RPS GROUP OF INSTITUTIONS
Balana, Satnali Road,
Mohindergarh, Haryana 123029
Ph.: 91-1285-241431
Mob.: 09466275566, 09416150201
E-mail: info@rpsinstitutions.org
Web : www.rpsinstitutions.org

Approved by **AICTE** (Govt. of India) & Affiliated to **M.D. University**, Rohtak

# LABORATORY MANUAL

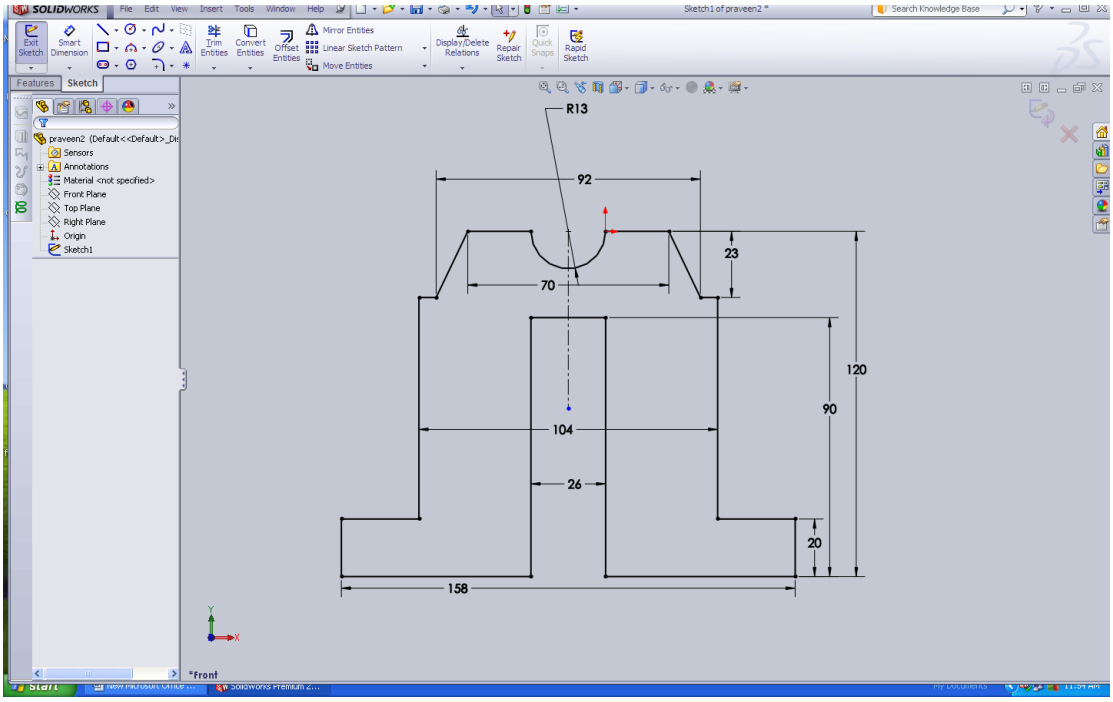
## COMPUTER AIDED DESIGN LAB

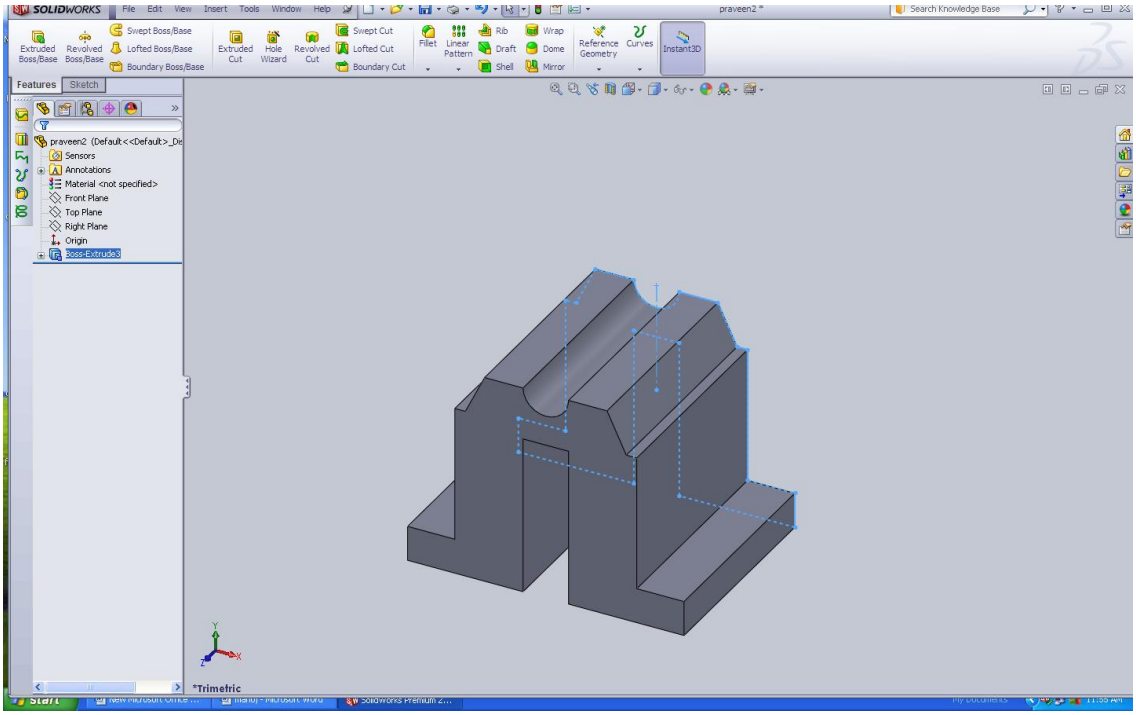
| Sr. No | Experiment Title |
|---|---|
| 1 | Setting up of drawing environment by setting drawing limits, drawing units, naming the drawing, naming layers, setting line types for different layers using various type of lines in engineering drawing, saving the file with .dwg extension. |
| 2 | Layout drawing of a building using different layer and line colors indicating all Building details. Name the details using text commands, Make a title Block. |
| 3 | To Draw Orthographic projection Drawings (Front, Top and side) of boiler safety valve giving name the various components of the valve. |
| 4 | Make an Isometric dimensioned drawing of a connecting Rod using isometric grid and snap. |
| 5 | Draw quarter sectional isometric view of a cotter joint. |
| 6 | Draw different types of bolts and nuts with internal and external threading in Acme and square threading standards. Save the bolts and nuts as blocks suitable for insertion. |
| 7 | Draw 3D models by extruding simple 2D objects, dimension and name the objects. |
| 8 | Draw a Spring using helix command |

RPS GROUP OF INSTITUTIONS
RPSGOI MOHINDERGARH
Balana, Satnali Road,
Mohindergarh, Haryana 123029
Ph.: 91-1285-241431
Mob.: 09466275566, 09416150201
E-mail: info@rpsinstitutions.org
Web : www.rpsinstitutions.org
Approved by **AICTE** (Govt. of India) & Affiliated to **M.D. University**, Rohtak

