

Assembly

This portion of SolidWorks program allows you to assemble the parts and give them motion to see how its move with each other.

For this lecture we are going to create the bolt and nut shown in figure (1), then we are going to give them a screw mate and see how they contact with each other.

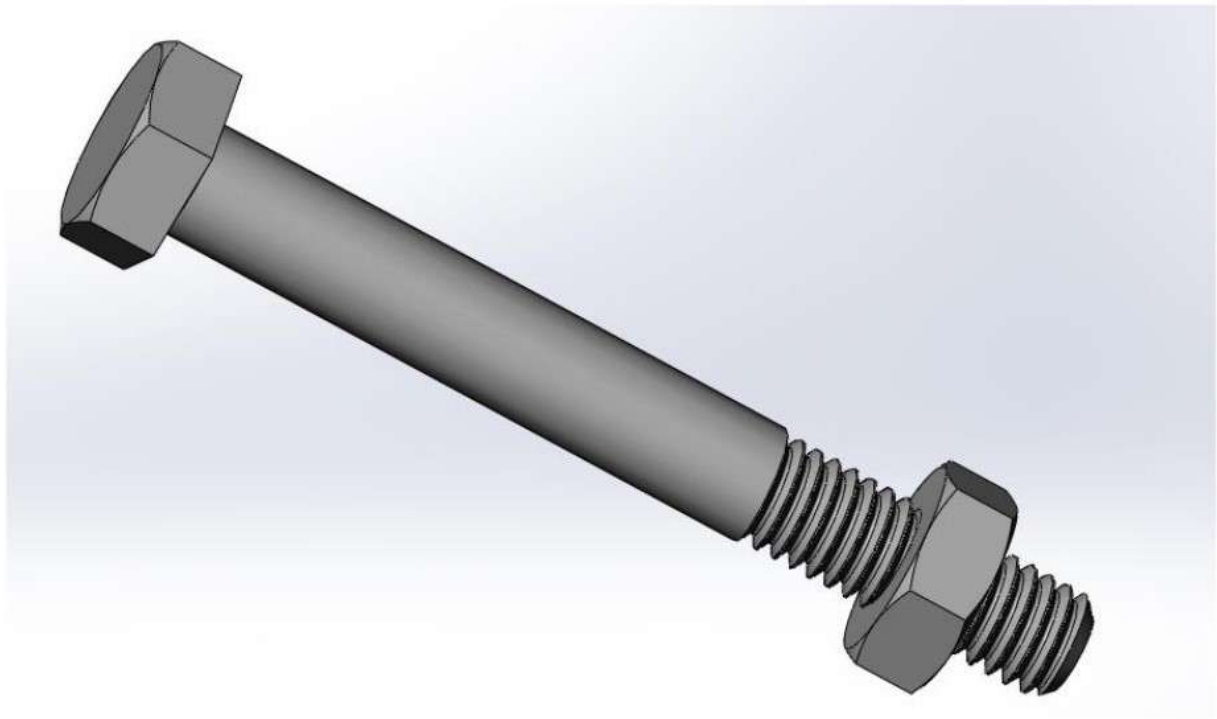


Figure (1)

⇒ On the front plane start by creating the six polygon sketch shown in figure (2), all the dimension are in Millimeter Unit

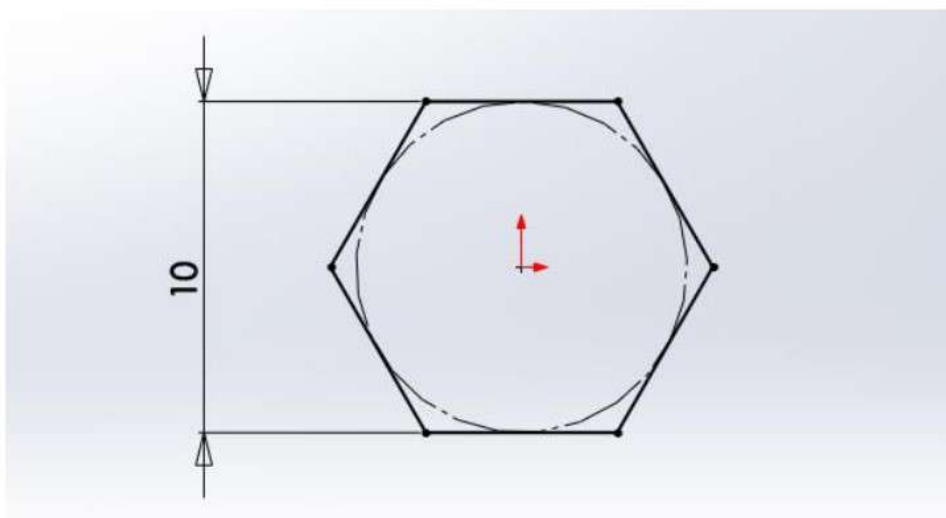


Figure (2)

- ⇒ Extrude to 4 mm
- ⇒ On the right plane sketch as figure (3)

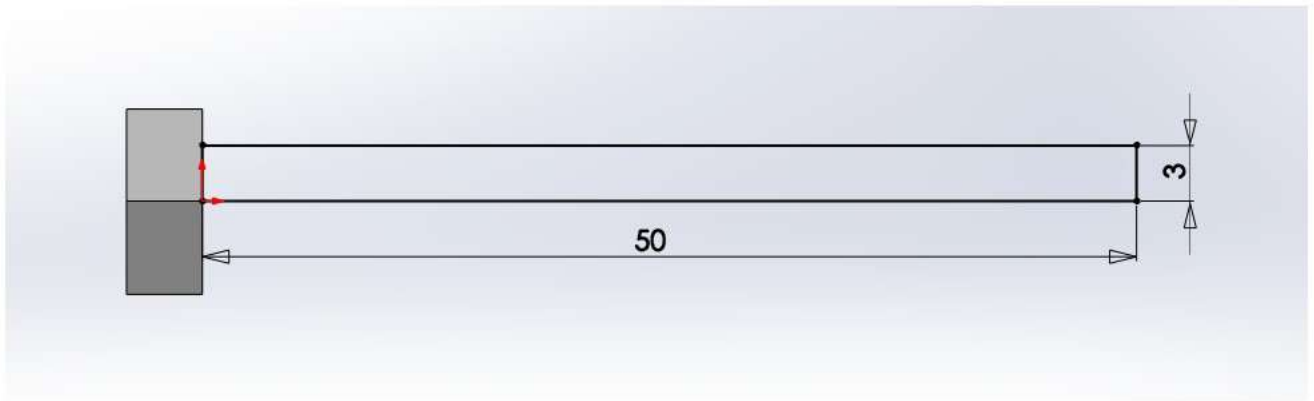


Figure (3)

- ⇒ Revolved the sketch 360° to make a cylindrical shape
- ⇒ Select Fillet from Features toolbar, see figure (4)

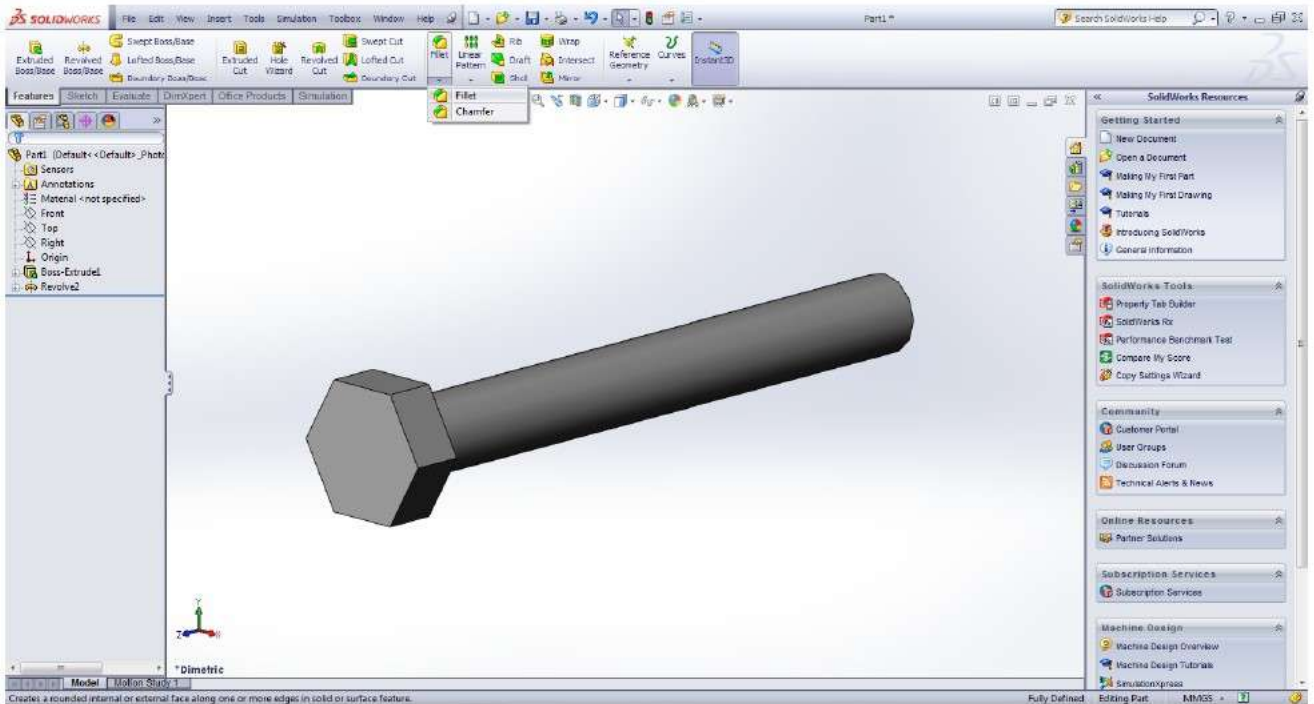


Figure (4)

