

Solid Parts

Revolved Boss/Base

This option is used to revolve a sketch around an axis.

For example

- ⇒ Sketch figure (1)
- ⇒ Select Revolved Boss/Base from the features toolbar
- ⇒ Select the vertical line as the axis of revolution
- ⇒ Click OK

The result should be as figure (2)

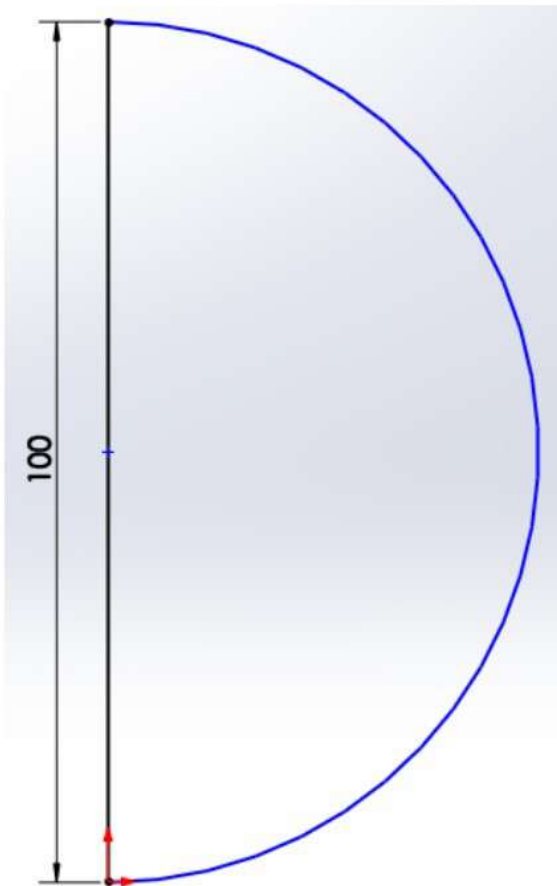


Figure (1)

