Solid Parts

SolidWorks program gives a third dimension to the sketches to make it solid three dimensional parts.

In order to have a solid part you have to use the Features Toolbar.

Features Toolbar

The Features Toolbar are shown in figure (1). It have different option to make your sketch a solid part, in these lectures we will explain some of these options.

Figure (1)

Extruded Boss/Base

This option allows you to give the sketch an extruded third dimension. To understand it better we are going to create simple solid part shown in figure (2)

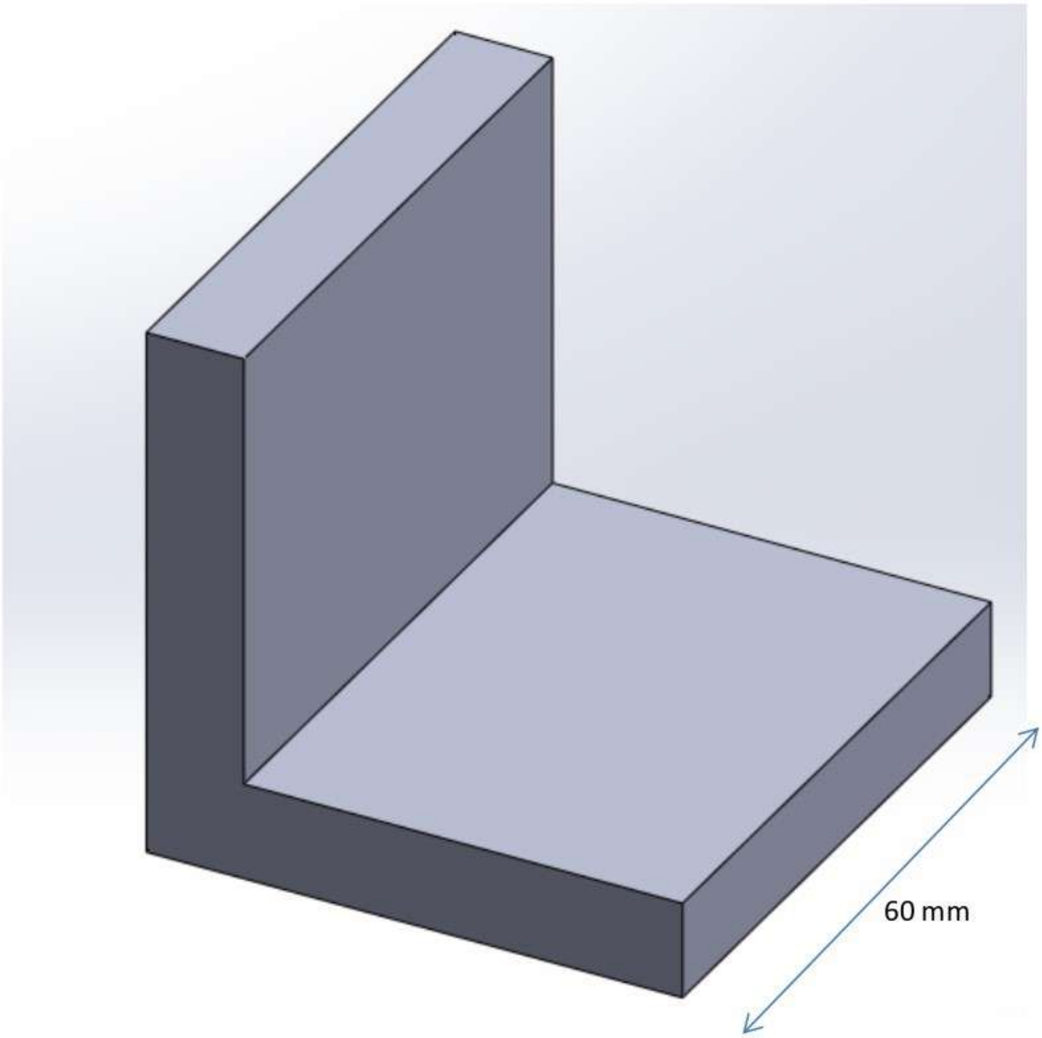

Figure (2)

- ⇨ First step Select front plane
- ⇨ Create the sketch shown in figure (3)
- ⇨ Select Extruded Boss/Base from the features toolbar
- ⇨ Set the third dimension
- ⇨ Click OK

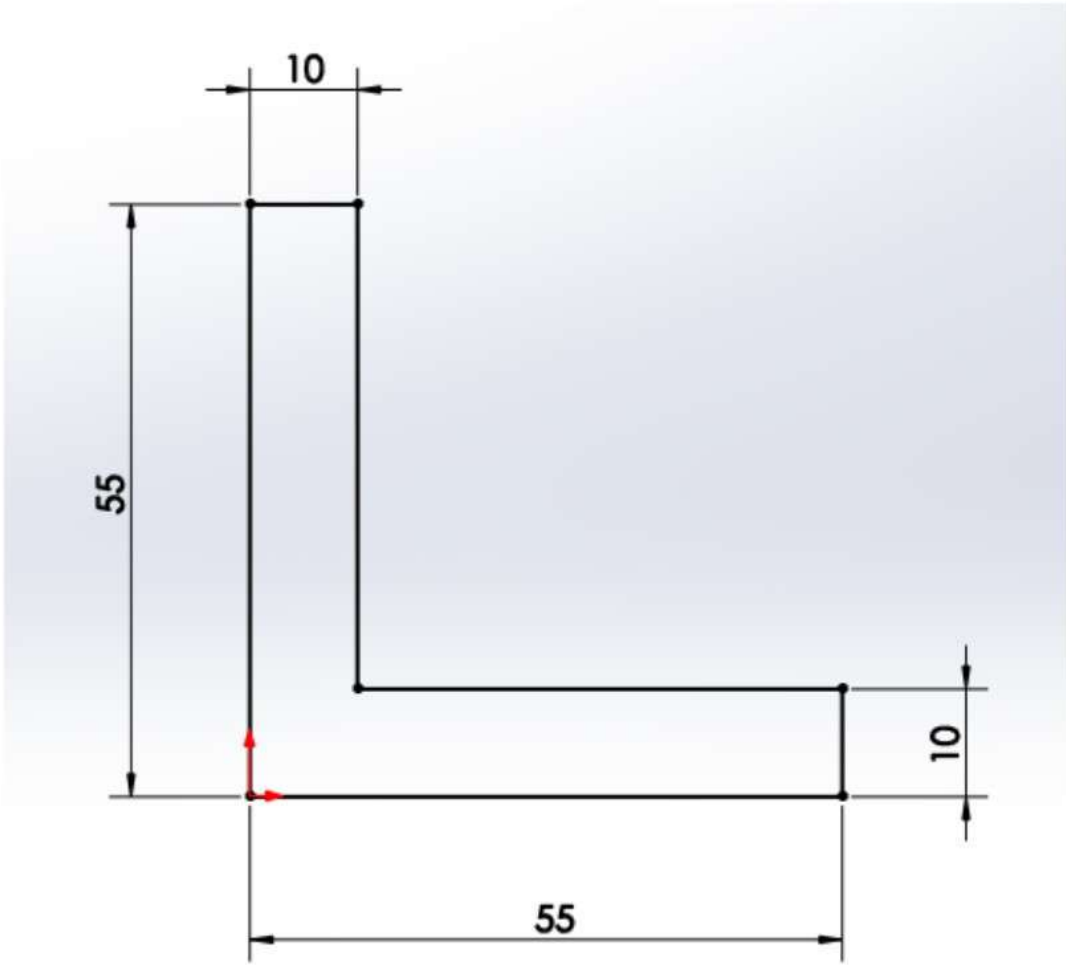

Figure (3)

Extruded Cut

This option gives you the ability to cut a specific sketch from the solid part you created. For example we will cut part of the solid part that was created earlier, as shown in figure (4).

