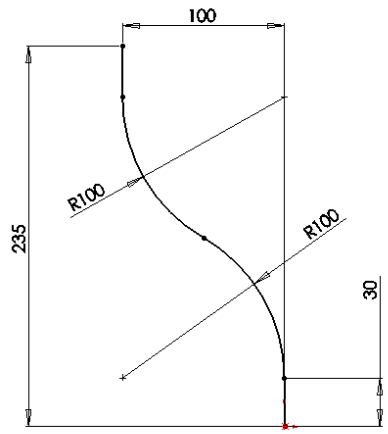
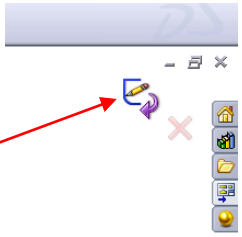
Pipe Adaptor 2 into 1 – Sweep and Mirror



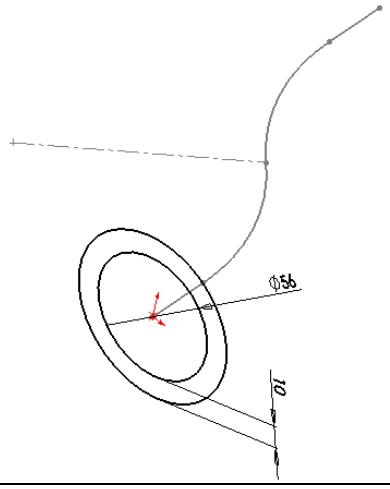
Sketch on to top plane.

Remember relations – you may need to add tangency relations between sketch entities.



Close the Sketch.

This sketch will become the path for the sweep feature.

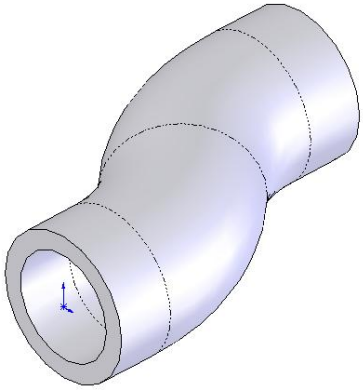


Insert a new sketch for the pipe diameter. Inner bore Ø 56 mm, wall thickness 10 mm (offset sketch tool useful here).

This will become the profile for the sweep feature.

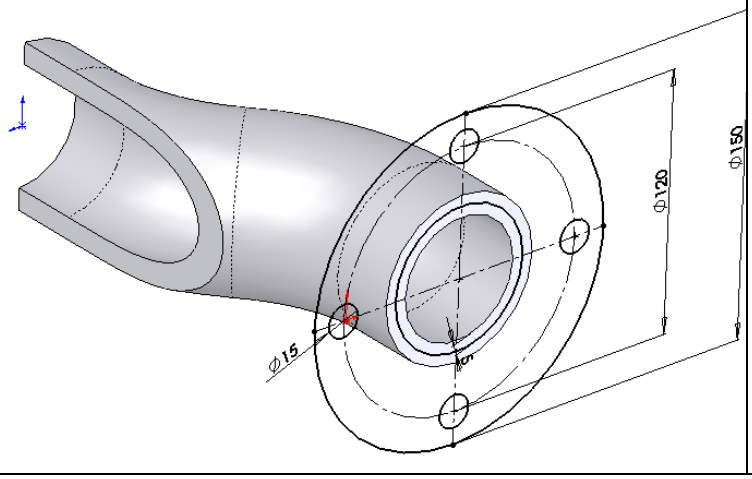
This sketch must also be closed, as above.

The sweep feature can not be started when inside an active sketch.



Remember when you sweep, sketch 2 is the profile and sketch 1 is the path. They can be selected in the graphics window.

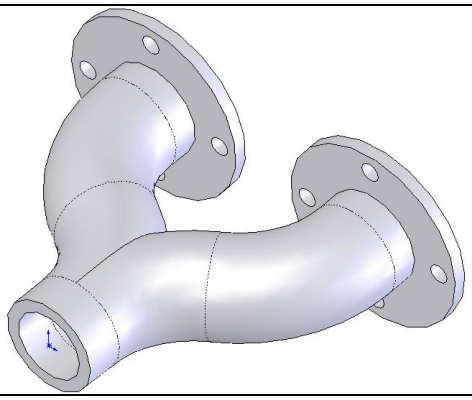
Your Sweep feature should look like this.



Select the RIGHT plane, then using the drop down menu's,

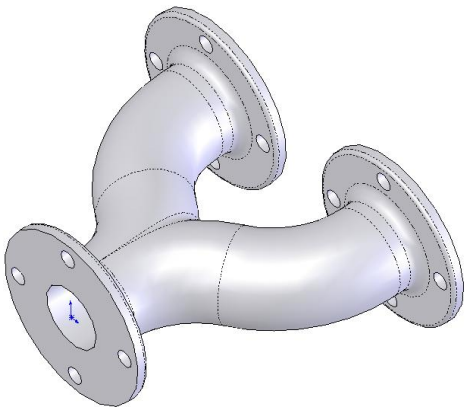
INSERT, CUT, WITH SURFACE.

Then select the end face, insert a new sketch; sketch the shape to produce the end flange. Use the Extrude feature, 14 mm thick flange.



Select the RIGHT plane or the CUT FACE, Select the mirror feature tool, select the body in the correct property manager selection box.

Mirror the body. The result should look like this.



On the single outlet face insert a sketch produce the end flange (identical to other ends), use the extrude feature.

Produce the fillets, 3 mm on the outer edge of the flange, and 10 mm on the three flange to pipe body corners and also on the corner where the two sweep features meet.

Add the cosmetic threads to the 15 mm holes.