- ⇒ Select Constant radius
- ⇒ Set the radius value as 0.4 mm
- ⇒ Select the edge as shown in figure (5)

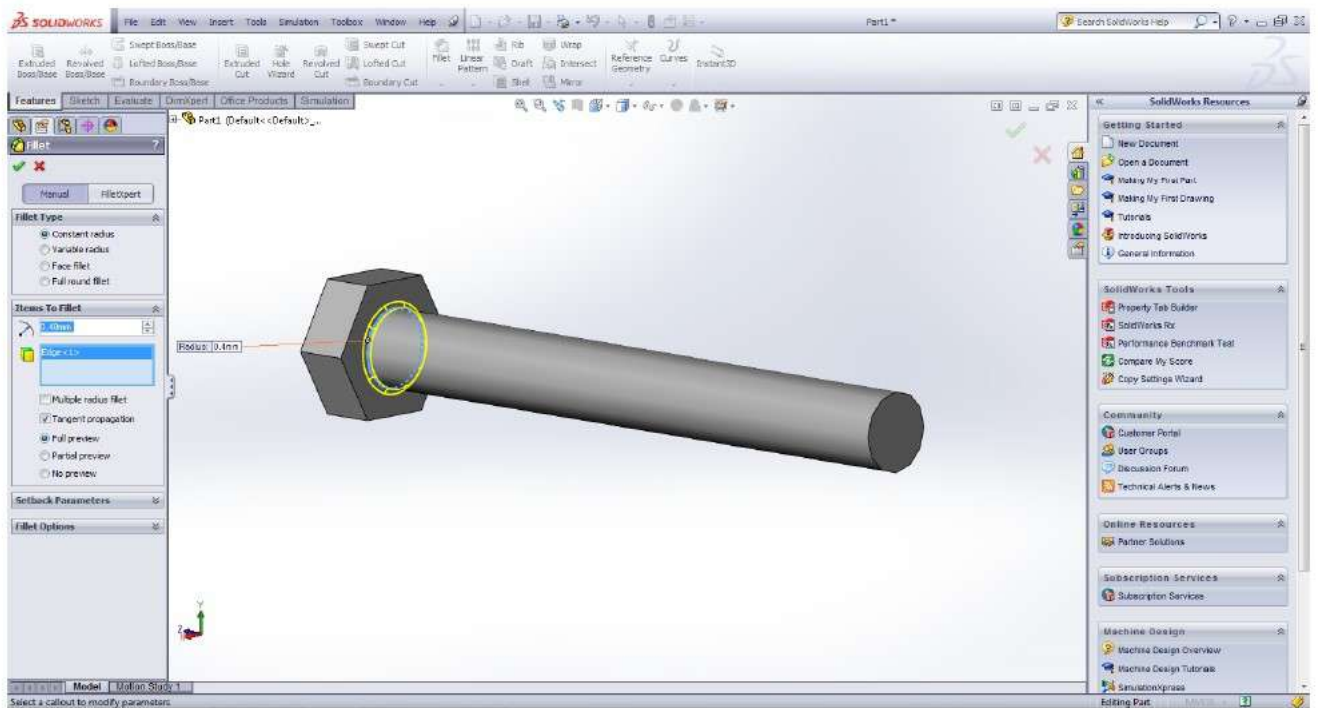


Figure (5)

- ⇒ Click OK
- ⇒ Select Chamfer from Features toolbar, see figure (6)

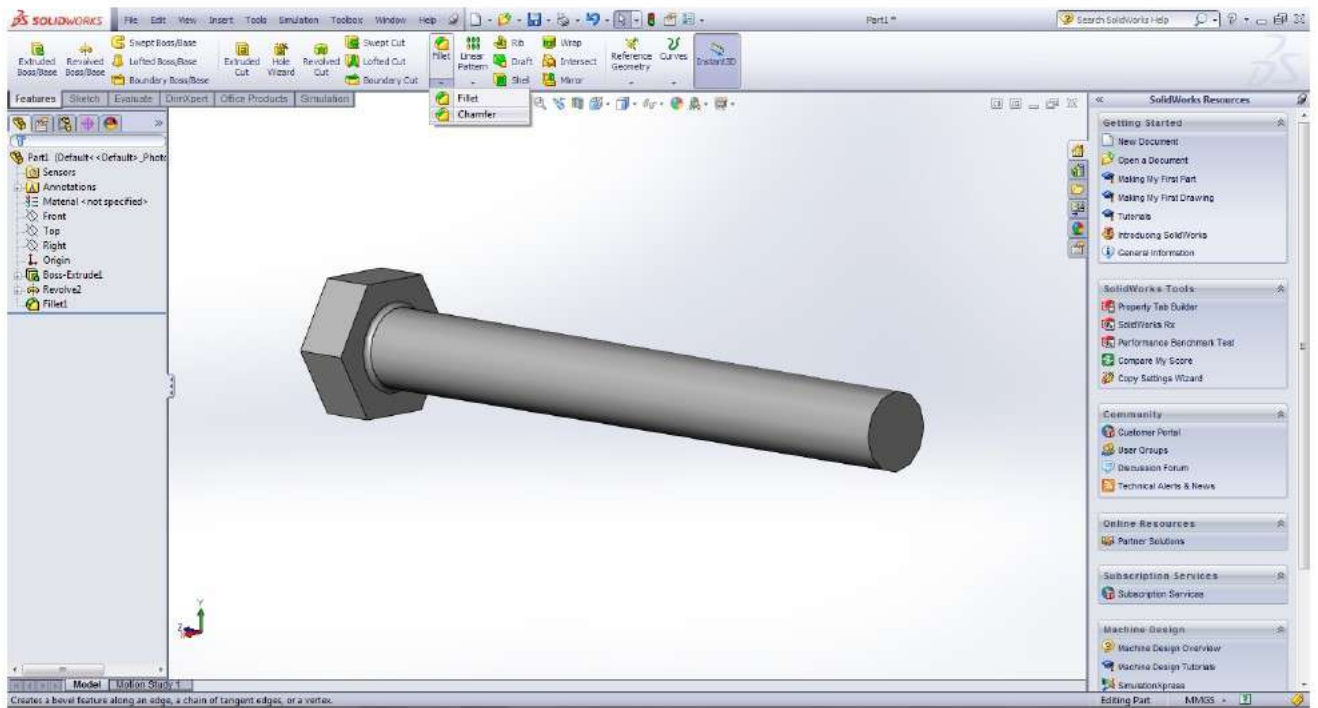


Figure (6)

- ⇒ Set the distance as 1 mm
- ⇒ Select the edge as shown in figure (7)