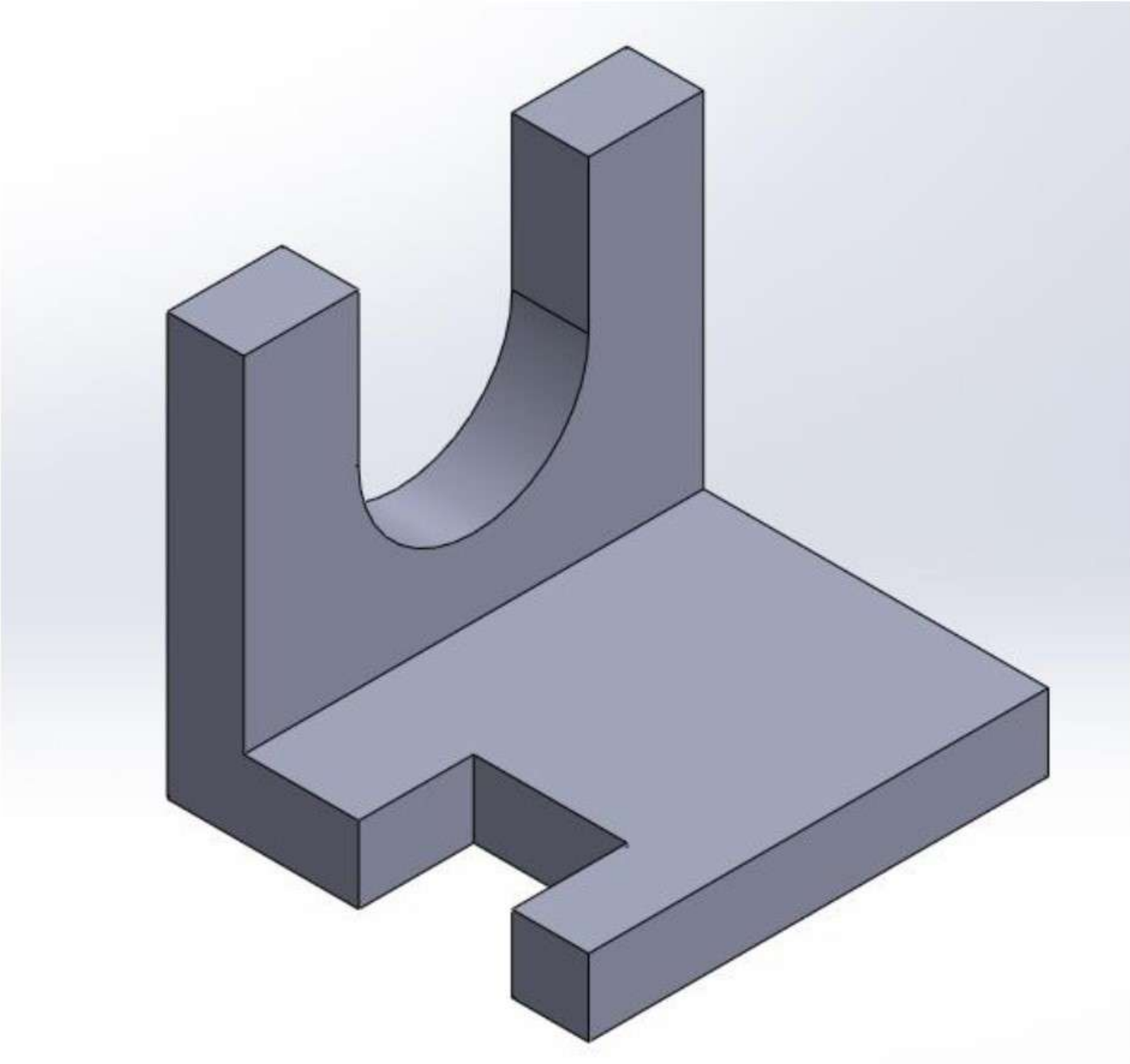

Figure (4)

- ⇨ Start by clicking on the face that you want to create the sketch on as shown in figure (5)
- ⇨ Make the sketch shown on figure (6)
- ⇨ Select Extruded Cut from the features toolbar
- ⇨ Set the direction as through all as shown in figure (7)
- ⇨ Click OK