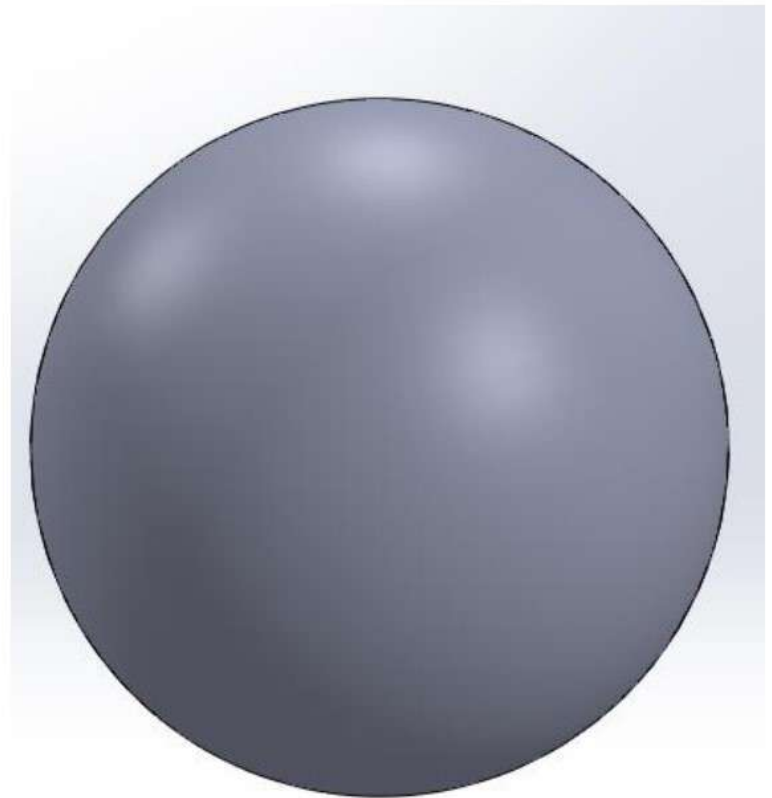
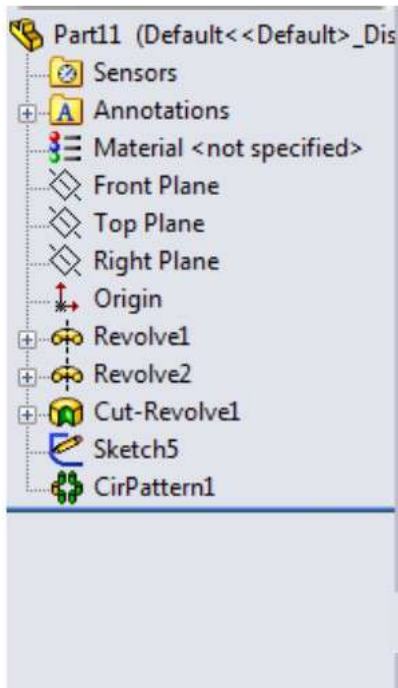
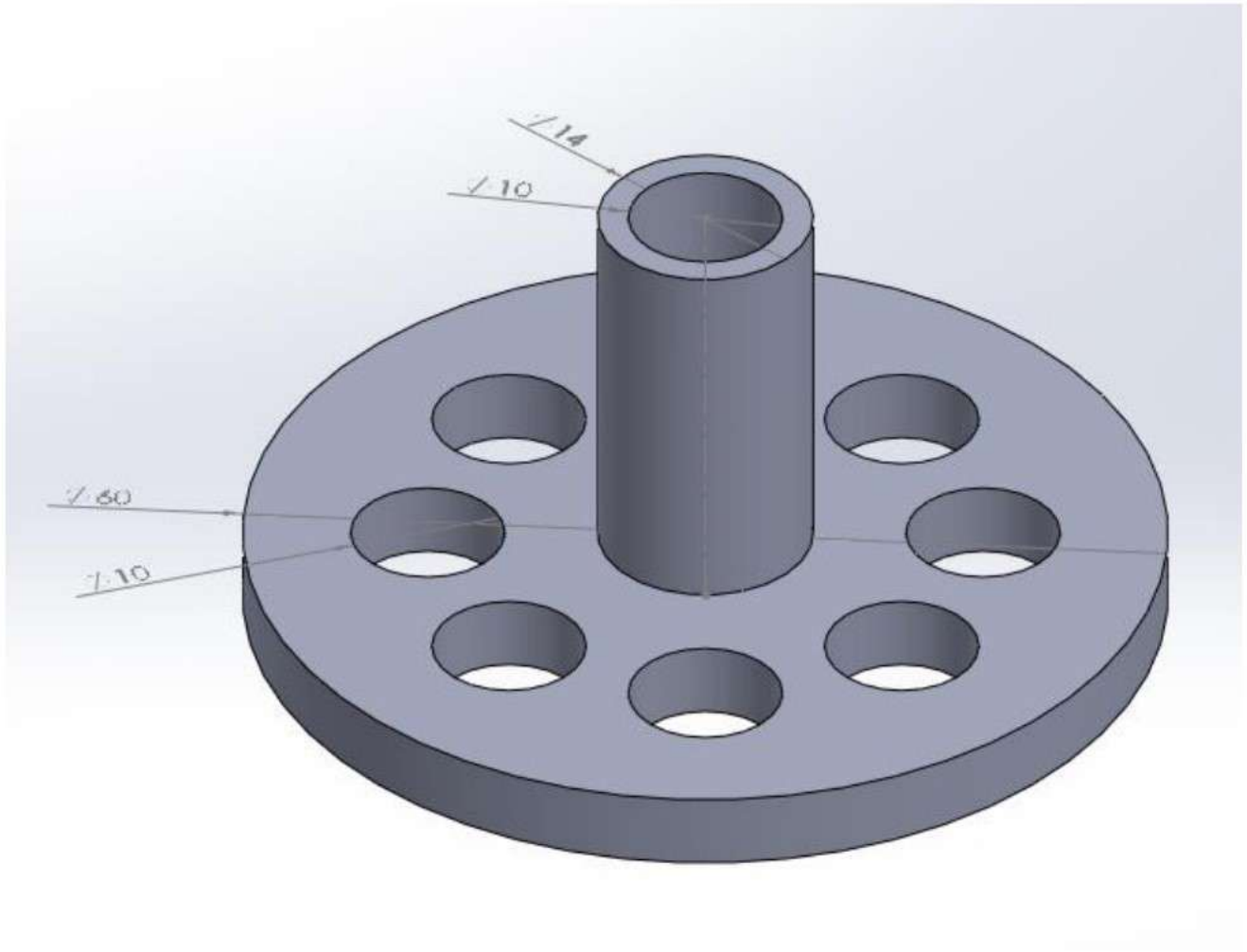


Figure (2)

HW

Use the design tree as reference to help you



Swept Boss/Base

This feature is used to extrude sketch on a path that you choose

Note that the feature Extruded gives you the possibility to extrude the sketch over the default(x, y, z) directions in a straight line, while the feature swept gives you the possibility to extrude the sketch over any path you have already sketch.