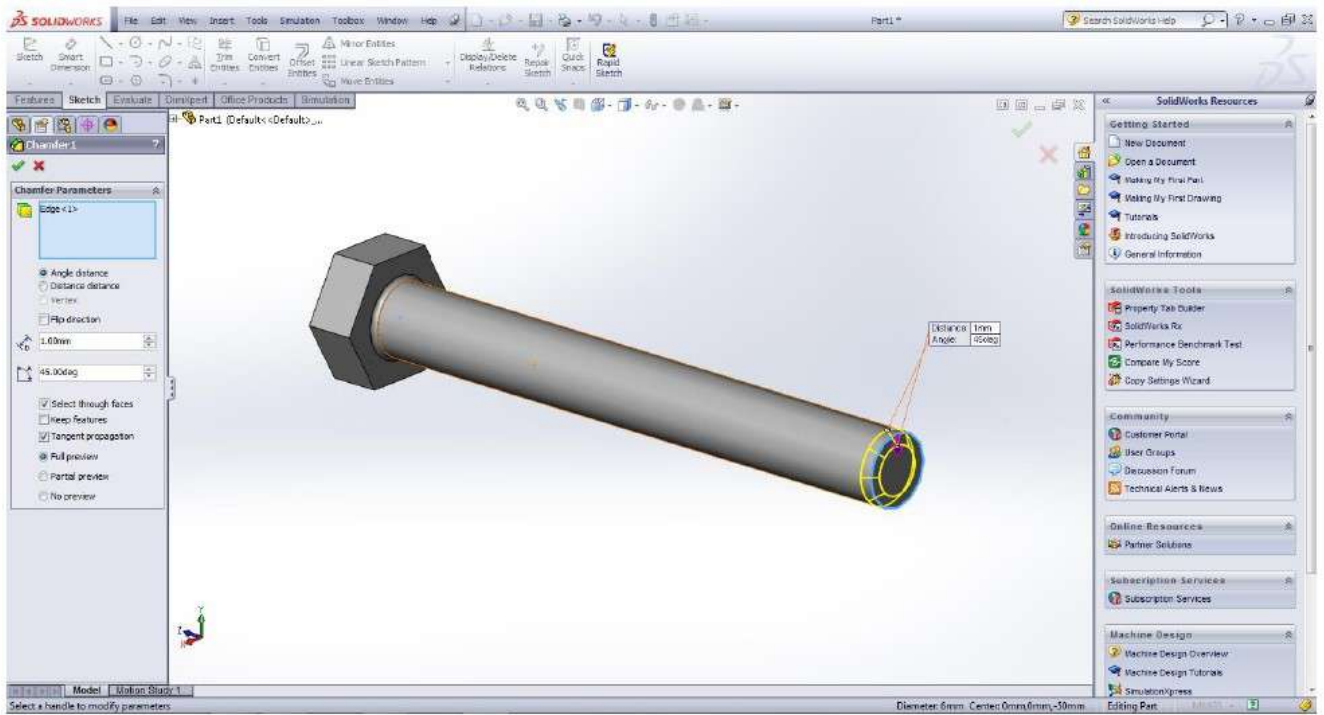


Figure (7)

- ⇒ Click OK
- ⇒ Sketch a 10 mm circle on the top surface of the geometry
- ⇒ From features toolbar select Extrude Cut
- ⇒ Check Flip side to cut
- ⇒ Activate draft
- ⇒ Set 60° draft angle
- ⇒ Click OK to figure (8)

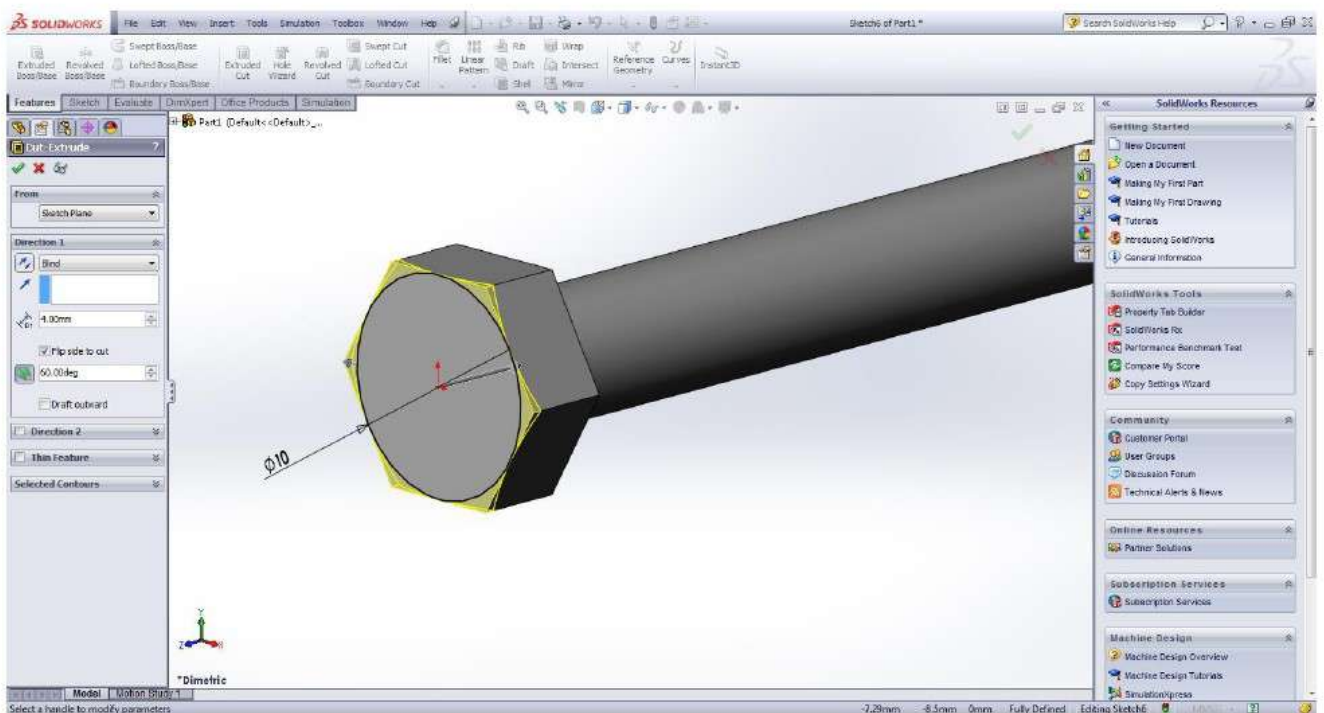


Figure (8)

⇒ Sketch a circle with 6 mm diameter on the base of the geometry as shown in figure (9)

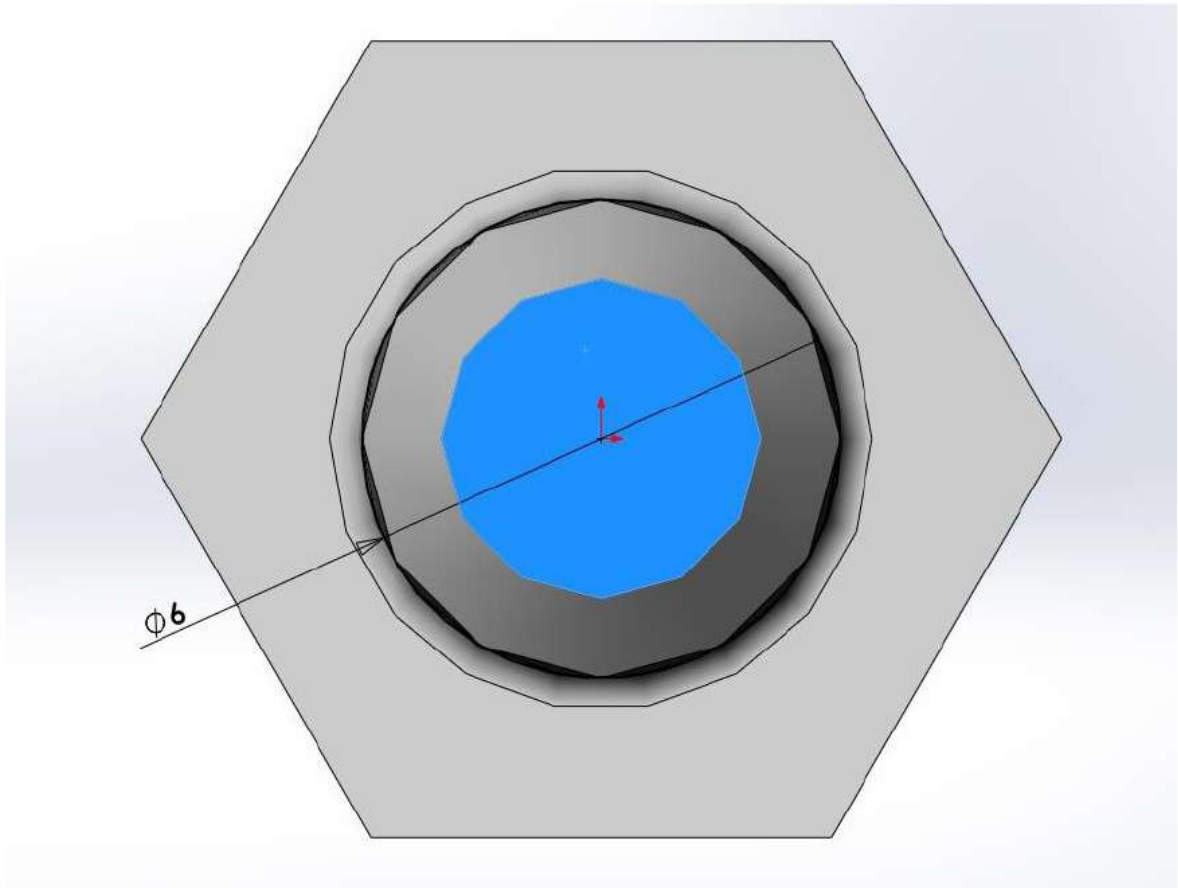


Figure (9)