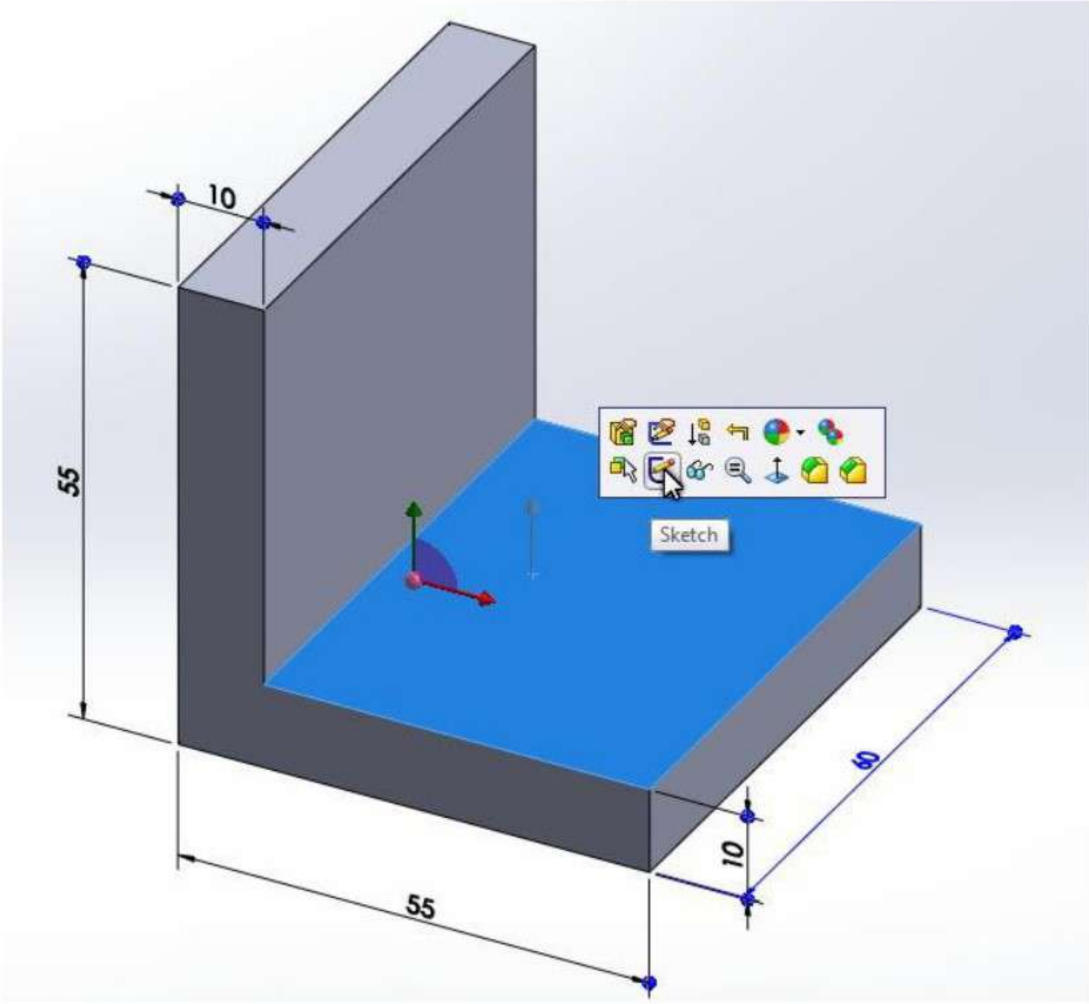

Figure (5)

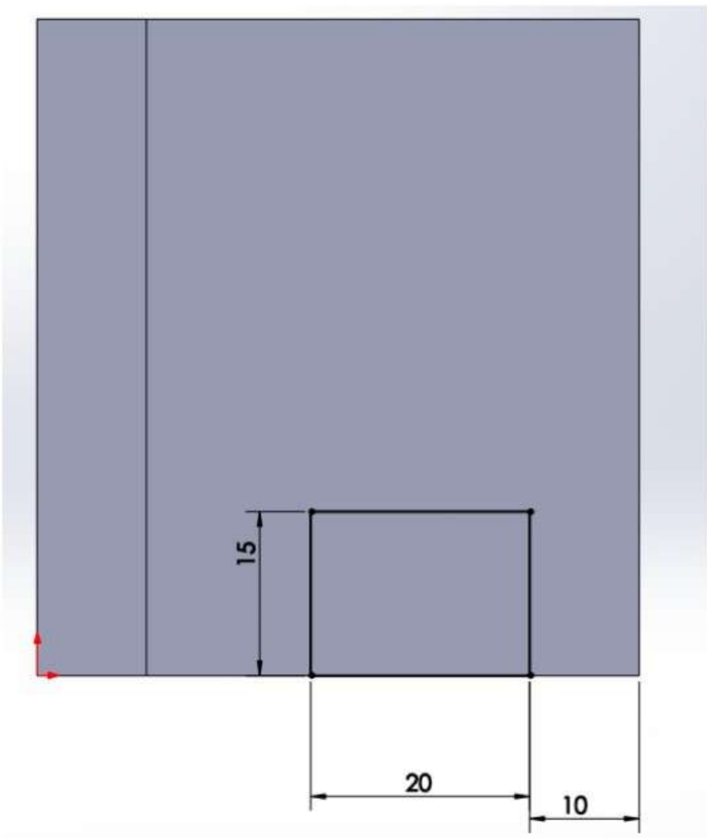

Figure (6)

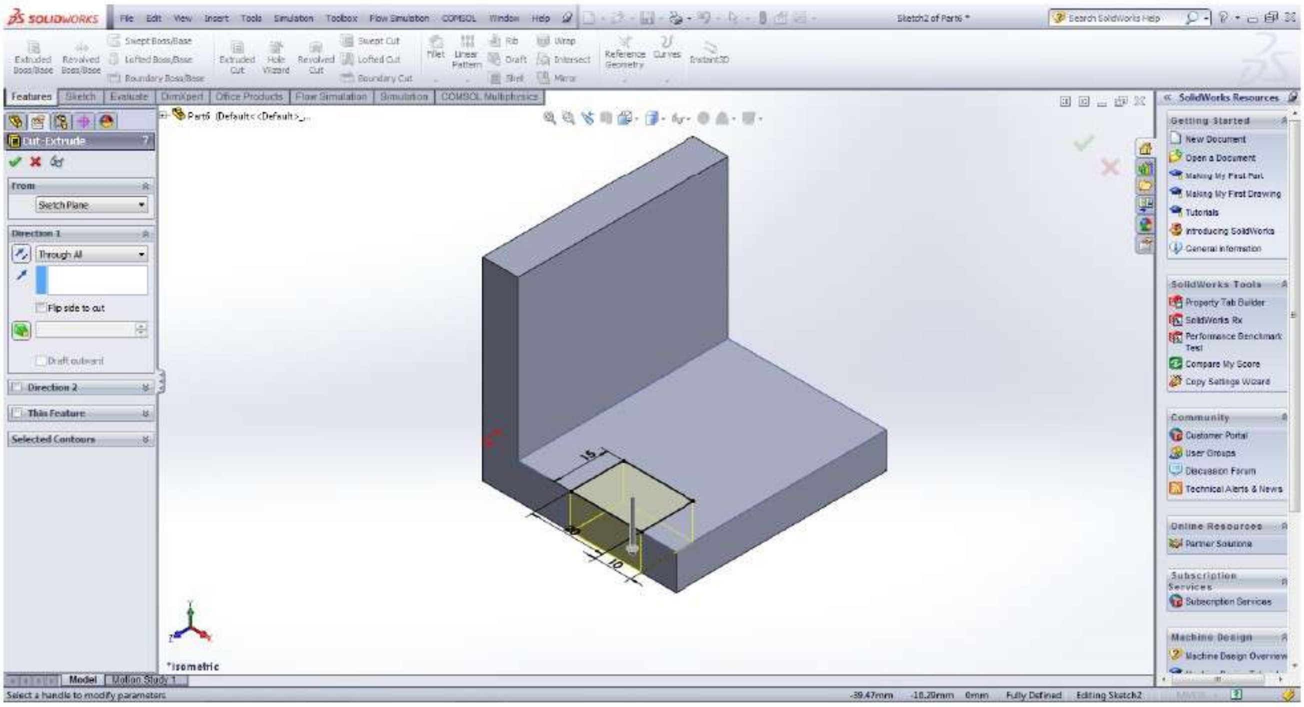

Figure (7)

⇨ Repeat the previous steps to create the other cut as shown in figures (8)