For example we are going to create the geometry shown in figure (3)

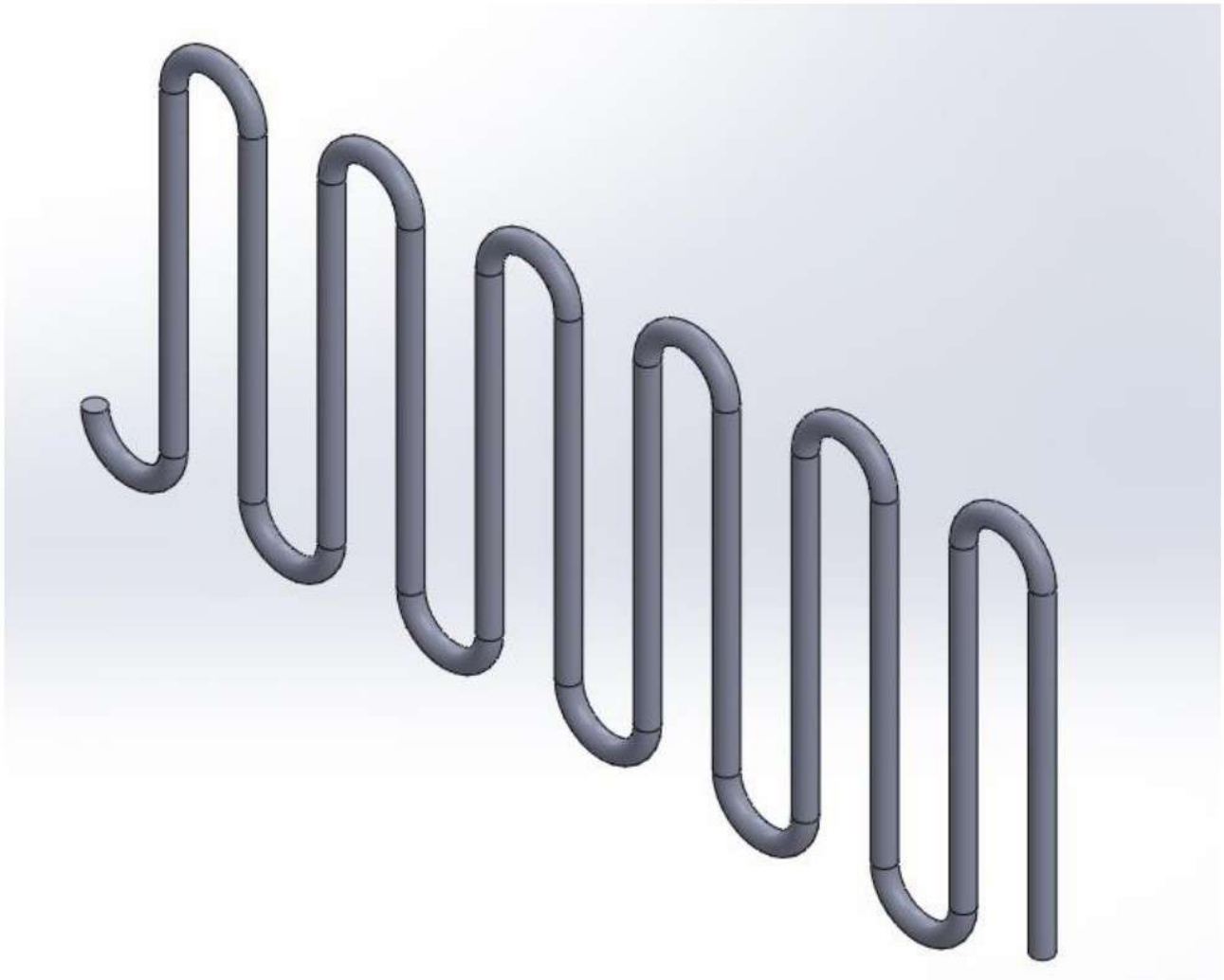


Figure (3)