- ⇒ Open insert and hover over Curve option
- ⇒ Select Helix/Spiral as shown in figure (10)

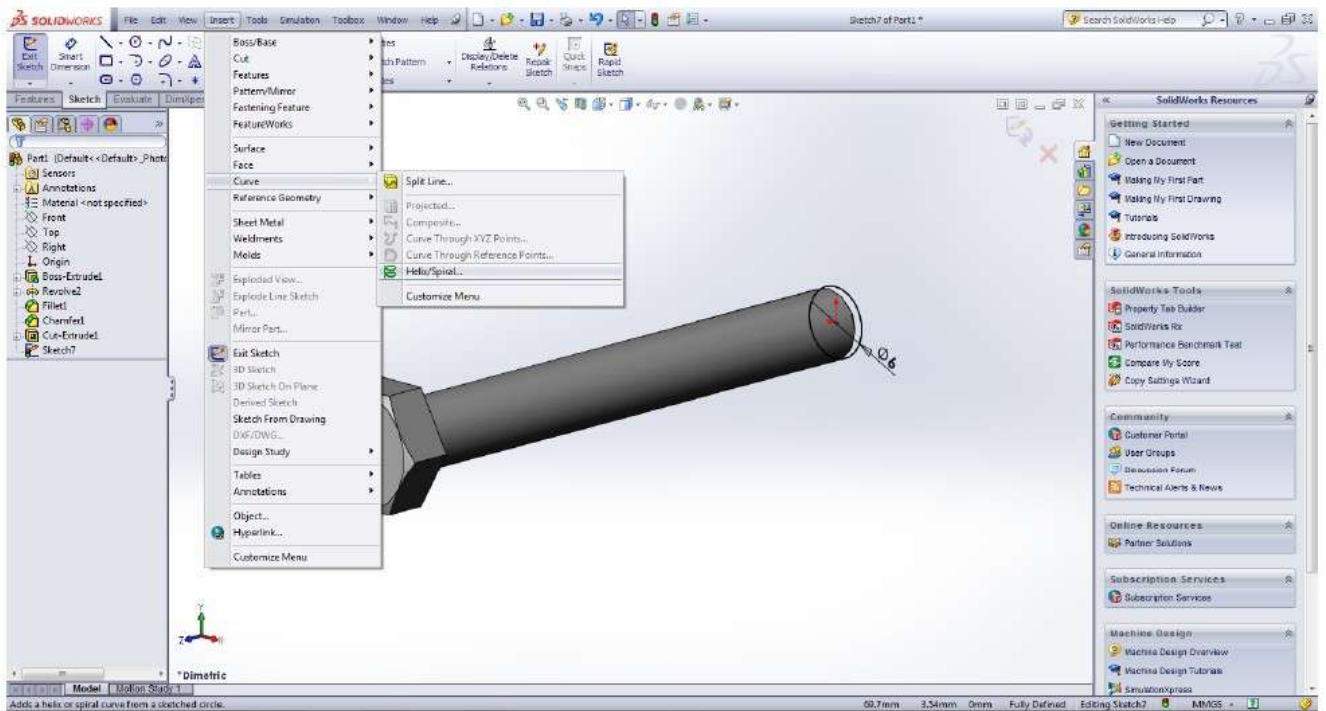


Figure (10)

- ⇒ Set the type as height and Pitch
- ⇒ Set the height as 18 mm
- ⇒ Set the Pitch as 1 mm
- ⇒ Change the start angle to 0
- ⇒ Reverse the direction if needed
- ⇒ Click OK , to figure (11)

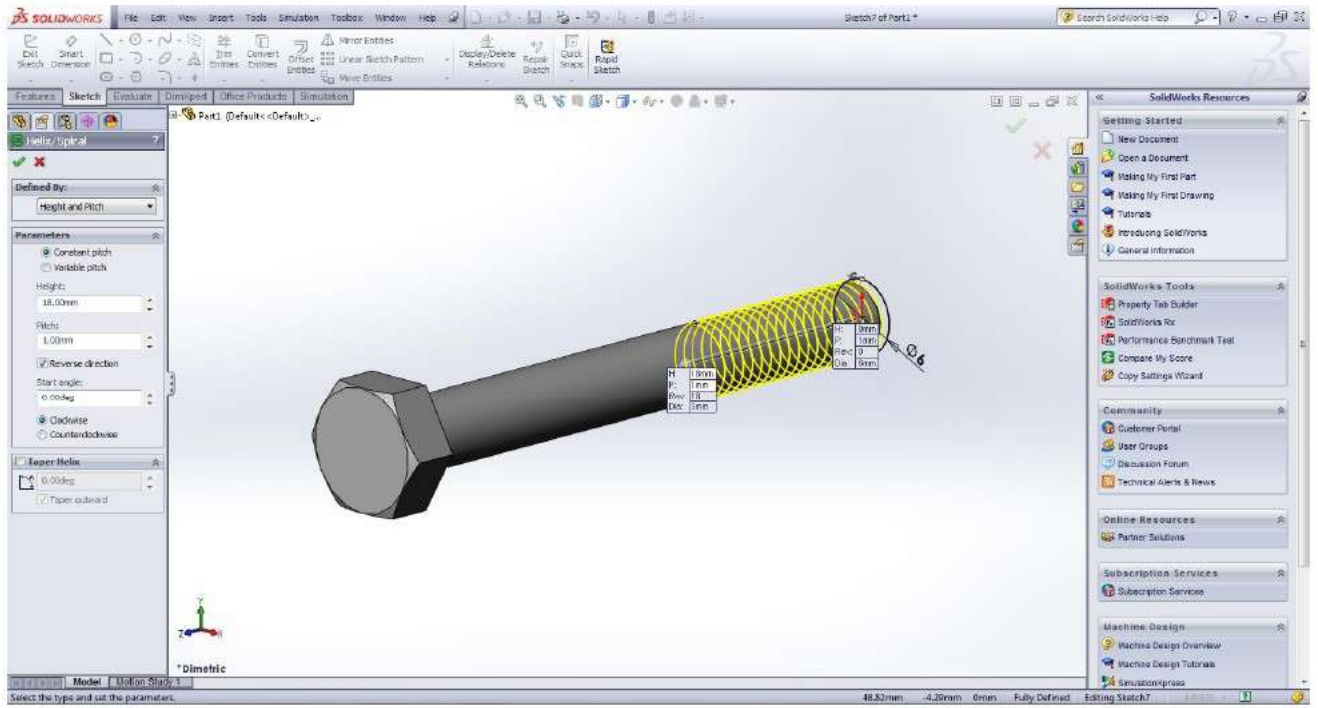


Figure (11)

⇒ On the Right plane create the sketch shown in figure (12)