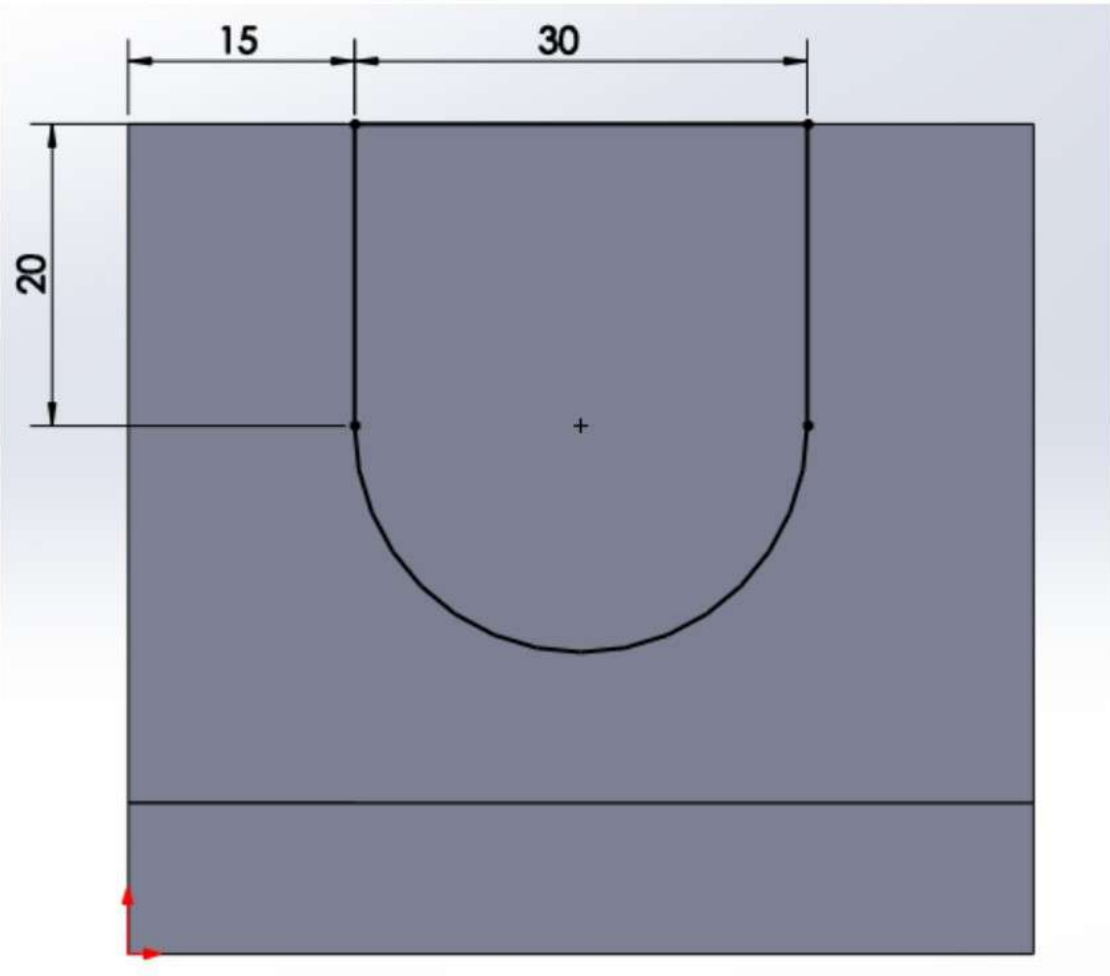

Figure (8)

Another example on the option Extruded Boss /Base and Extruded Cut are shown in figure (9)

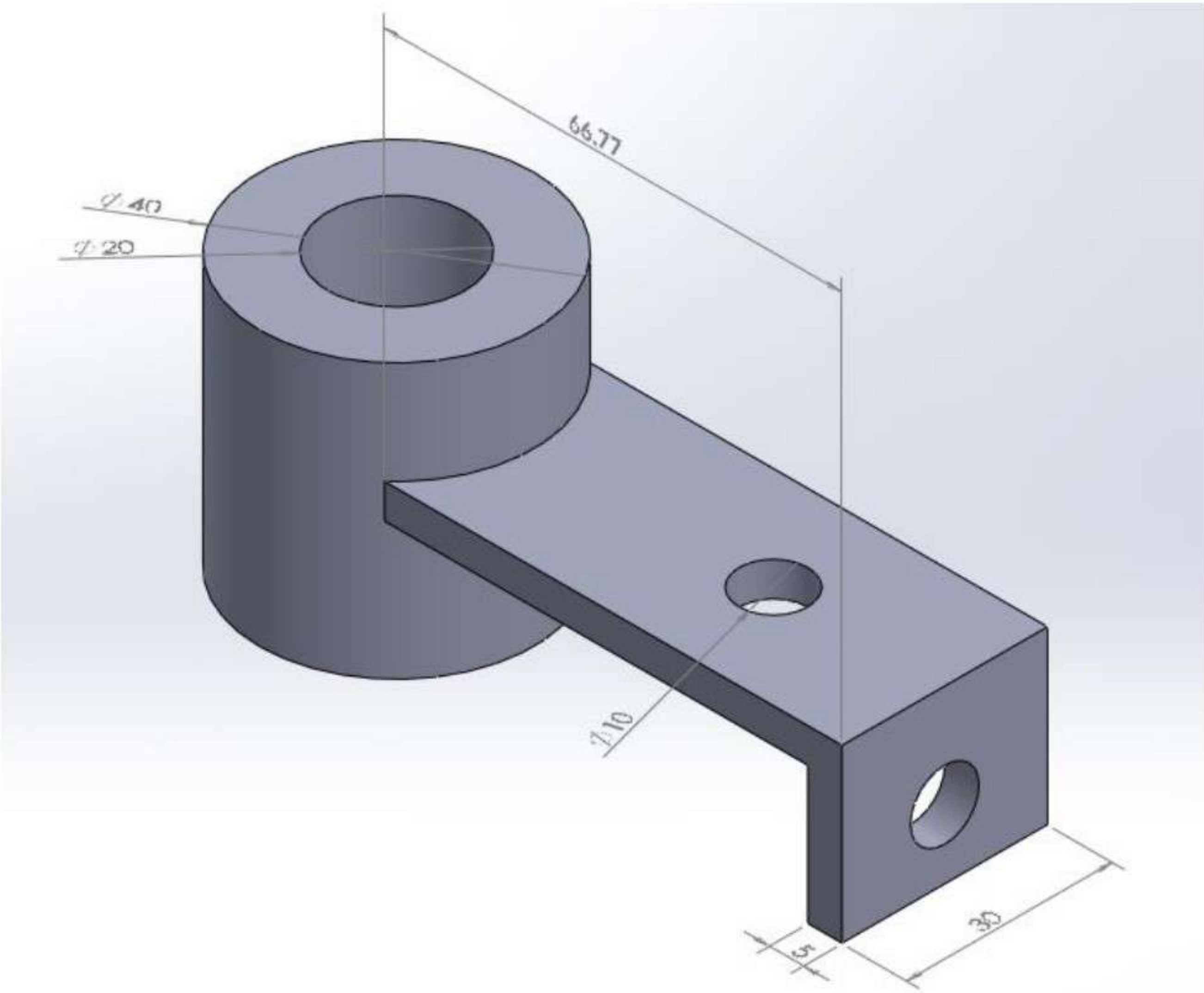

Figure (9)