# EXPERIMENT:-1

## OBJECTIVE:- TO DRAW A SIMPLE 2-D OBJECT AND EXTRUDE IT

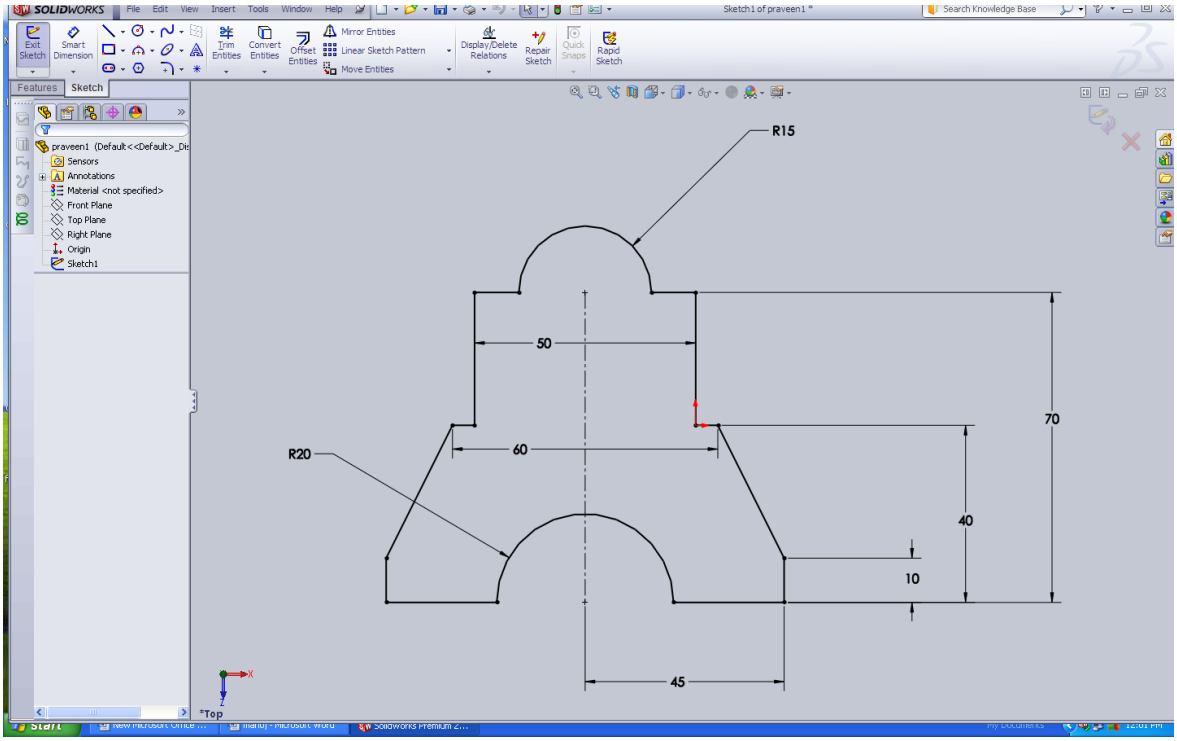


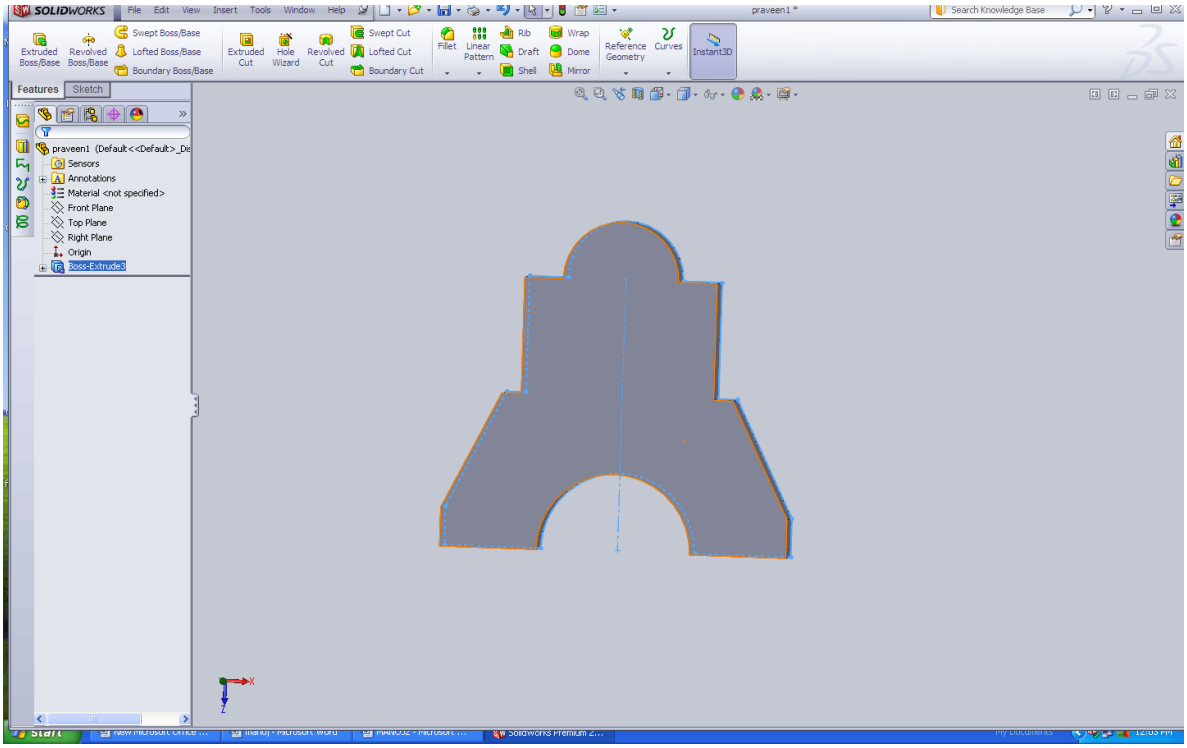



RPS GROUP OF INSTITUTIONS
Balana, Satnali Road,
Mohindergarh, Haryana 123029
Ph.: 91-1285-241431
Mob.: 09466275566, 09416150201
E-mail: info@rpsinstitutions.org
Web : www.rpsinstitutions.org
Approved by **AICTE** (Govt. of India) & Affiliated to **M.D. University**, Rohtak