- ⇒ On the front plane make the sketch shown in figure (4)
- ⇒ Select linear pattern from the sketch toolbar
- ⇒ Make a linear pattern as shown in figure (5)
- ⇒ Exit the sketch (it's important here to exit the sketch to activate the feature swept, and also to make different sketch on different plane)

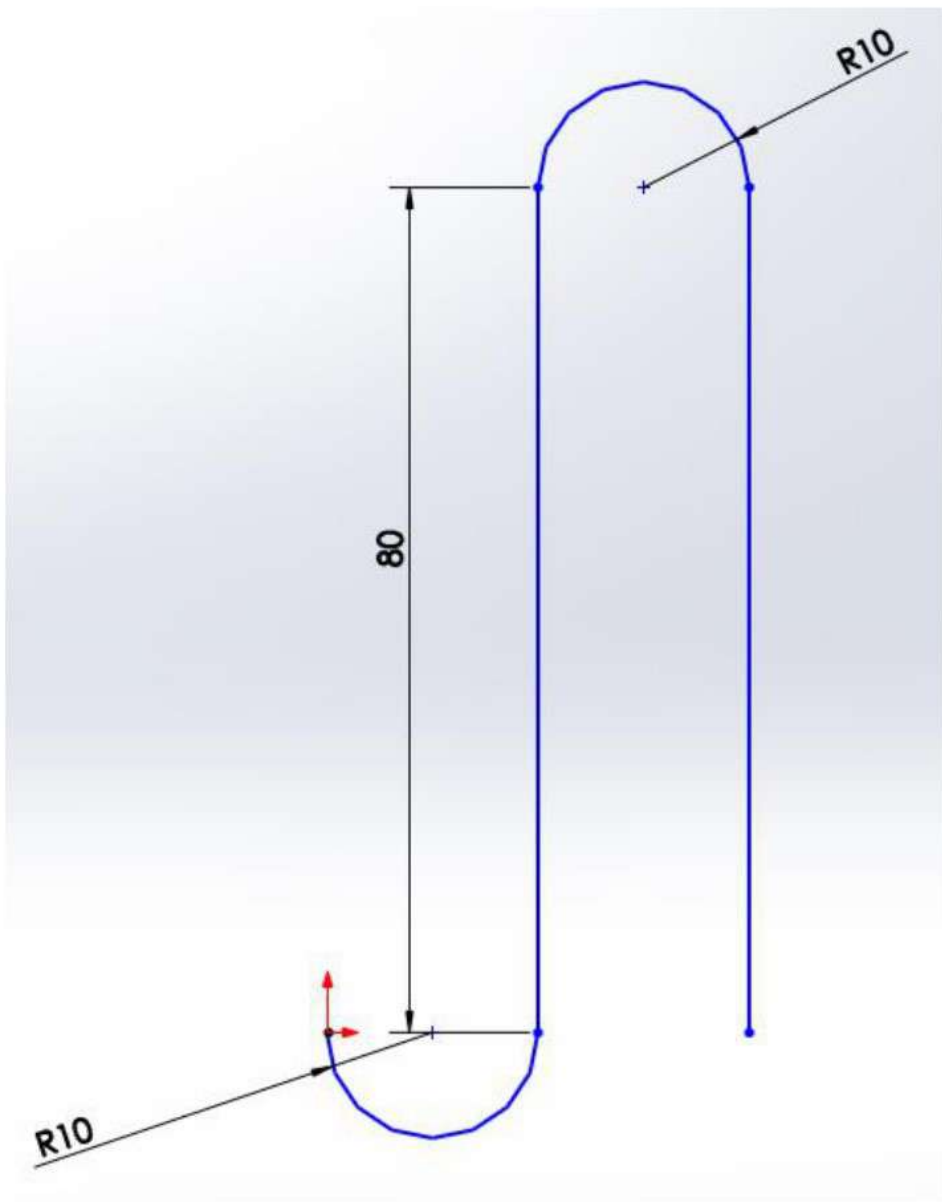


Figure (4)