⇨ On the top plane create the sketch shown in figure (10)

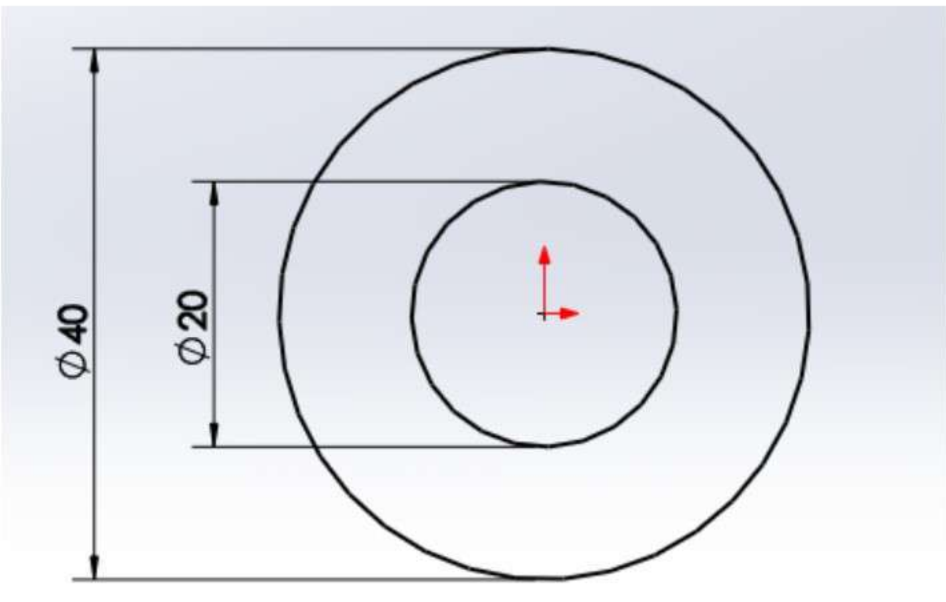

Figure (10)

⇨ Select Extruded

⇨ Set the depth as 40 mm

Note that SolidWorks gives you the ability to create other than the default planes on the original point

⇨ Select reference plane from features toolbar as shown in figure (11)

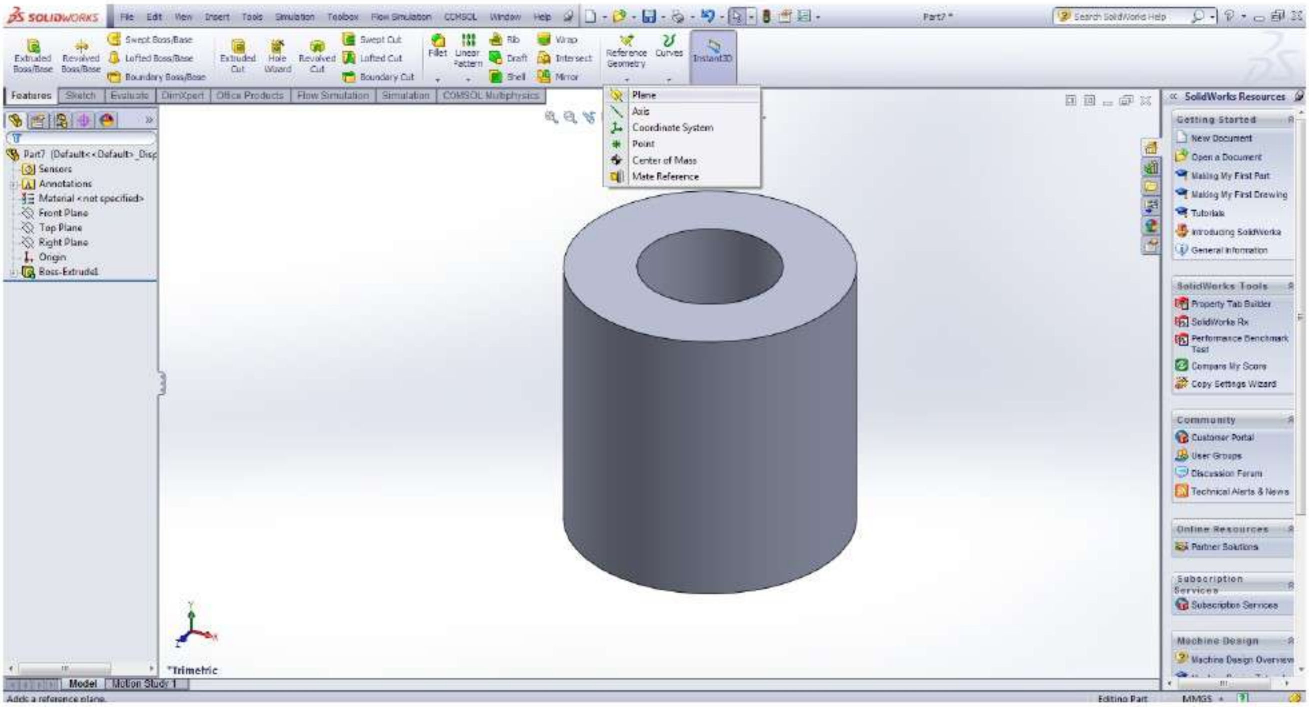

Figure (11)

⇨ Select the right plane as reference

⇨ Set the offset distance as 80 mm as shown in figure (12)