# EXPERIMENT NO:-2

## OBJECTIVE:- TO DRAW A SIMPLE 2-D OBJECT AND EXTRUDE

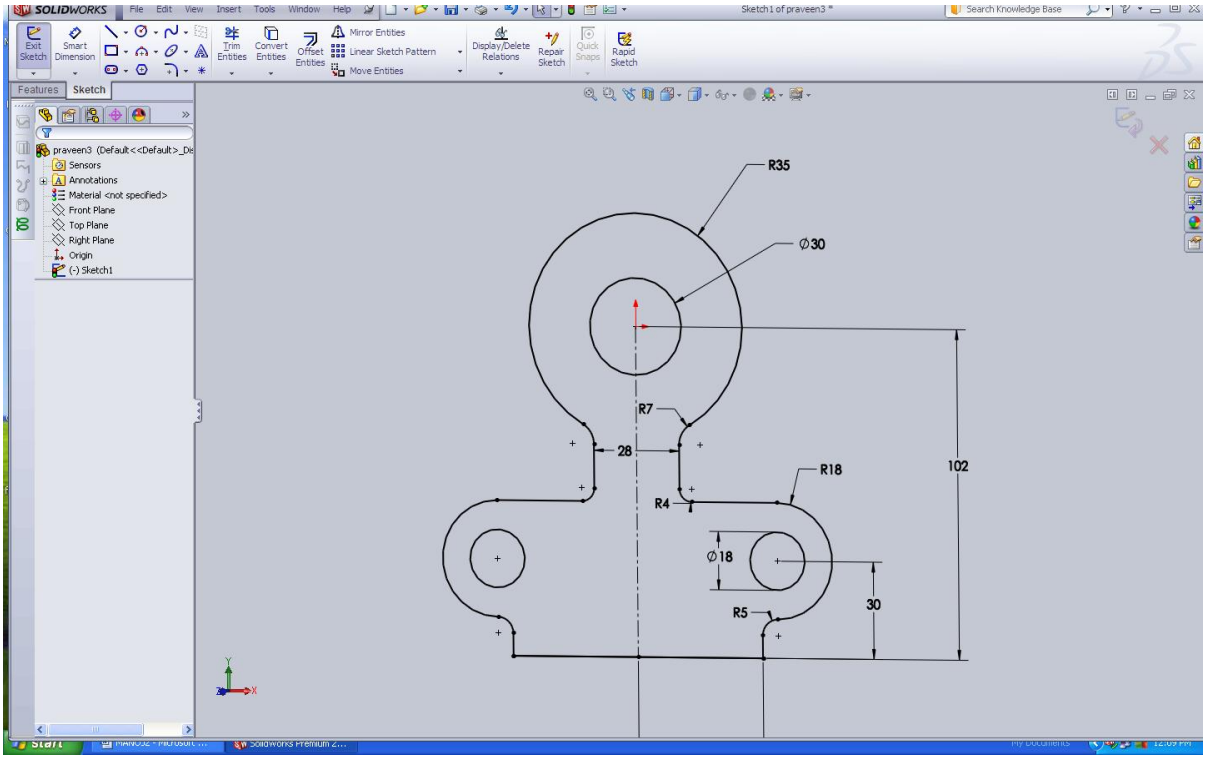


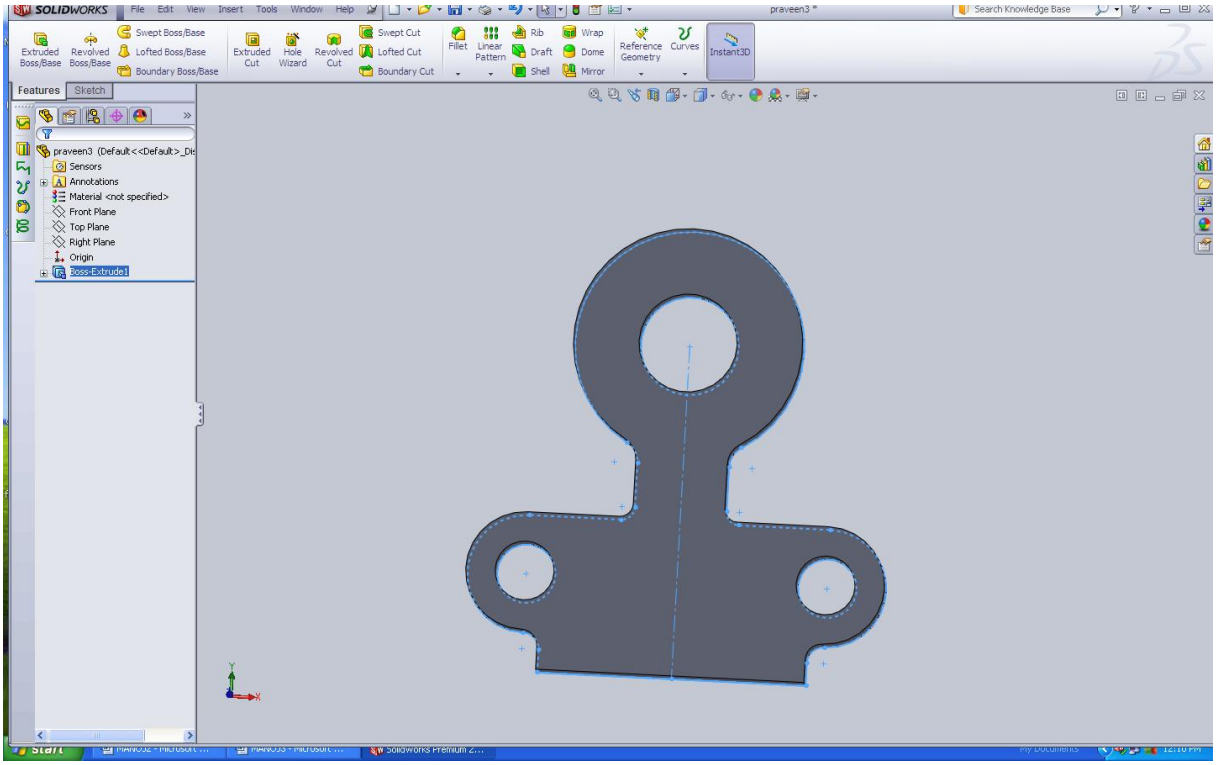



RPSGOI MOHINDERGARH
RPS GROUP OF INSTITUTIONS
Balana, Satnali Road,
Mohindergarh, Haryana 123029
Ph.: 91-1285-241431
Mob.: 09466275566, 09416150201
E-mail: info@rpsinstitutions.org
Web : www.rpsinstitutions.org
Approved by **AICTE** (Govt. of India) & Affiliated to **M.D. University**, Rohtak

# EXPERIMENT:-3

## OBJECTIVE:- TO DRAW A SIMPLE 2-D OBJECT AND EXTRUDE IT.

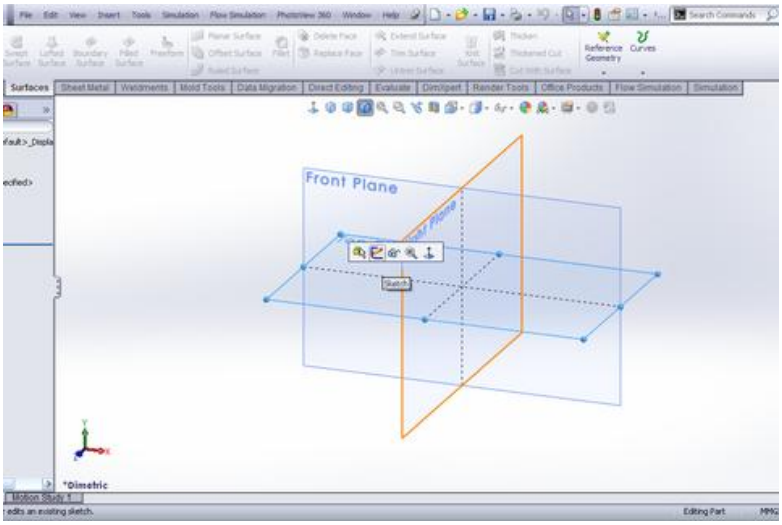


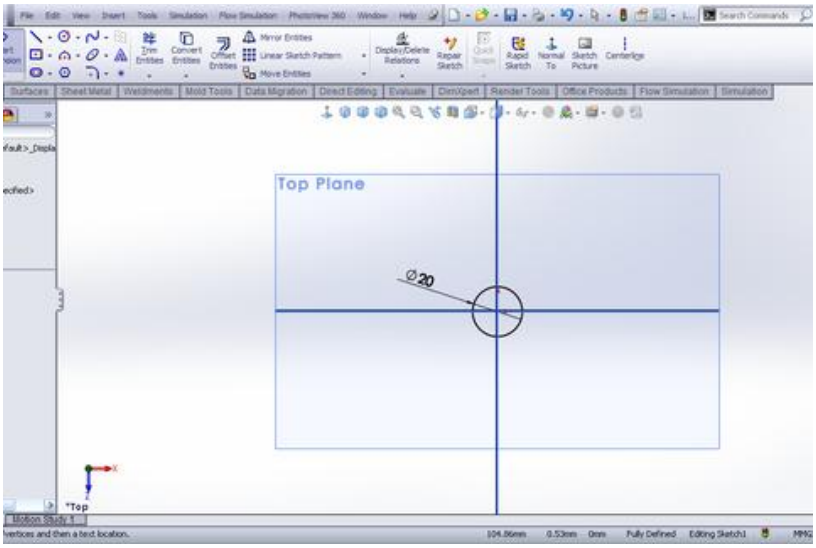
EXPERIMENT:-4

OBJECTIVE:-TO DRAW ACME BOLT

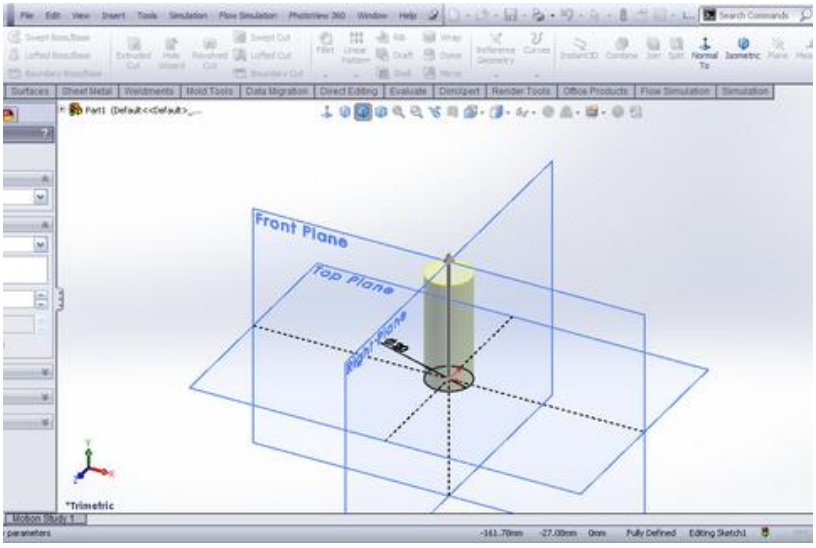
**Step 1-** Top plane>>Sketch.



