm from the centre.

Finish Sketch.



Step 7 –

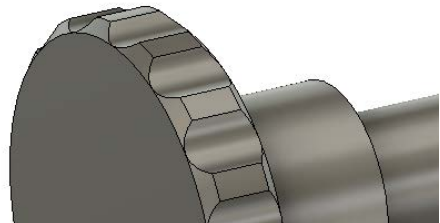
Extrude the entire circular profile through the body using the Cut mode. You can use the “All” distance, or set a specific depth.



Step 8 –

Create > Pattern > Circular Pattern

Use Features Type, select the last extrude you did on the timeline. Axis is the outer cylindrical face of your part. 16 qty, in “identical” compute mode.



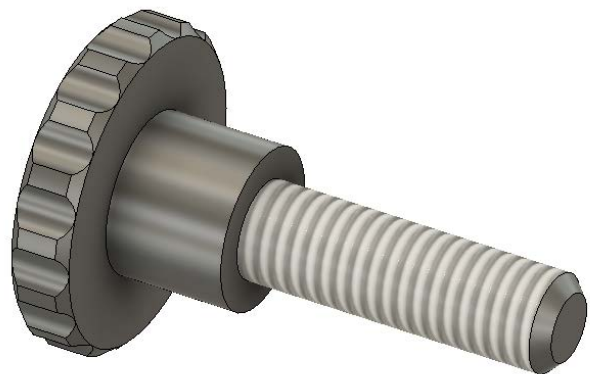
Step 9 –

Create > Thread

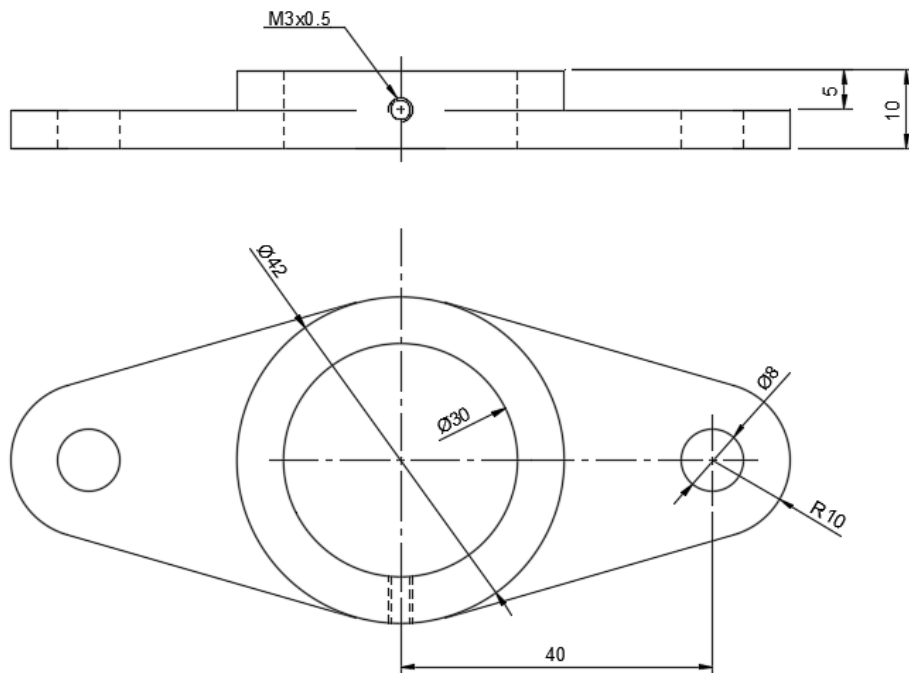
M5x0.8 thread on the section indicated on the drawing. Not modelled.

Add a 0.8mm chamfer on the end of the threaded section.

Finished. If 3D printing the thread (it's a little small) you can change this to modelled AFTER the chamfer has been added.)