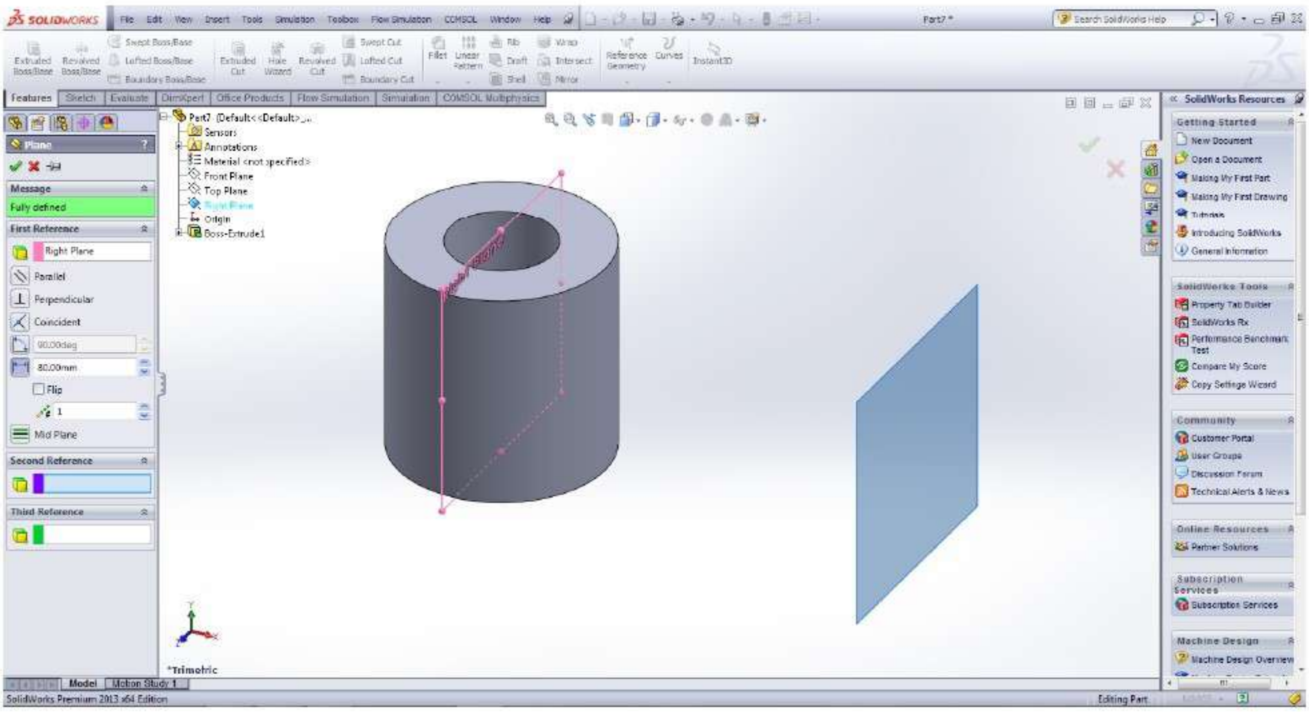

Figure (12)

⇨ Create the sketch shown in figure (13)
⇨ Select Extruded
⇨ Set the direction as up to surface
⇨ Select the outside surfaces of the cylinder as shown in figure (14)
⇨ Click OK

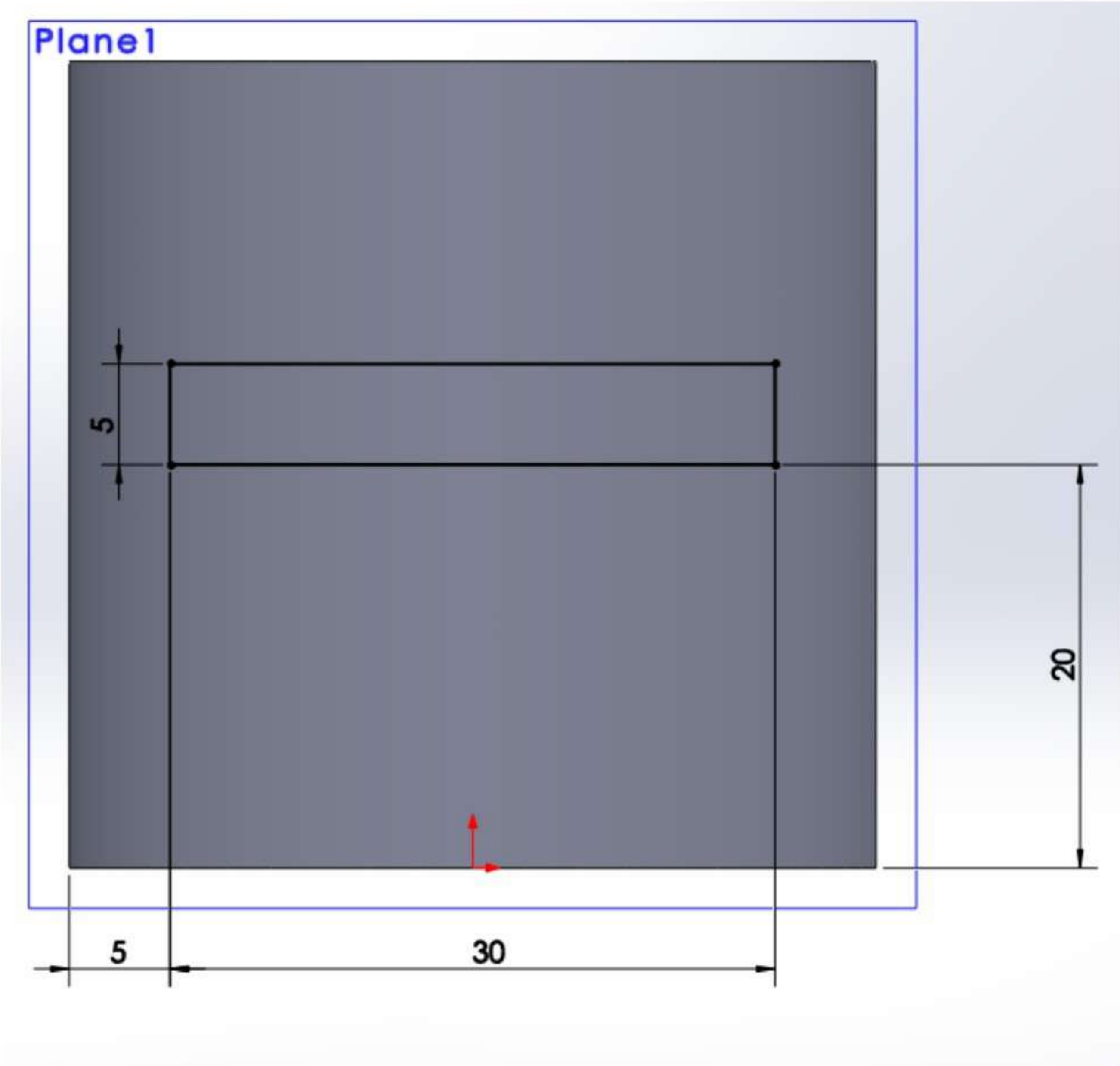

Figure (13)