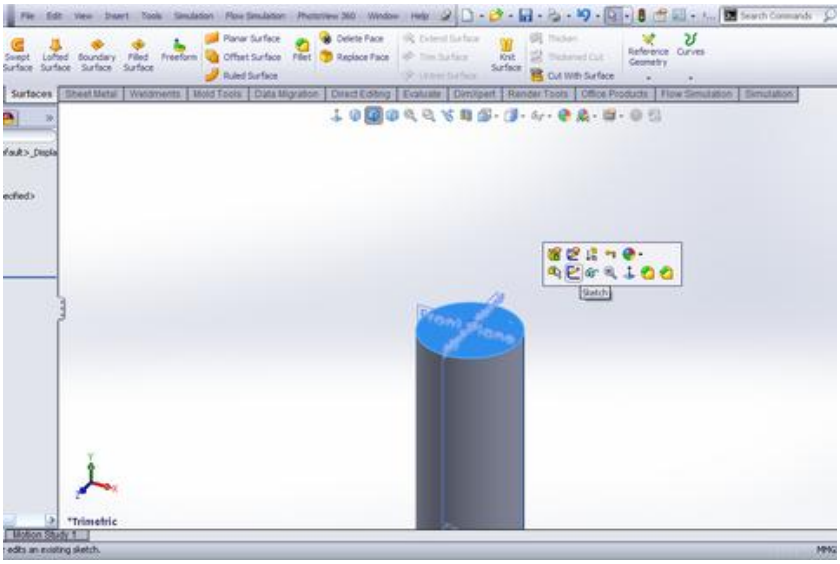

**Step 2-**Draw a circle of 20mm dia.



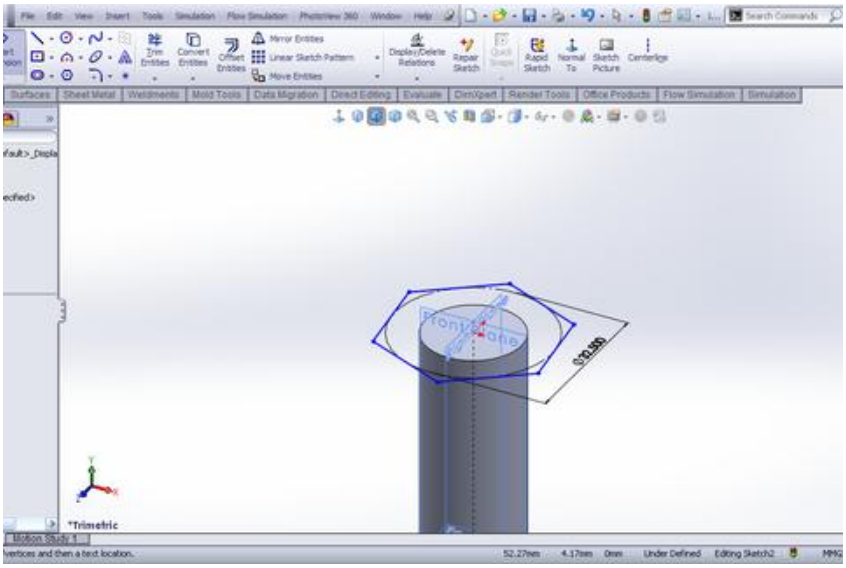

**Step 4-**Extrude it by 50mm.



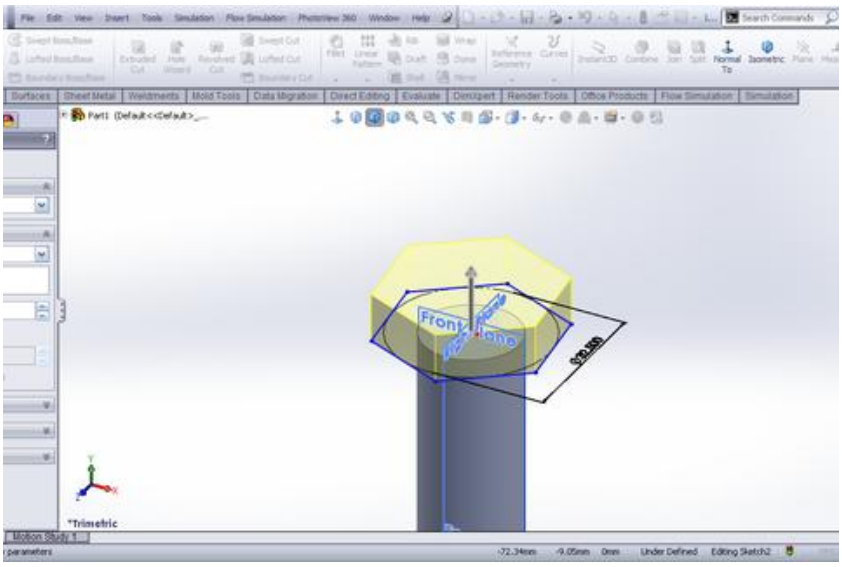

**Step 5**-Top face>>Sketch.



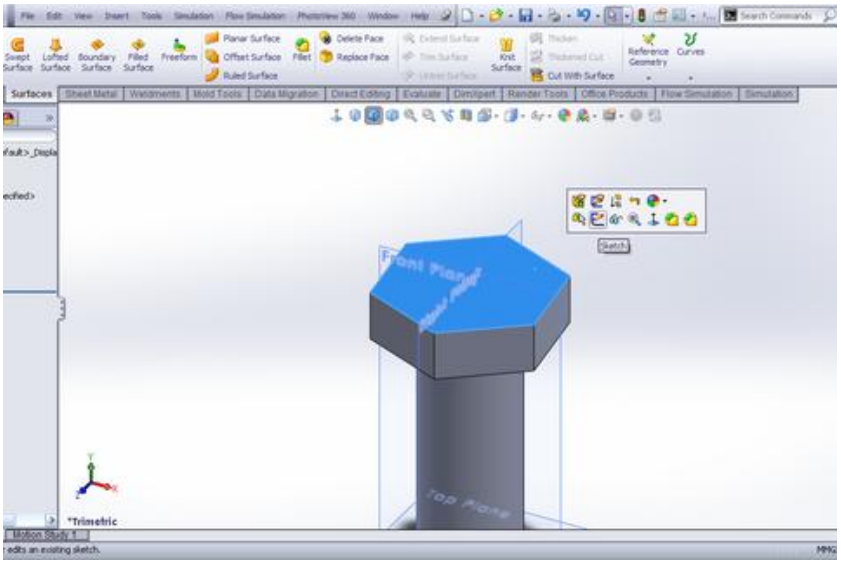

**Step 6**-Draw a polygon inside a circle of 32.5mm dia.



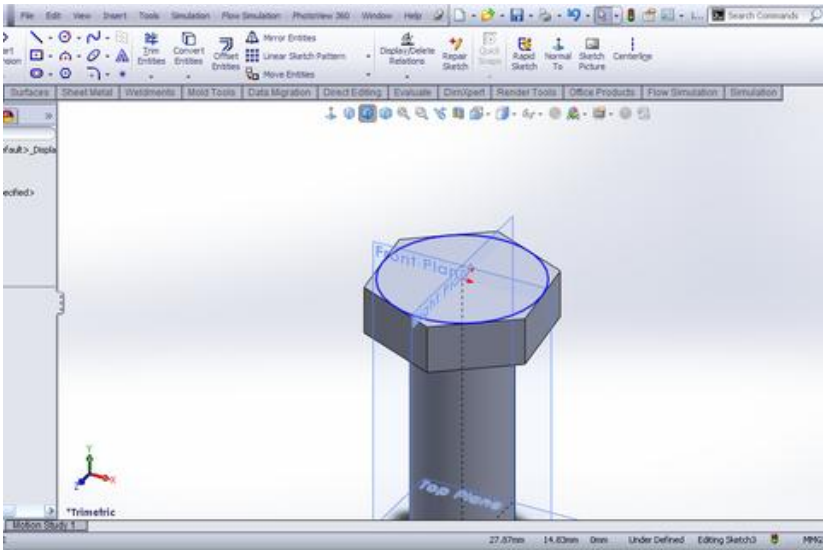

**Step 7**-Extrude it by 10mm.



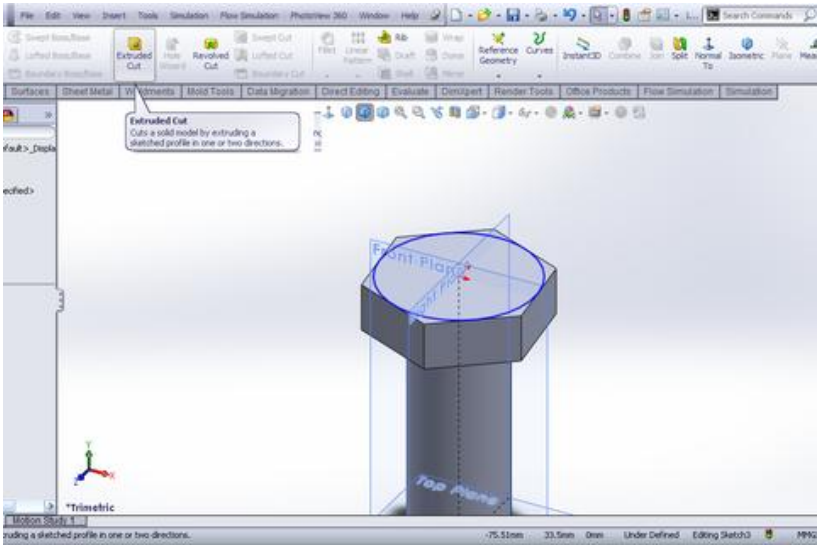

**Step 8**-Top face>>sketch.



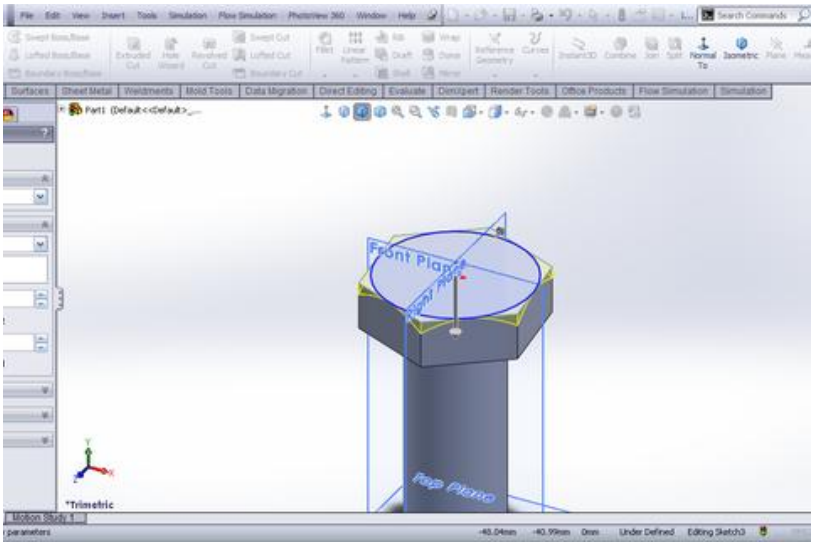