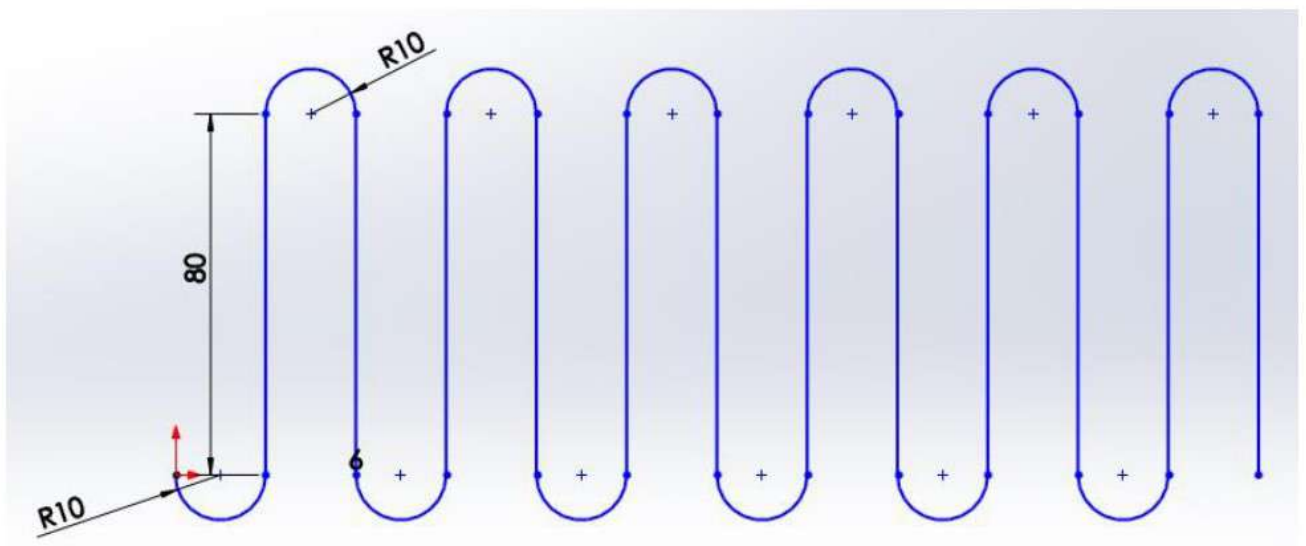


Figure (5)

- ⇒ On the top plane create the sketch shown in figure (6)
- ⇒ Make sure that the center of the circle is on the end of the path line
- ⇒ Exit the sketch