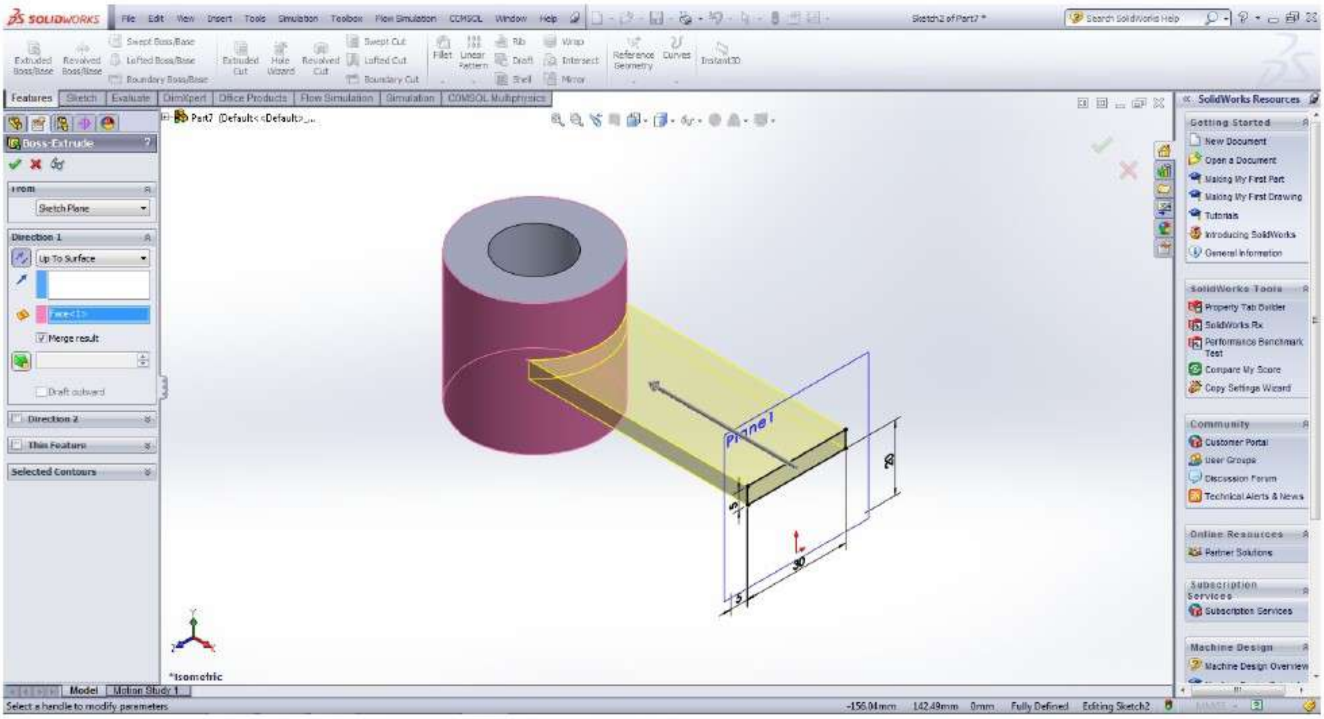

Figure (14)

⇨ On the top plane create the sketch shown in figure (15)

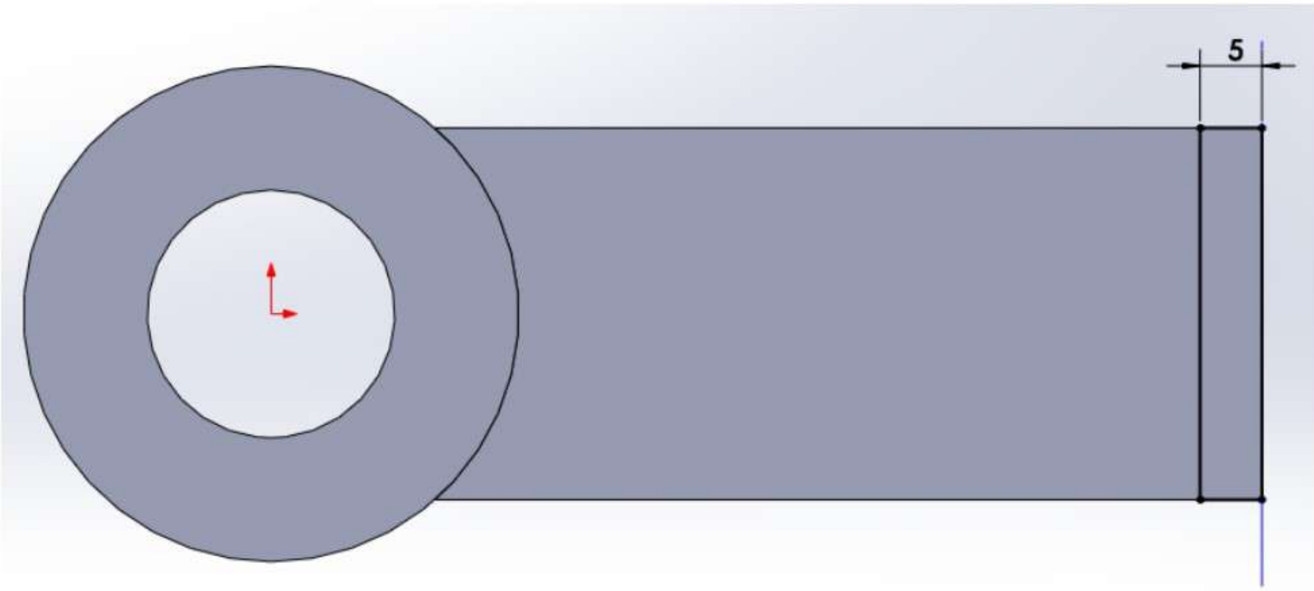

Figure (15)

⇨ Extrude as up to next, figure (16)

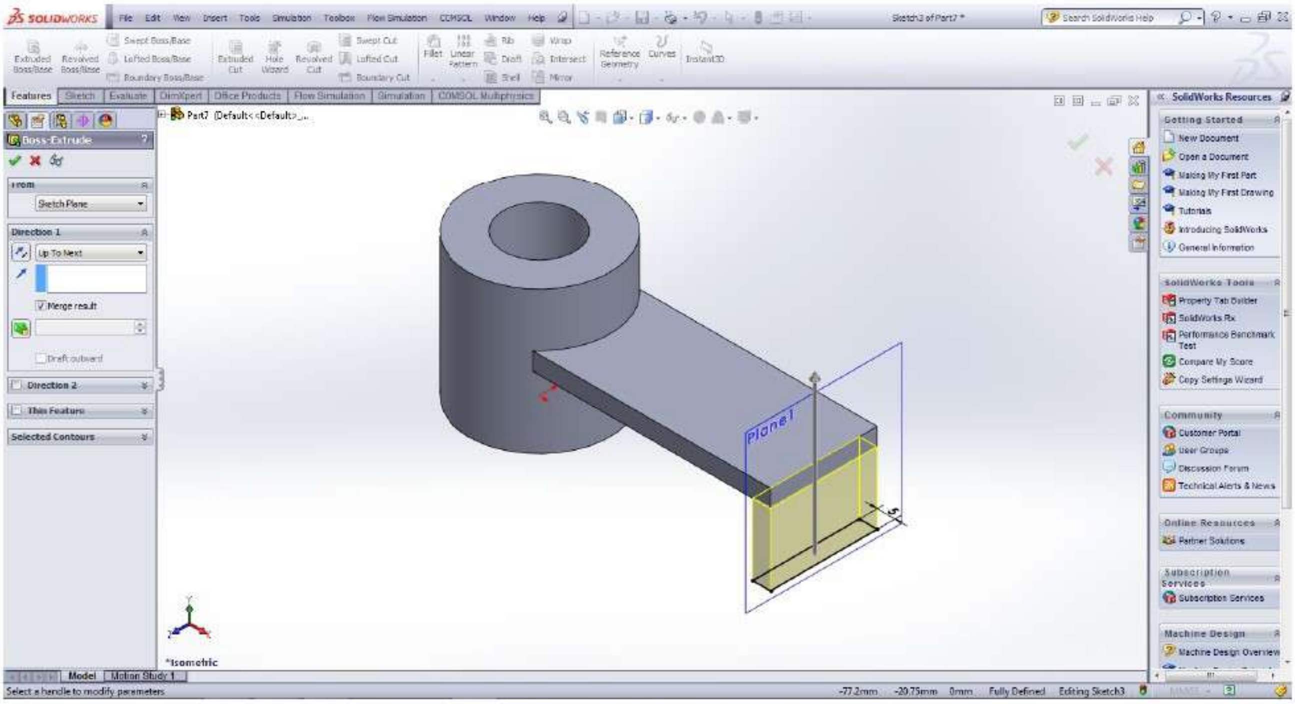

Figure (16)