Part 3

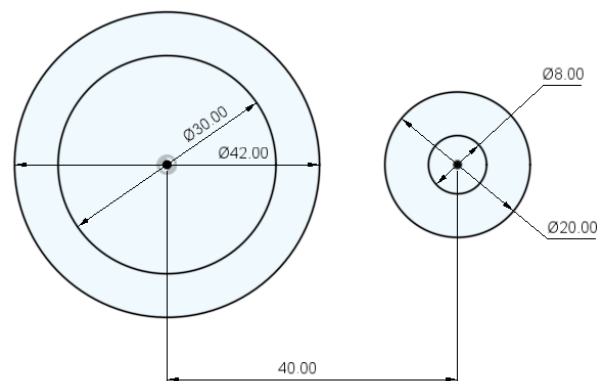


Step 1 –

New component, new sketch.

Step 2 –

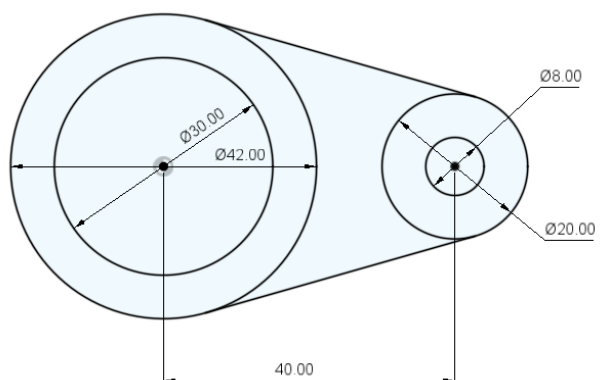
Draw two concentric circles on the origin, diameter 30 mm and 42mm. Horizontally aligned, 40 mm to the right draw two more concentric circles, diameter 8 mm and 20 mm.



Step 3 –

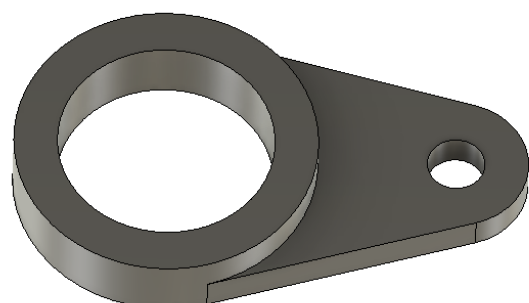
L for line tool, click and drag a tangent line from the 20mm circle to the 42mm circle. Repeat at the bottom.

Finish the sketch.



Step 4 –

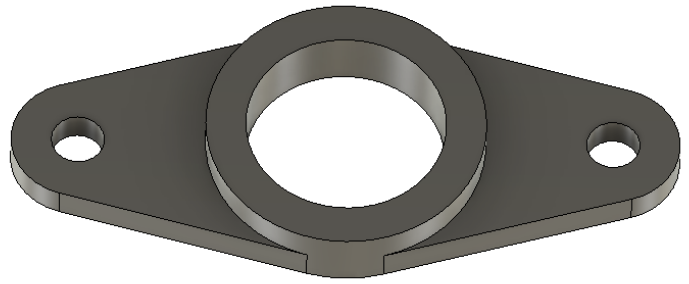
Extrude the central boss by 10 mm. Turn the visibility of the sketch back on and extrude the outer flange by 5 mm.



Step 5 –

Create > Pattern > Circular Pattern

Use Features Type, select the last extrude you did on the timeline. Axis is the inner cylindrical face of the large boss. 2 qty, in “identical” compute mode.



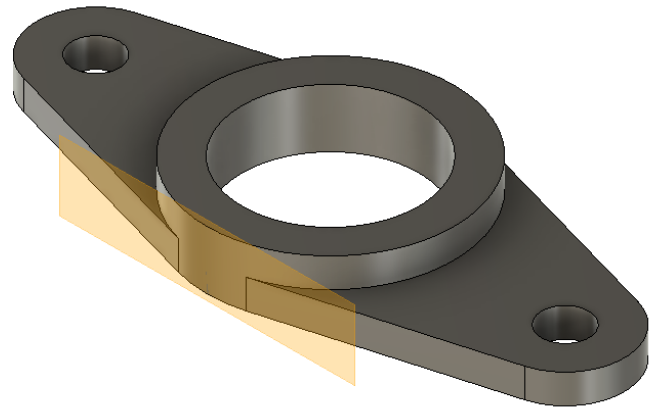
Step 6 –

Construct > Tangent Plane

Click on the outside face of the central boss.

Turn on the visibility of the Origin in the browser tree and set the reference plane to the appropriate origin plane.