**Step 9**-Draw a circle tangent to sides of polygon.



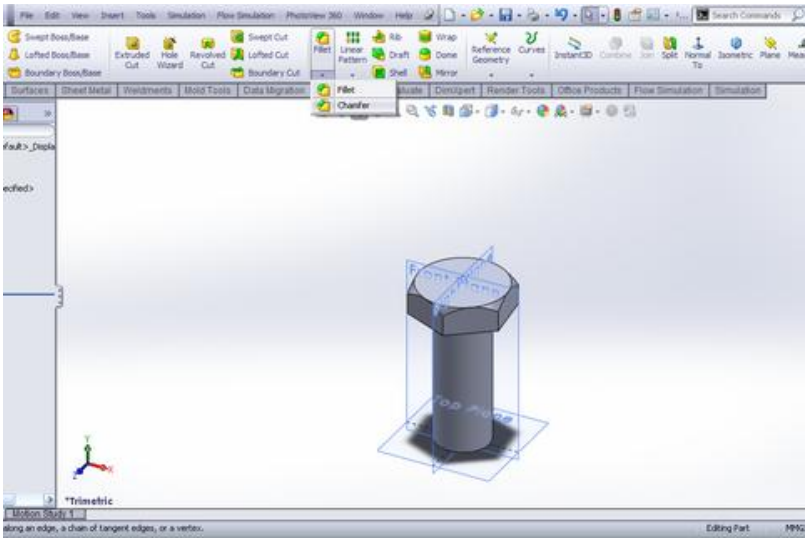

**Step 10**-Extruded cut.



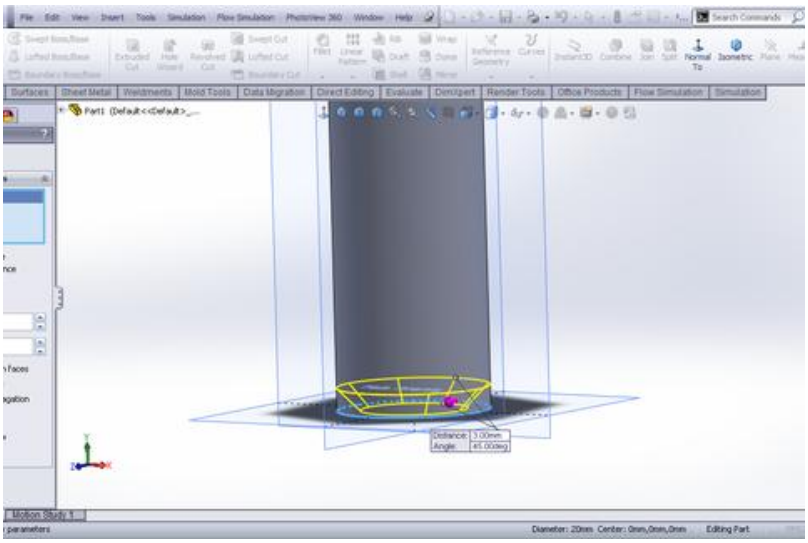

**Step 11-**Check flip side to cut. Draft enable at 45.00º.



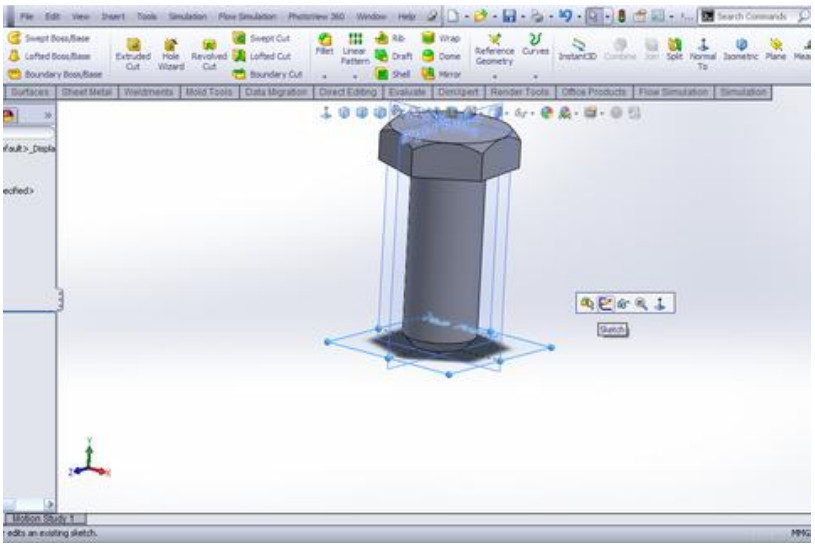

**Step 12-**Chamfer.



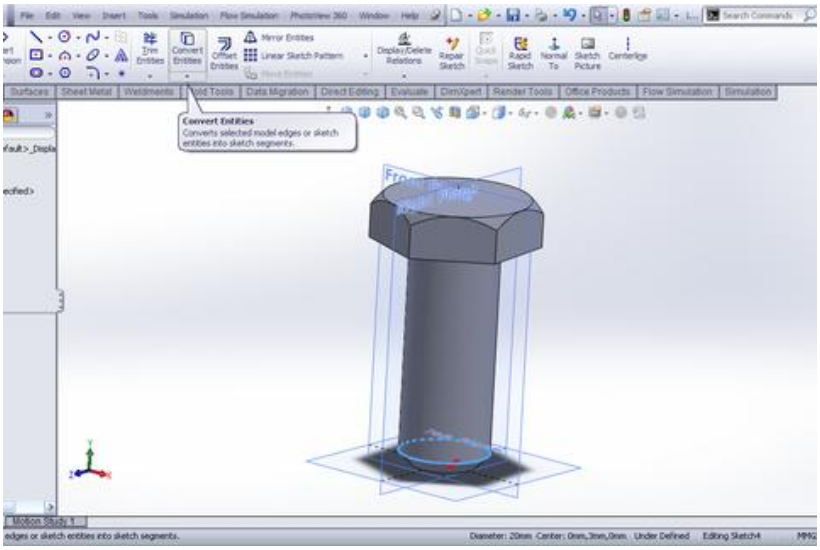

**Step 13-**At distance of 3mm & angle 45º.



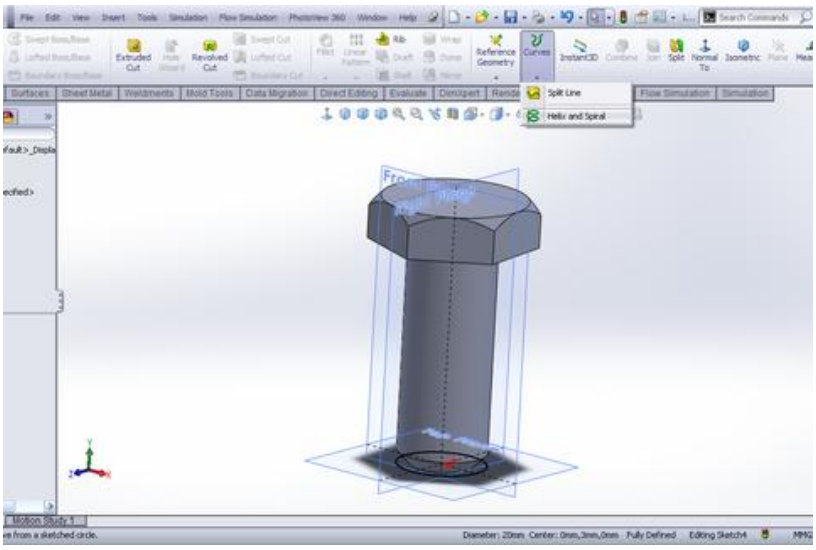
**Step 14-**Top plane>>sketch.



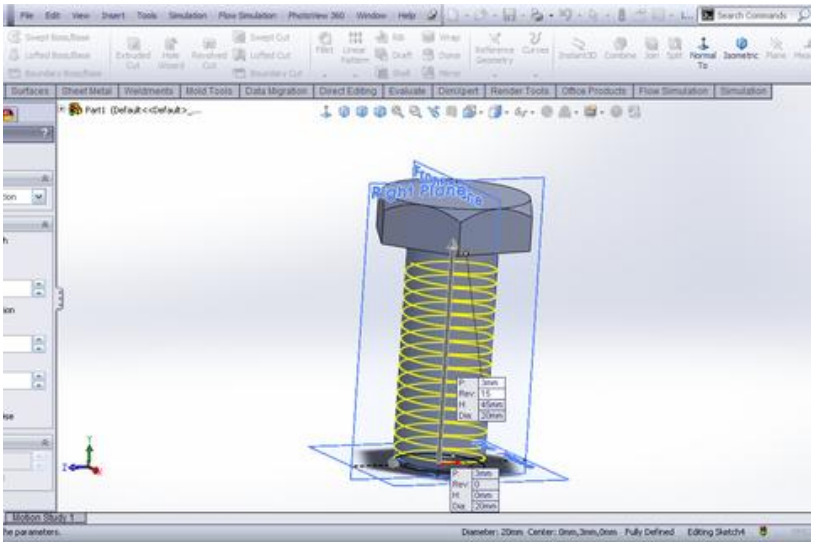
**Step 15-**Select the outer edge and then convert entities.