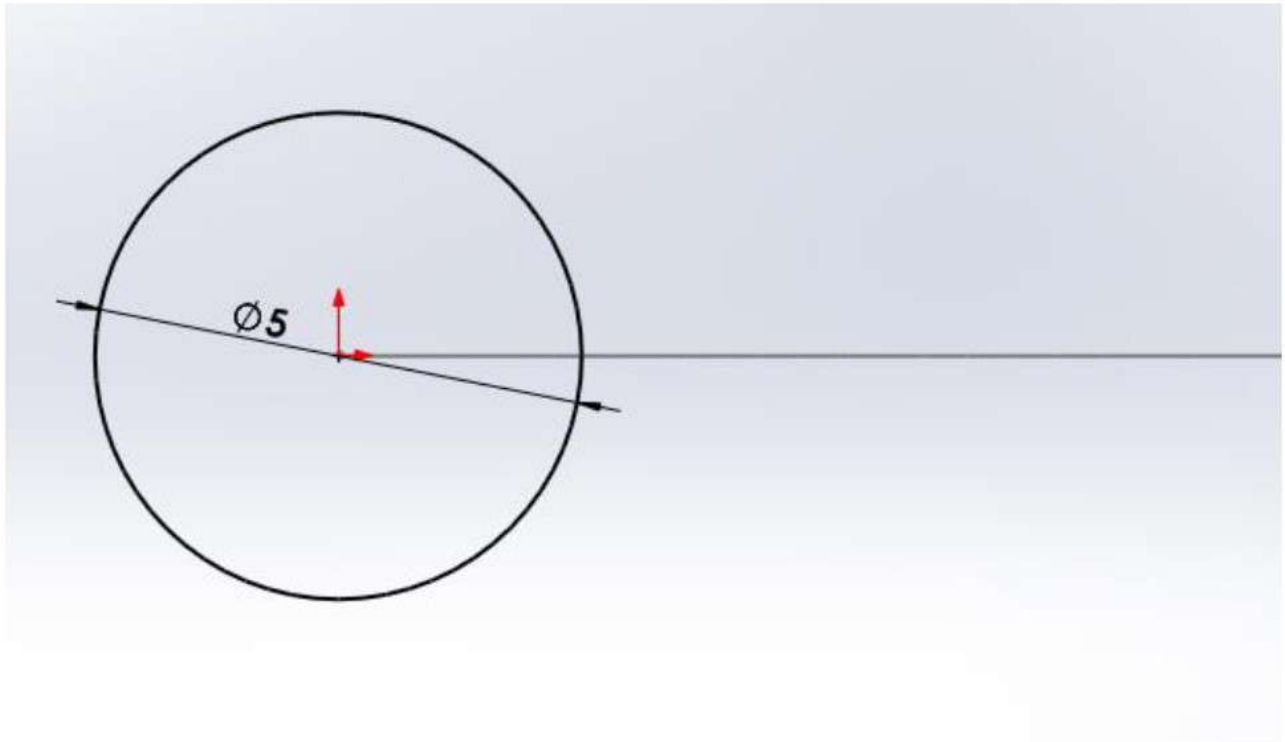


Figure (6)

- ⇒ From the feature toolbar select Swept Boss/Base
- ⇒ Select the curved line as the path, figure (7)
- ⇒ Select the circle as the profile, figure (7)
- ⇒ Click OK

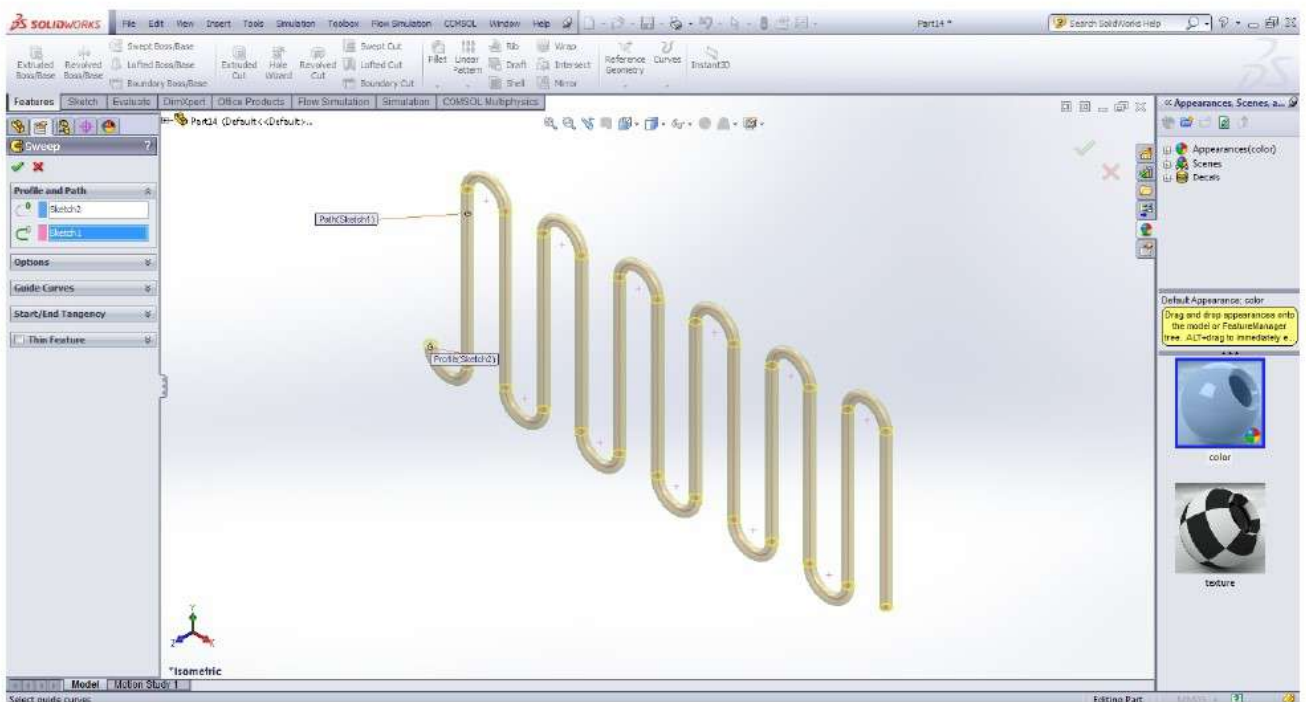


Figure (7)

HW

Use the design tree as reference to help you