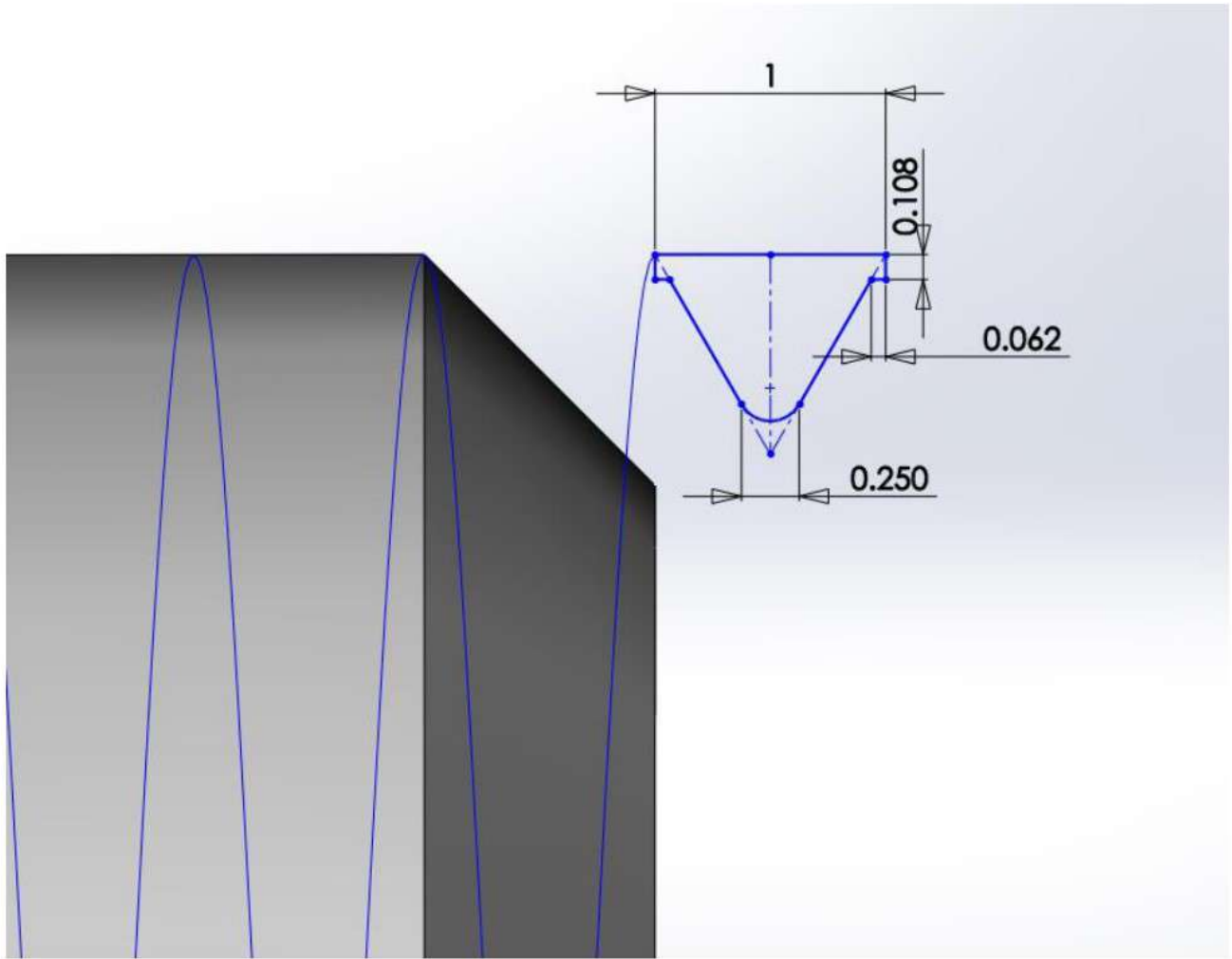


Figure (12)

- ⇒ Exit the sketch
- ⇒ From features toolbar select Sweep Cut
- ⇒ Sweep the profile sketch over the helix
- ⇒ Click OK

Save the Part as Bolt

Open a new Part

- ⇒ On the front plane create the six polygon sketch with circle in the middle as shown in figure (13), all the dimension are in Millimeter Unit

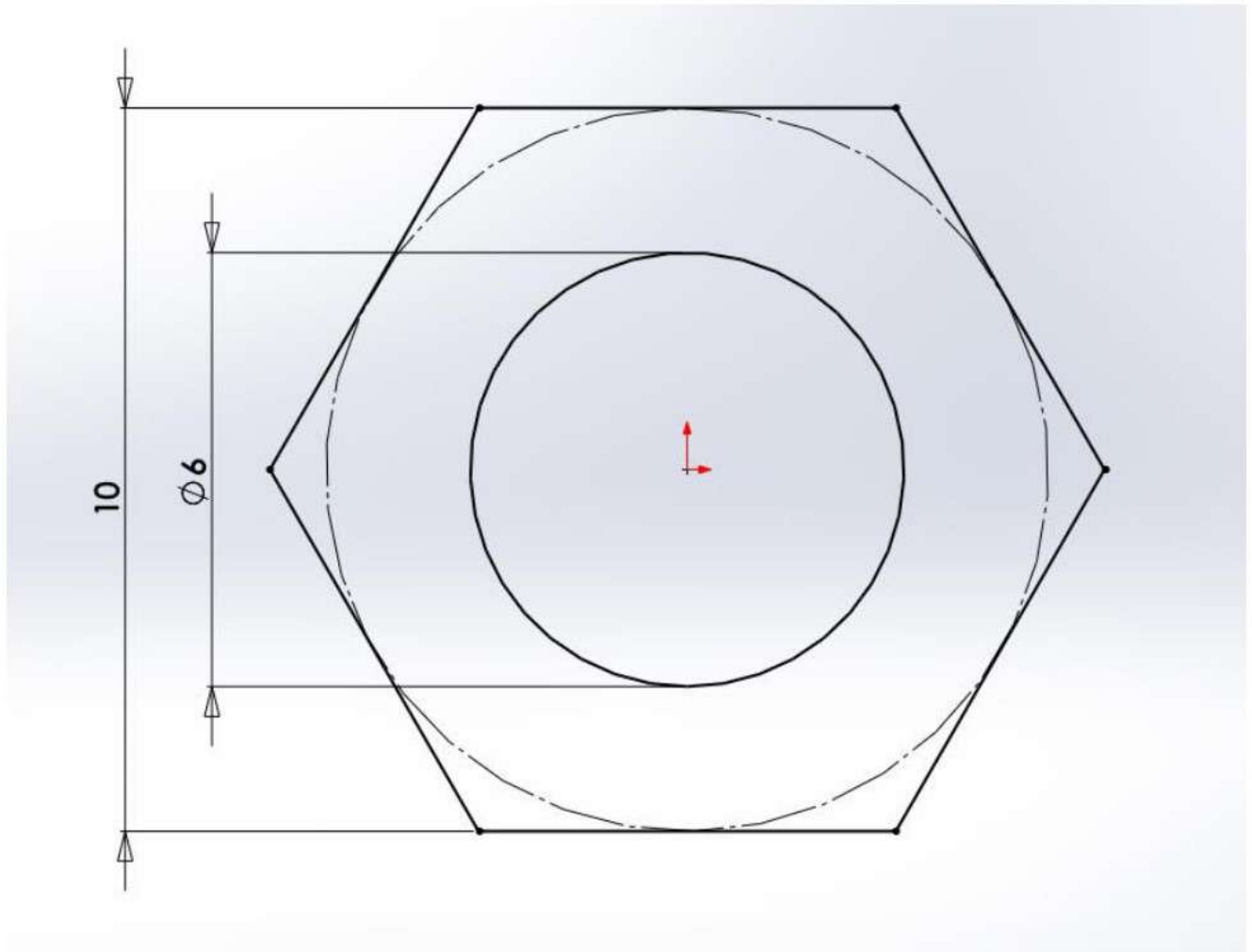


Figure (13)

⇒ Extrude to 4 mm

- ⇒ Sketch a 10 mm circle on one of the polygon surfaces of the geometry
- ⇒ From features toolbar select Extrude Cut
- ⇒ Check Flip side to cut
- ⇒ Activate draft
- ⇒ Set 60° draft angle
- ⇒ Click OK to figure (14)