⇨ Sketch and cut the two circles, the sketches are shown in figure (17) and figure (18)

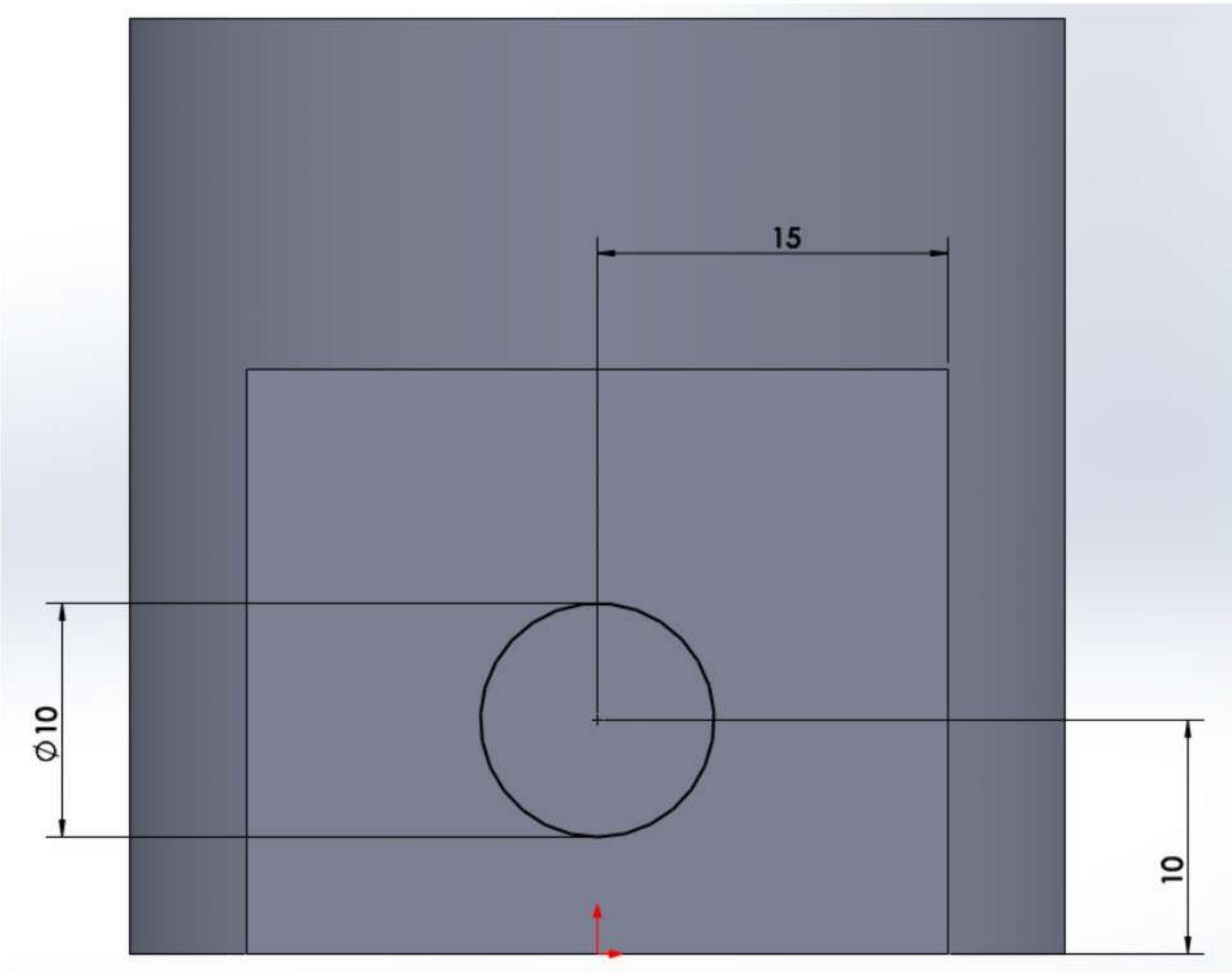

Figure (17)

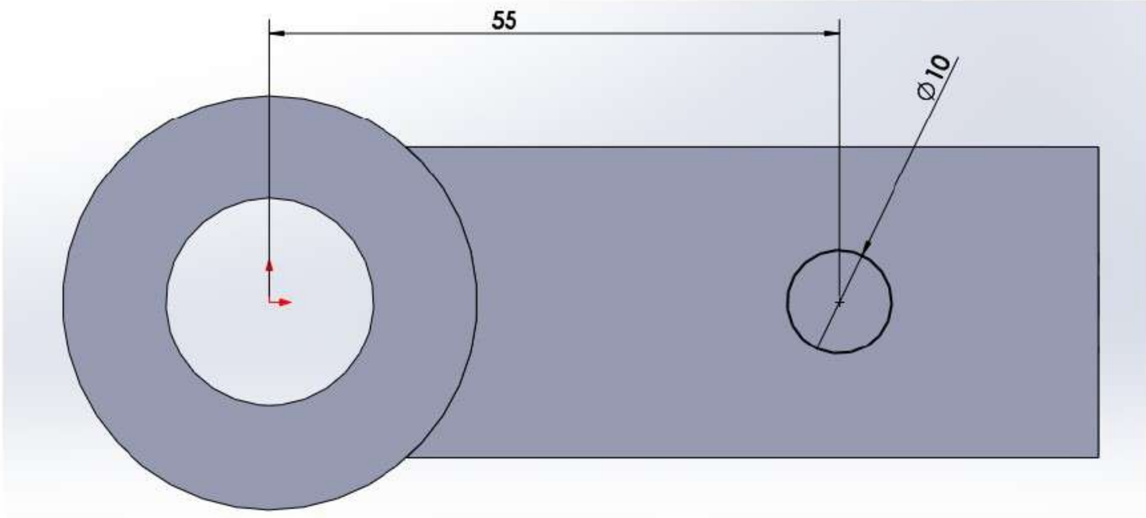

Figure (18)

HW

Use the design tree as reference