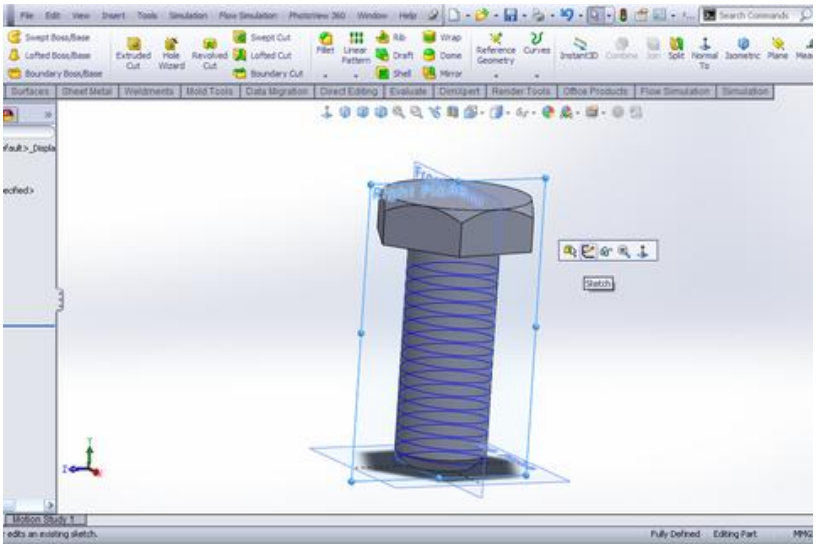

**Step 16-**Curves>>Helix and spiral.



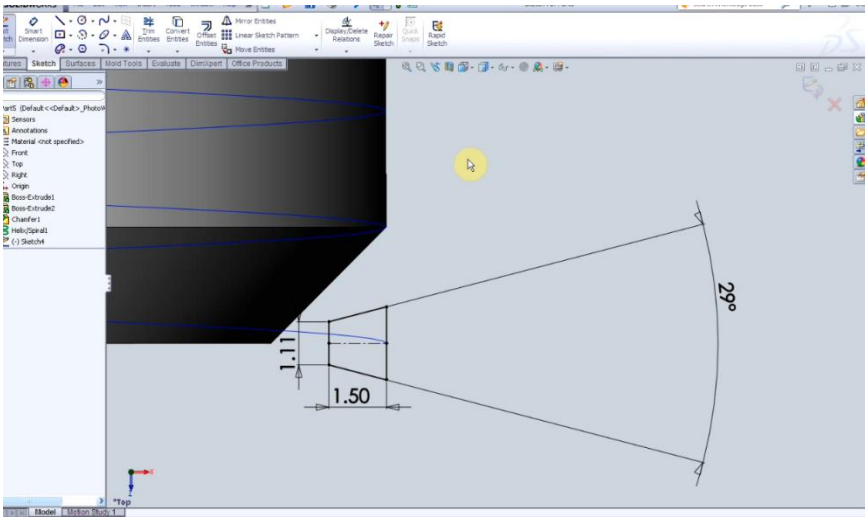

**Step 17**-Defined by Pitch and revolution. Pitch=3mm and revolution = 15.



**Step 18**-Right plane>>sketch.

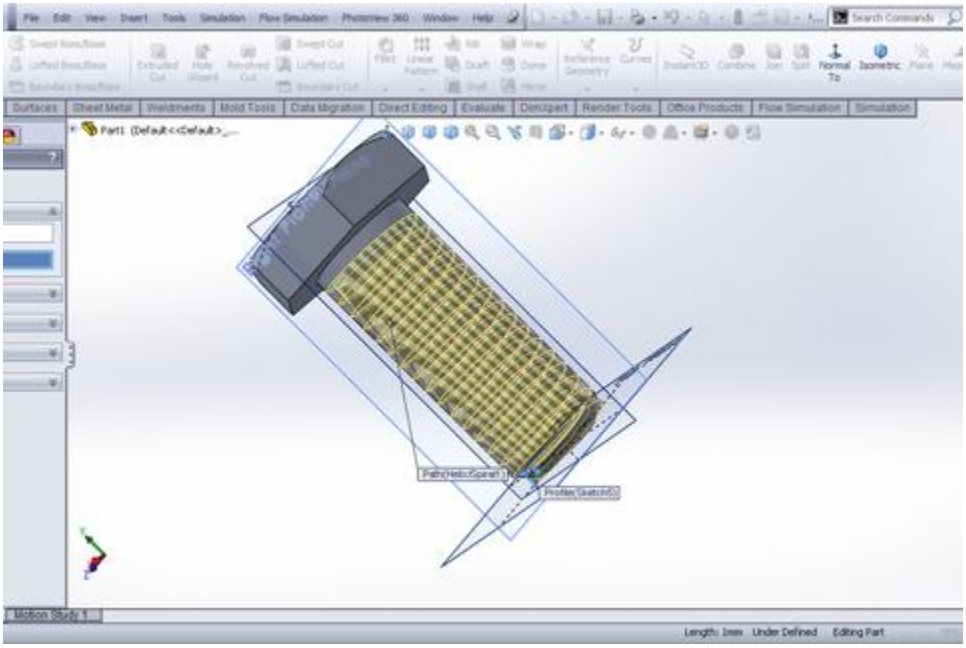


**Step 19**-Draw a profile like this. Since pitch is 3mm so keep the length be little less than pitch since at 3mm it will have intersection error.

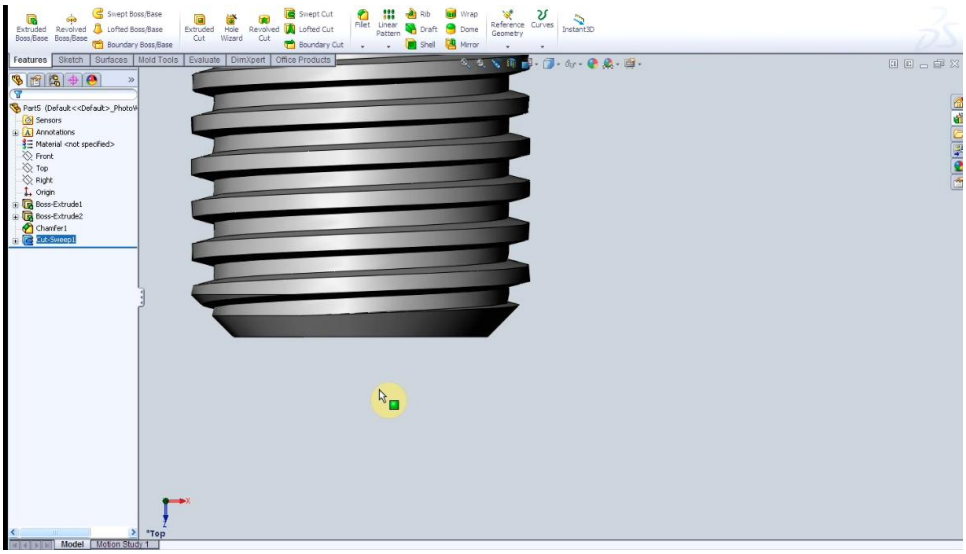



**Step 21**-Sweep cut.

Select the profile and the helix as the path.



**Step 22**-And we have the ACME screw threads.

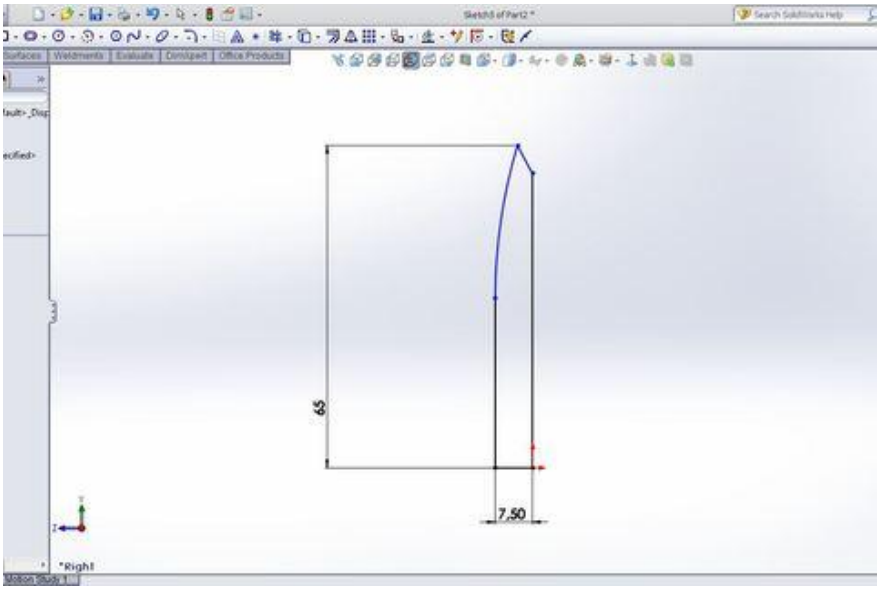


## EXPERIMENT:5

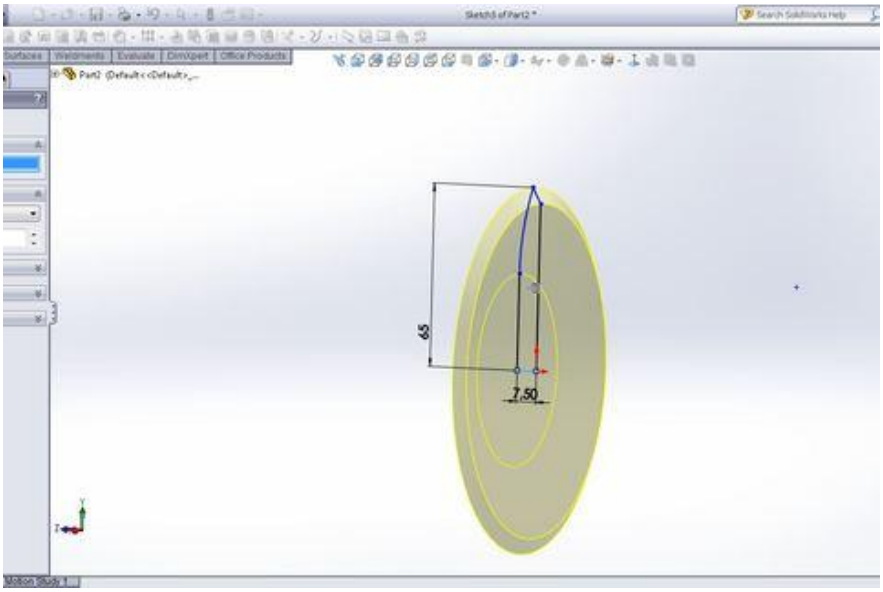
**Objective- Draw 3-D model of Crank-shaft**

THEORY- The crankshaft transforms the linear motion of the pistons into a rotational motion that is transmitted to the load. Crankshafts are made of forged steel. The forged crankshaft is machined to produce the crankshaft bearing and connecting rod bearing surfaces. The rod bearings are eccentric, or offset, from the center of the crankshaft. This offset converts the reciprocating (up and down) motion of the piston into the rotary motion of the crankshaft. The amount of offset determines the stroke (distance the piston travels) of the engine.