Hide the Origin again when done.



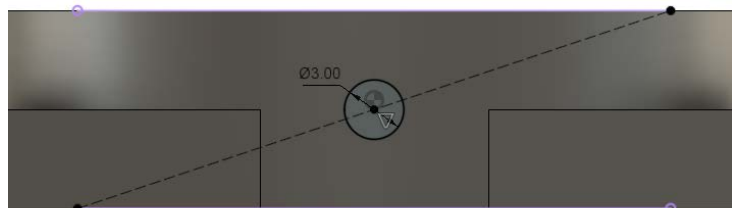
Step 7 –

Create a new sketch on the tangent plane.

P for Project. Project the inner face of the bore by clicking on it.

Draw a construction line from one corner to the other.

Draw a 3mm circle and constrain it to the midpoint of the construction line. Finish sketch.



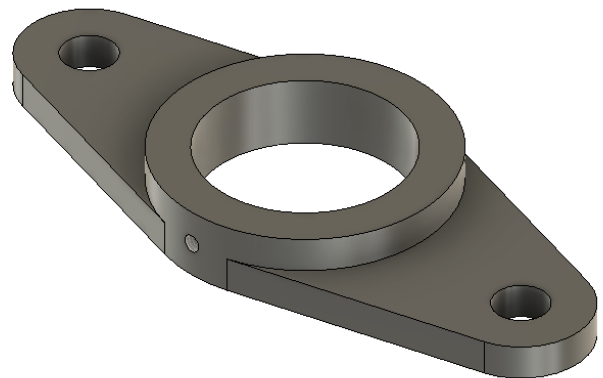
Step 8 –

Create > Hole (H)

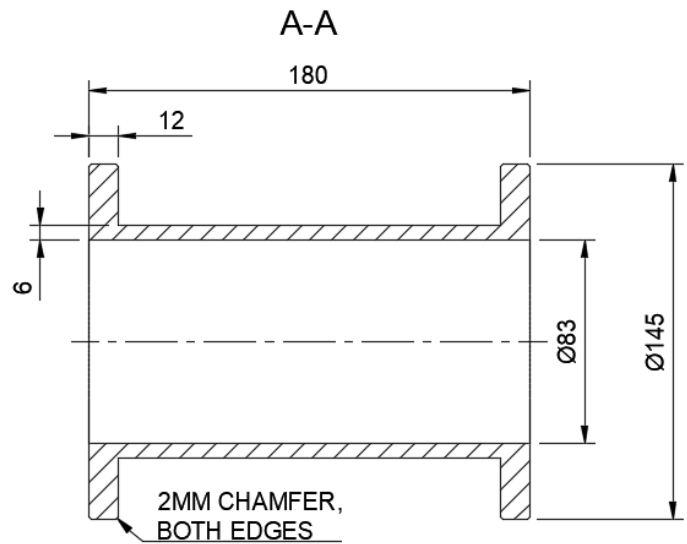
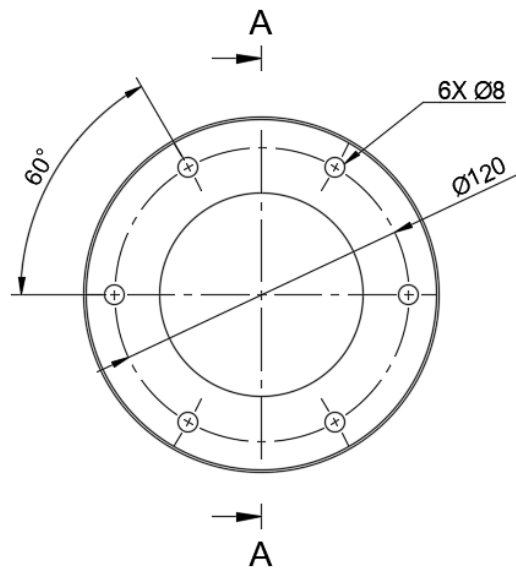
Select the 3mm circle we just drew.

Extents - To, select the inner bore.
Simple hole, tapped, Size 3mm, Designation M3.

Finished.