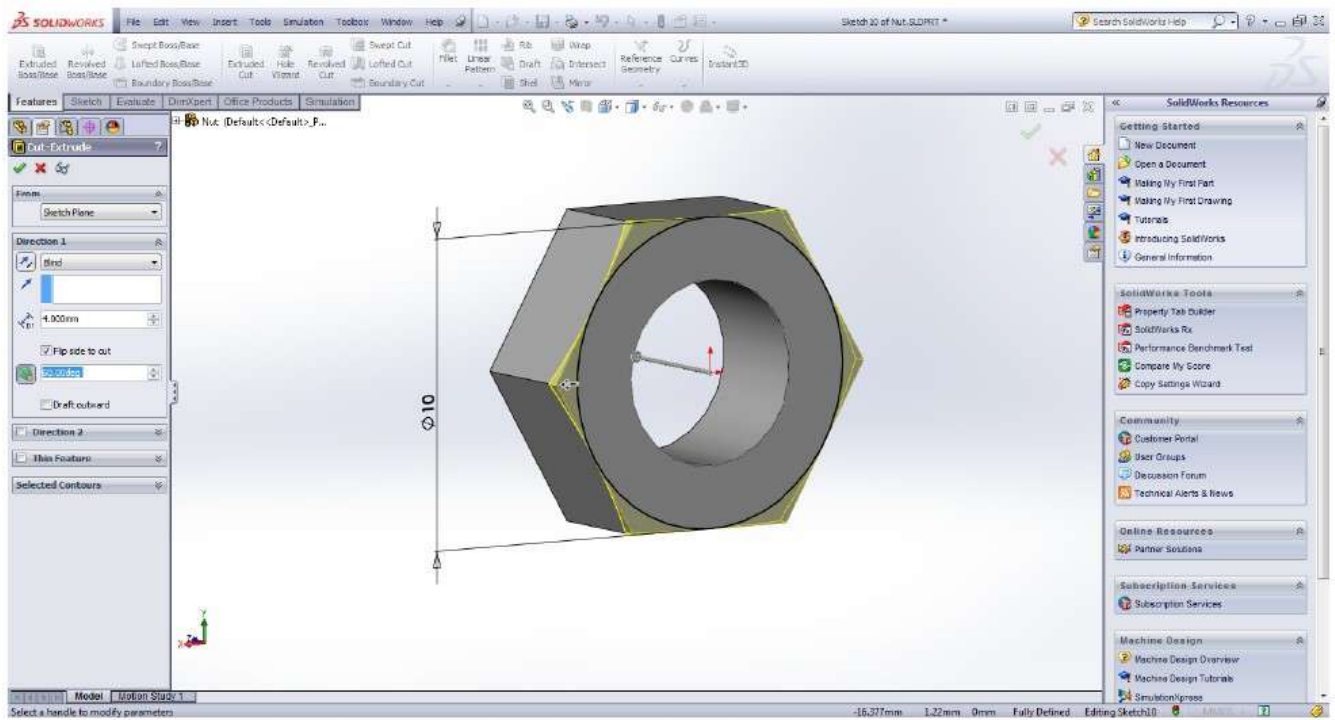


Figure (14)

- ⇒ Repeat the steps on the other polygon surfaces
- ⇒ On one of the polygon surfaces create a 6 mm circle sketch as shown in figure (15)

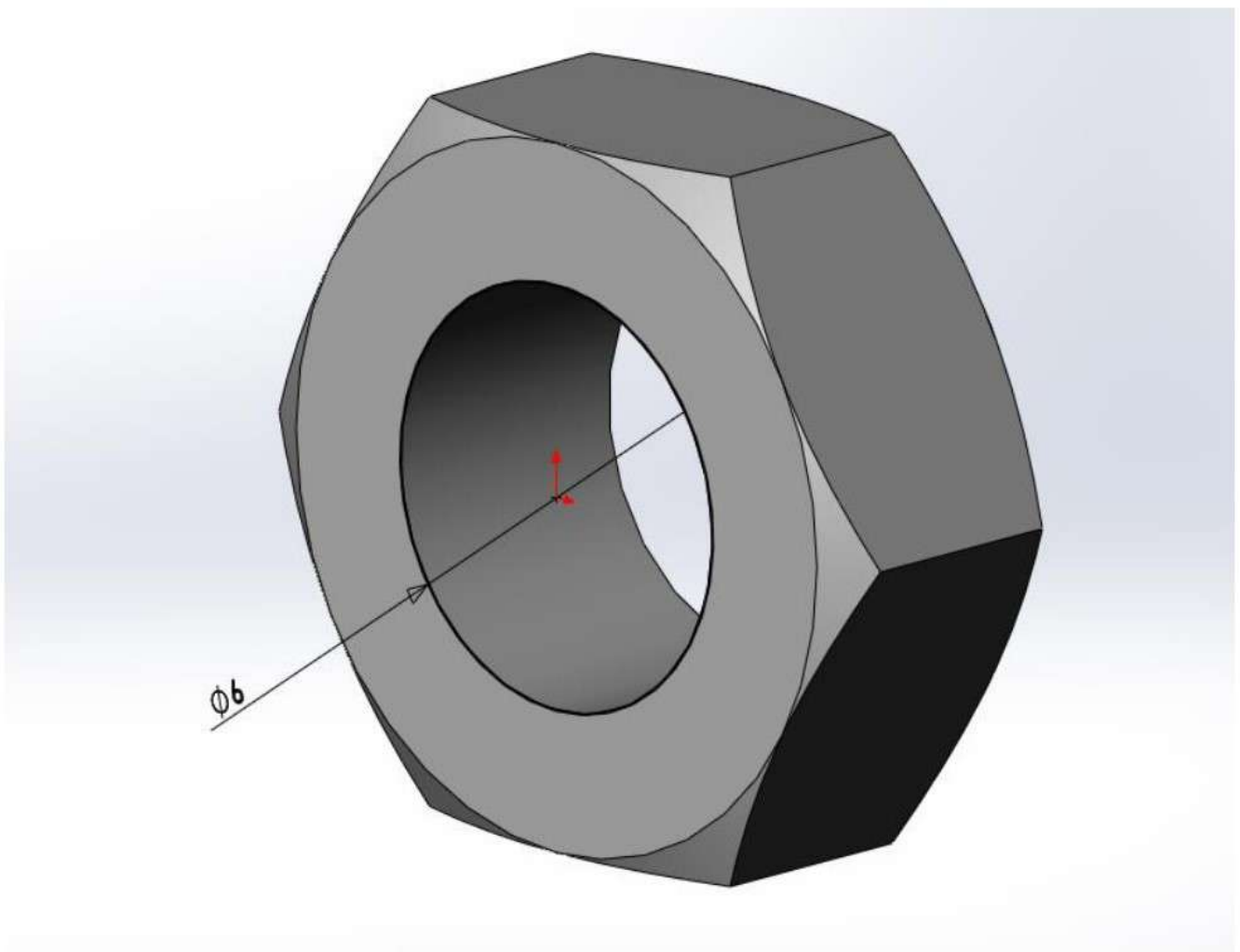


Figure (15)

⇒ Create a height and Pitch helix, with 4 mm height and 1 mm pitch, as shown in figure (16)

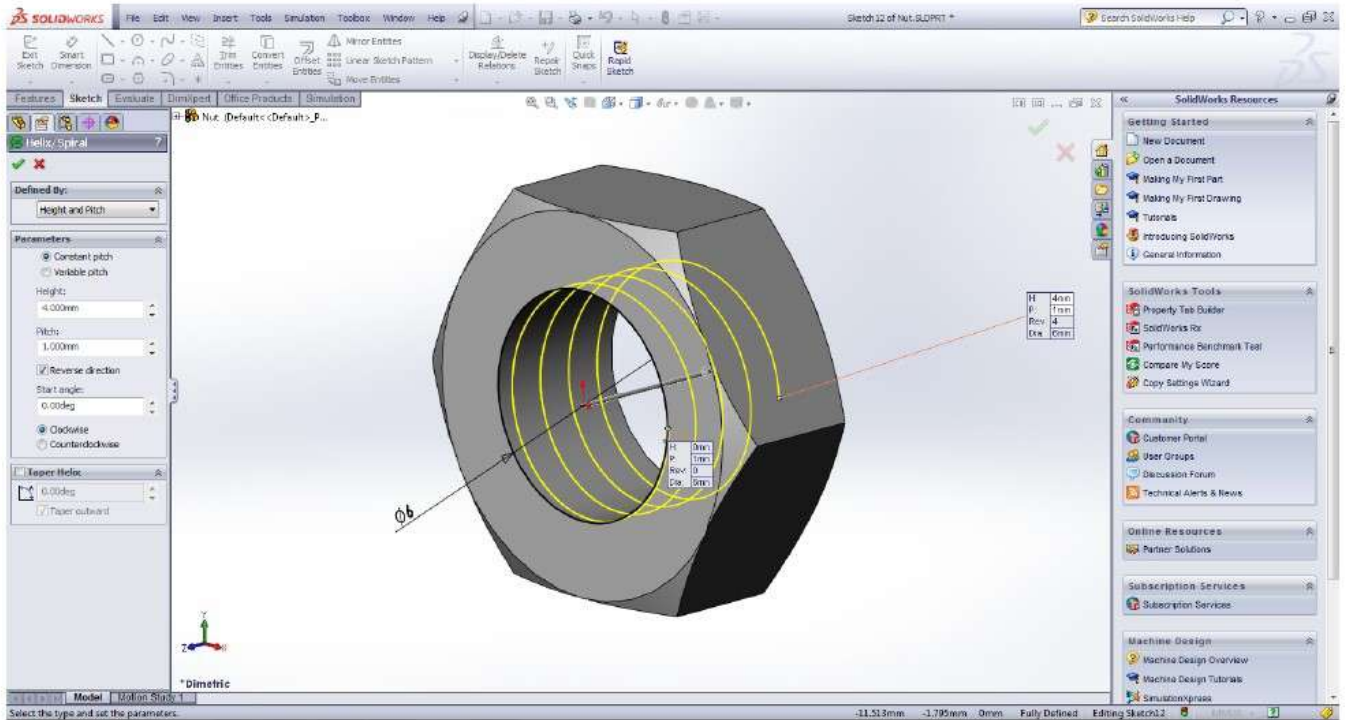


Figure (16)