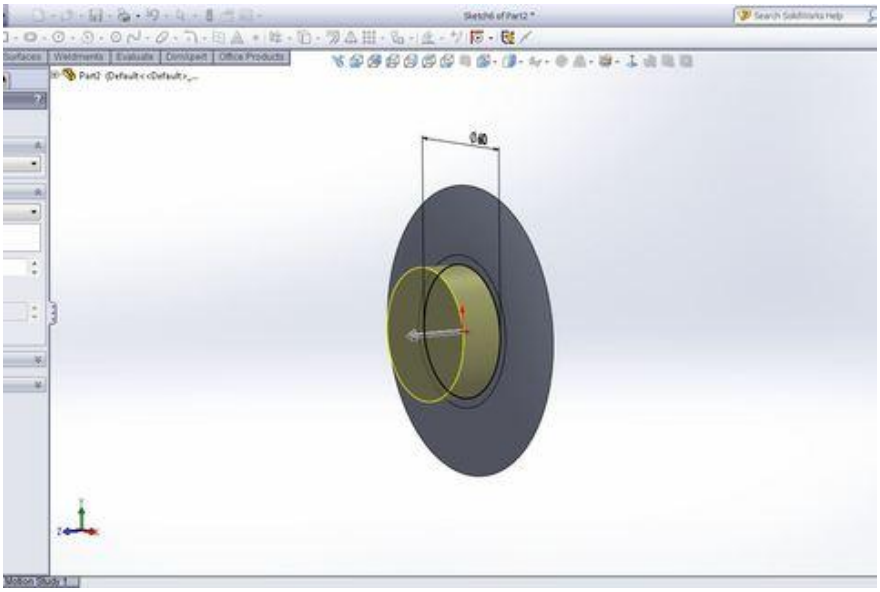
**Step 1-**Open a sketch on the right plane and draw a sketch like shown



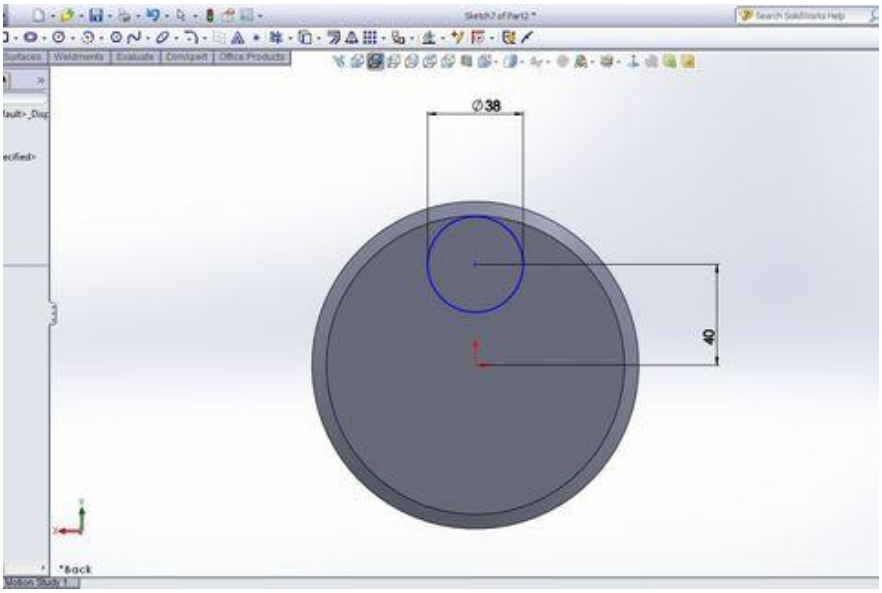

**Step 2**- Use revolve command



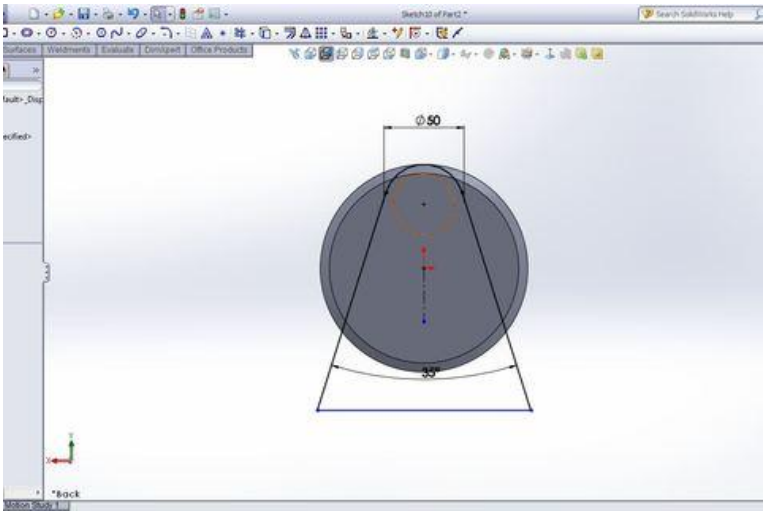
**Step 3-**Open a sketch on the body. Draw a 30mm radiuses circle and extrude it 20mm



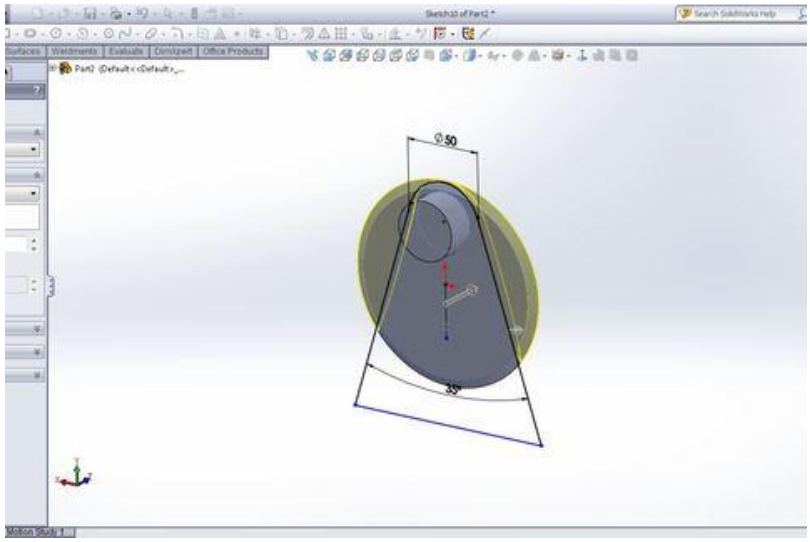
**Step 4-**Go to other side draw shown sketch and extrude it 20 mm