Part 4



Step 1 –

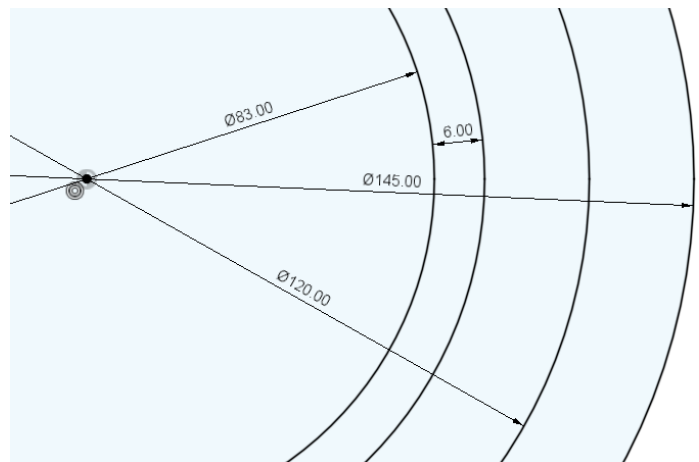
New component, new sketch.

Step 2 –

Draw three concentric circles. One diameter 83 mm, one 145 mm and one 120 mm.

Add one more circle which is 6 mm larger in radius than the 83 mm one.

Make the 120mm circle a construction line.

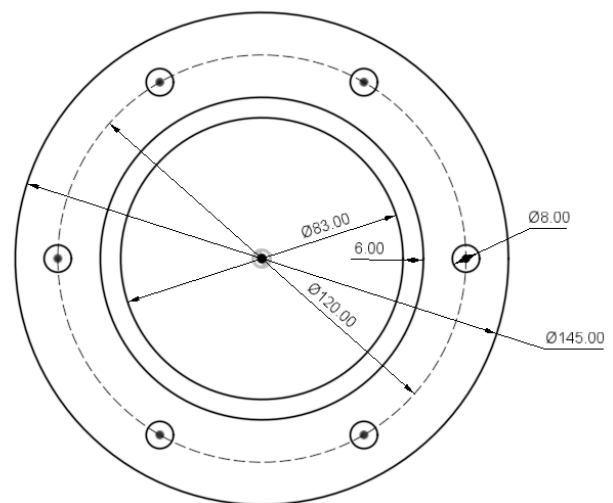


Step 3 –

On the 120 mm construction line circle place an 8mm diameter circle, horizontally constrained to the centre point of the circles.

Create > Circular Pattern

Select the 8mm circle as the object you wish to pattern, the centre circles as the axis/centrepoint. Six copies.



Step 4 –

Finish Sketch.

Extrude the central ring (the pipe wall) 180mm.

Turn the visibility of the sketch back on, then extrude the flange portion by 12 mm in the same direction. Make sure the “join” mode is used.

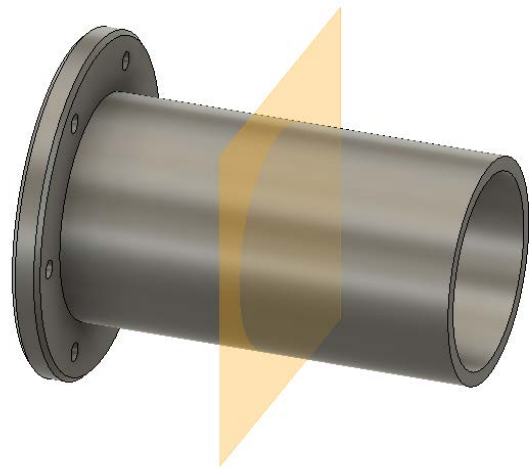
Modify > Chamfer

Add a 2 mm chamfer on the front and back edge of the flange.

Step 5 –

Construct > Mid Plane

Select the front face of the flange, the back face of the pipe.



Step 6 –

Create > Mirror

Type – Features

Select the chamfer and the second extrude (which created the flange) from the timeline as the objects you want to mirror.

Select the mid-plane construction plane as the mirror plane. Identical compute mode.

Hide the construction planes in the browser tree.

Finished.



Extra Drawings

<https://blog.naver.com/studycadcam/222016536309>

<https://blog.naver.com/studycadcam/222108693689>

<https://blog.naver.com/studycadcam/221579788456>

<https://blog.naver.com/studycadcam/221552388350>

<https://blog.naver.com/studycadcam/221552223715>