⇒ Click OK

⇒ On the top plane create the sketch shown in figure (17)

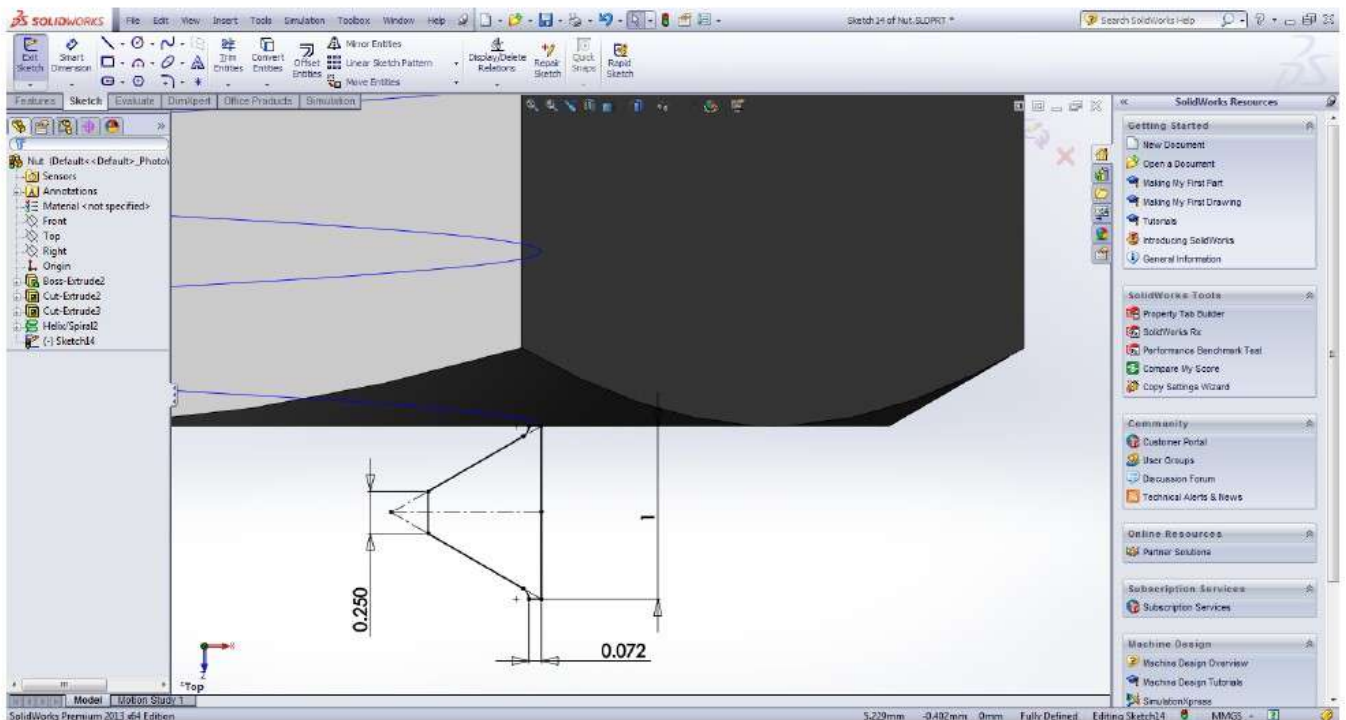


Figure (17)

⇒ Exit the sketch

⇒ From features toolbar select Sweep

⇒ Sweep the profile sketch over the helix

⇒ Click OK

⇒ In order to cut the access geometry, create a rectangular on the top plane as shown in figure (18)

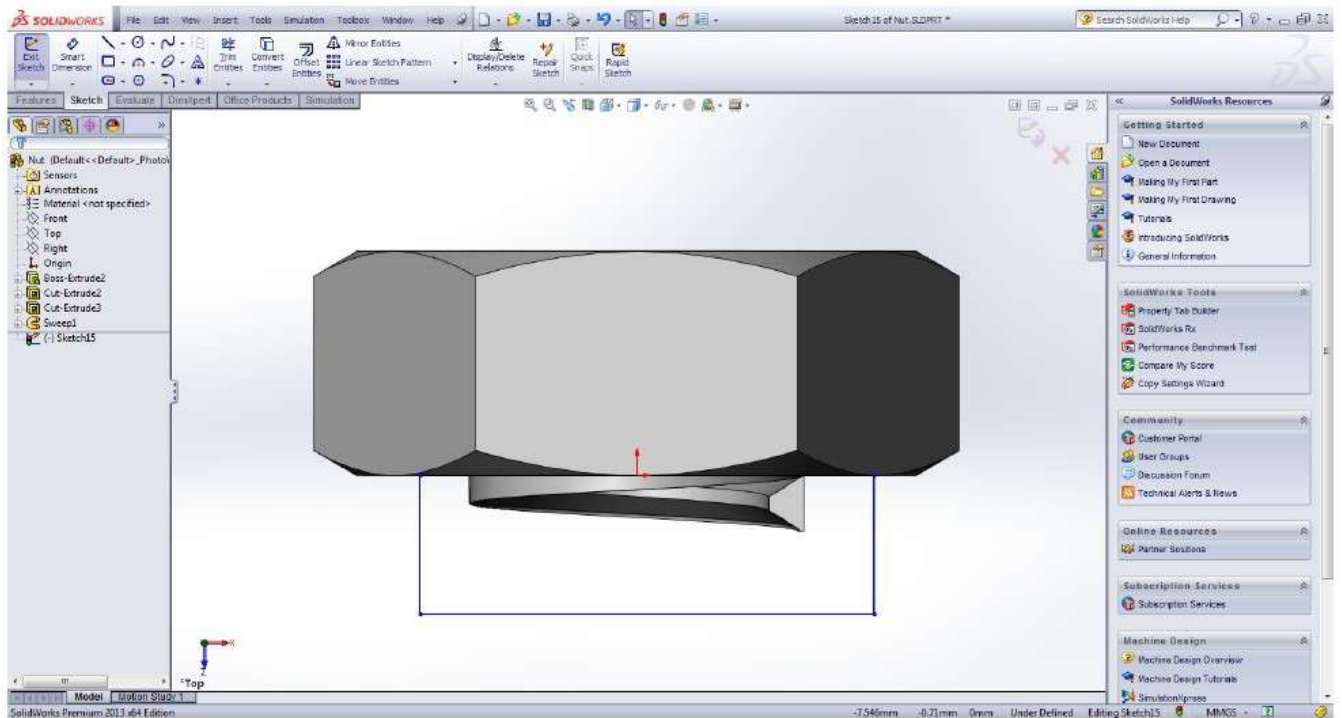


Figure (18)

- ⇒ Extrude Cut through all in both directions
- ⇒ Save as Nut

Open New Assembly

- ⇒ From Assembly toolbar, figure (19), select insert components



Figure (19)

- ⇒ Browse the Bolt and Nut parts you just created one by one
- ⇒ Click on Mate from assembly toolbar
- ⇒ Select the cylindrical surface on the bolt and the circular edge on the nut
- ⇒ Make them concentric
- ⇒ Click OK to figure (20)

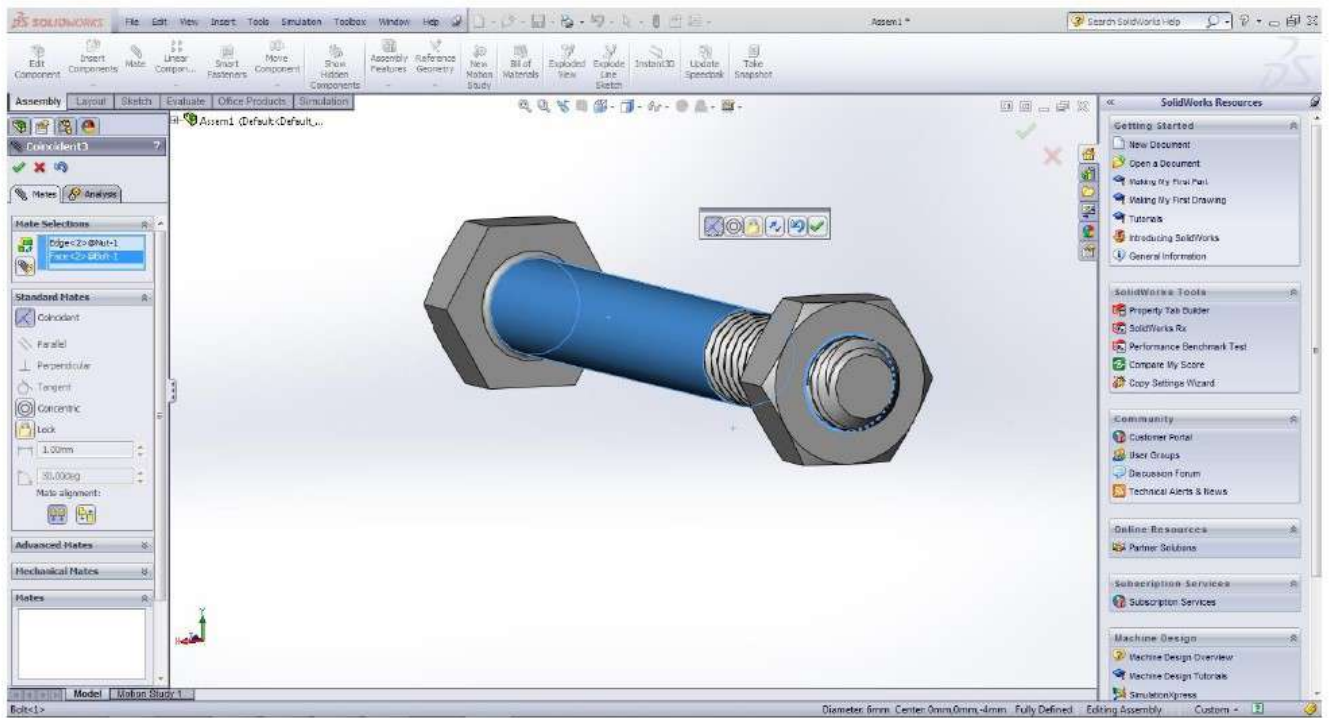


Figure (20)

- ⇒ Click on section view
- ⇒ Adjust the position of the nut threads with the bolt threads, as shown in figure (21), by clicking on the part and drag

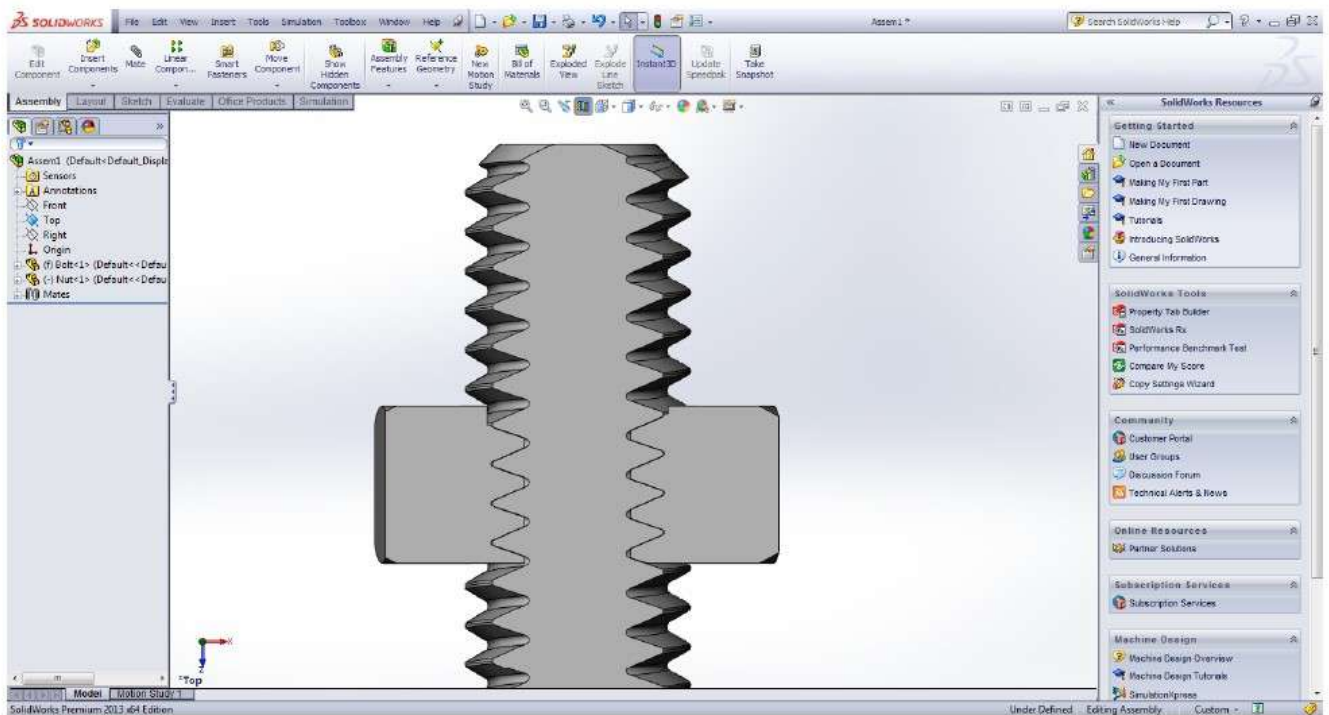


Figure (21)

- ⇒ Select Mate from assembly toolbar
- ⇒ Open Mechanical mates
- ⇒ Click Screw mate
- ⇒ Select Distance/revolution
- ⇒ Set the value as 1 mm, which represent the pitch of the bolt

- ⇒ Select the circle edge on the nut and the cylindrical surface on the bolt
- ⇒ Click Ok to figure (22)

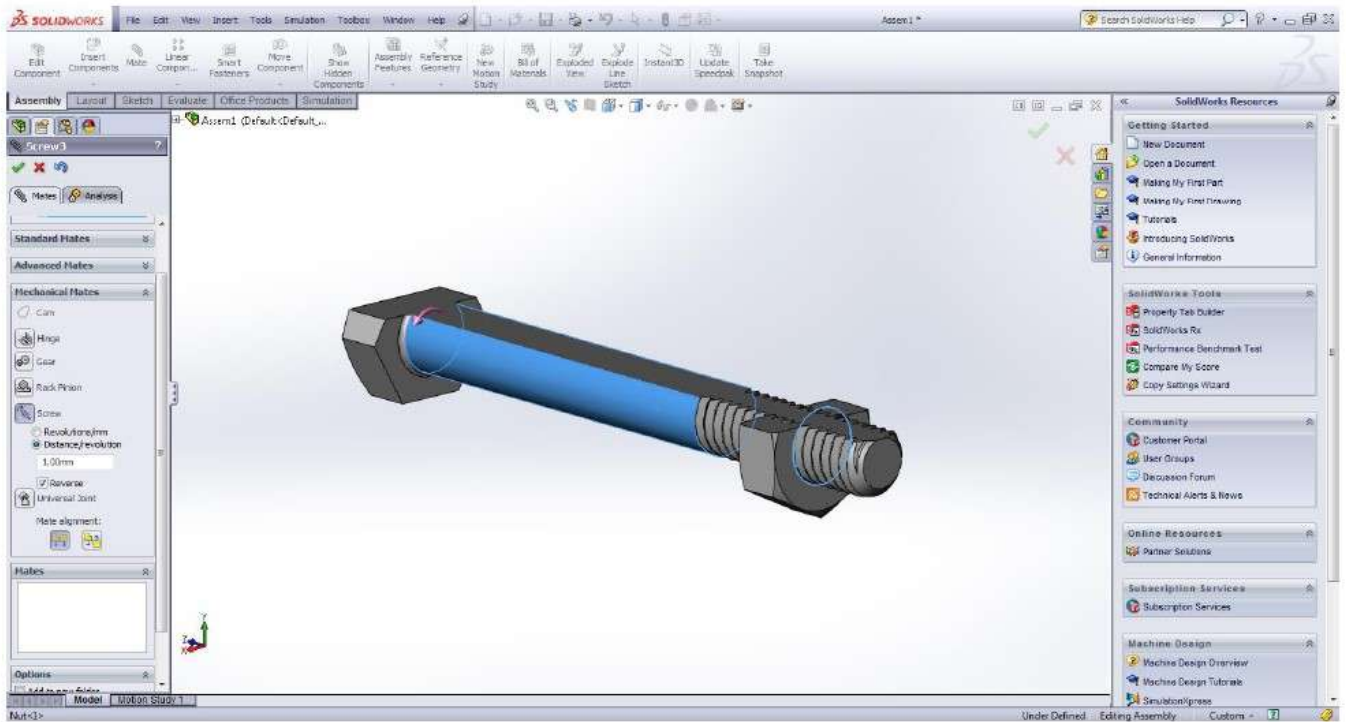


Figure (22)

To check if there is any interfaces between your components

- ⇒ Go to Evaluate toolbar ,figure (23)



Figure (23)

- ⇒ Click Interference Detection
- ⇒ Select the bolt and the nut
- ⇒ Click on Calculate
- ⇒ If there is any interference it should be detected with in your set limits

H.W