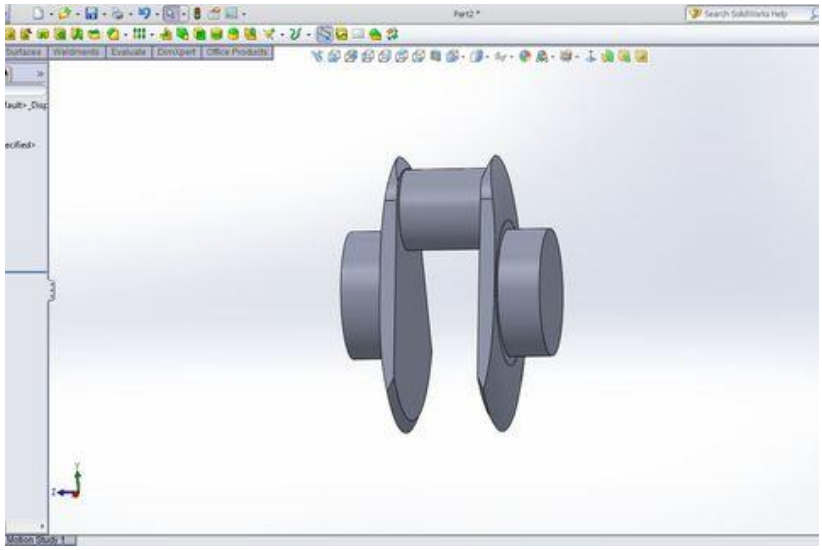

**Step 5-**Open a sketch on same surface and draw shown sketch



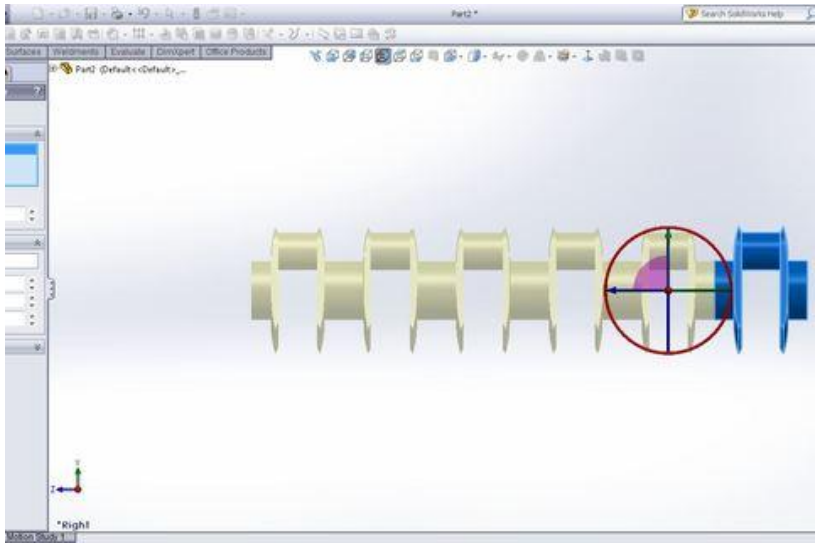
**Step 6**-Use cut extrude command and tick 'flip side to cut'



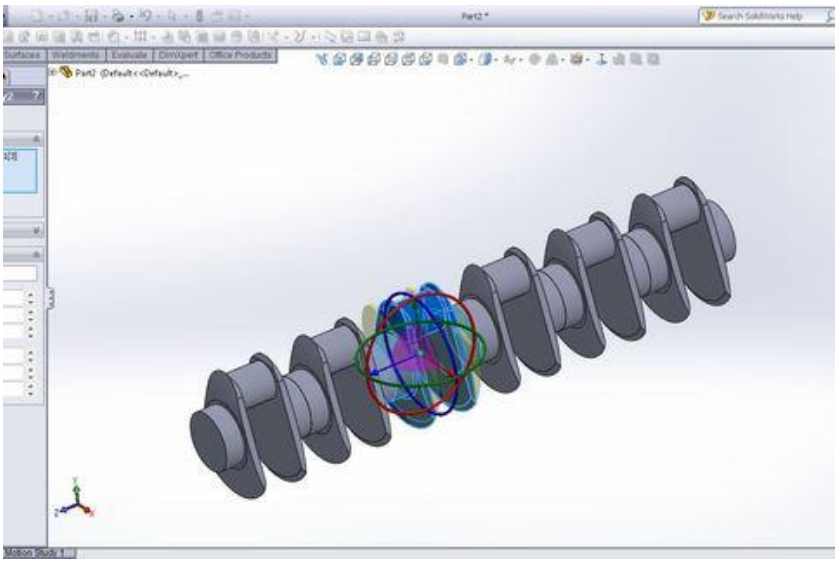

**Step 7**-Now use mirror command. crank began to take shape



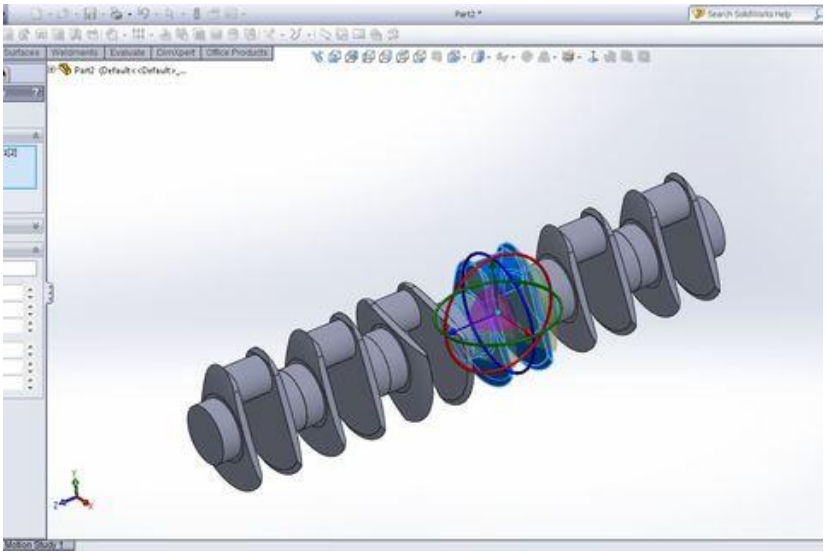
**Step 8**-Use move copy body command. Enter values



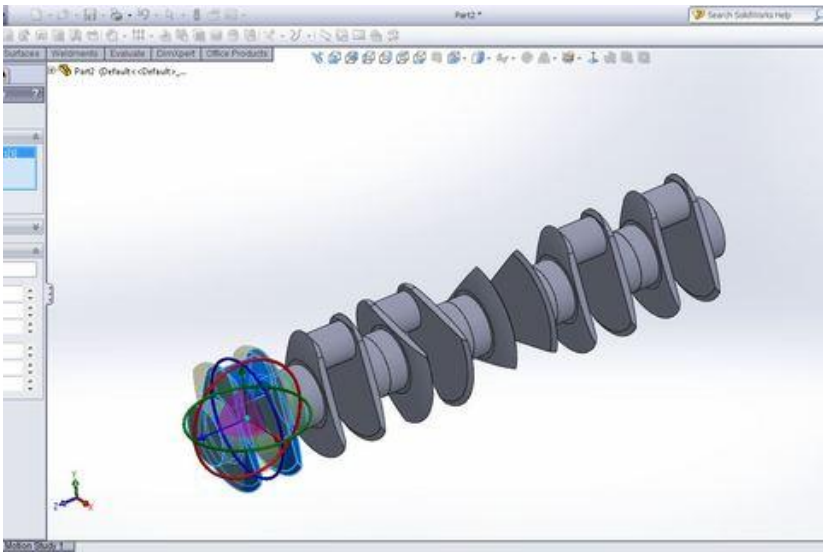

**Step 9**-Rotate this (4th from right side) body 60degree



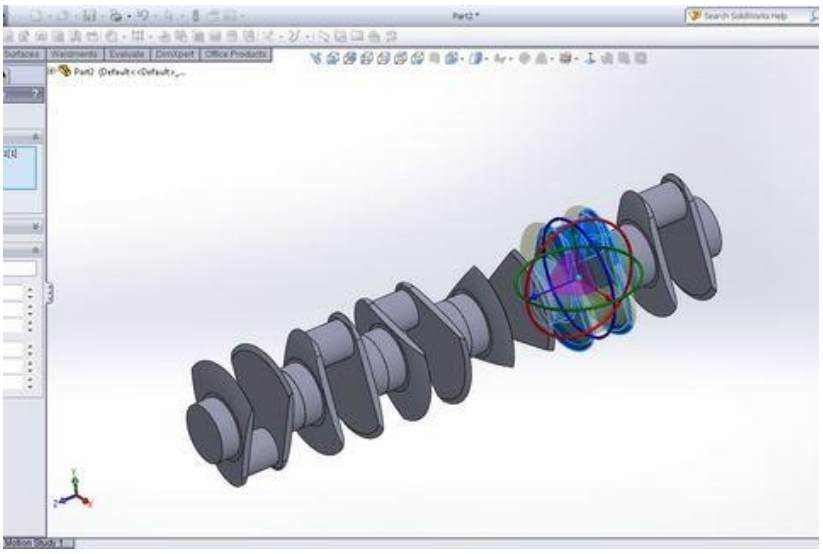
**Step 10**-Rotate 3th body 120 degree



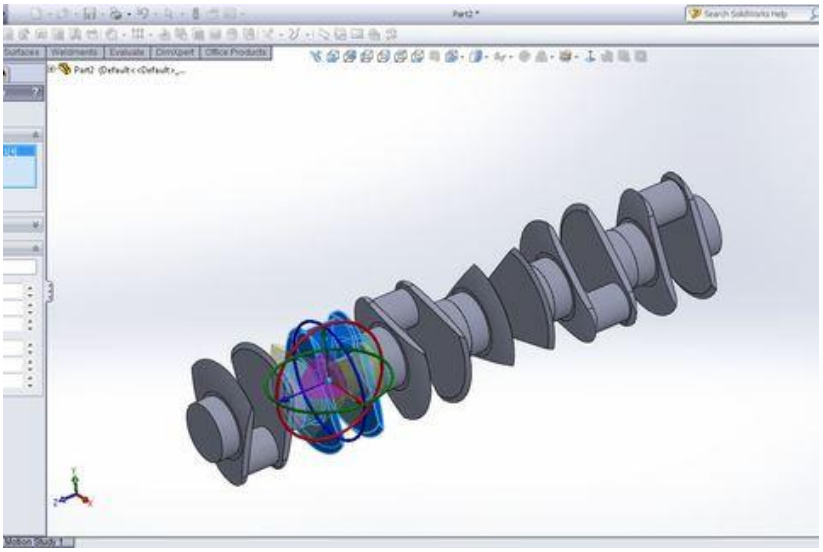
**Step 11**-Rotate sixth body 180 degree



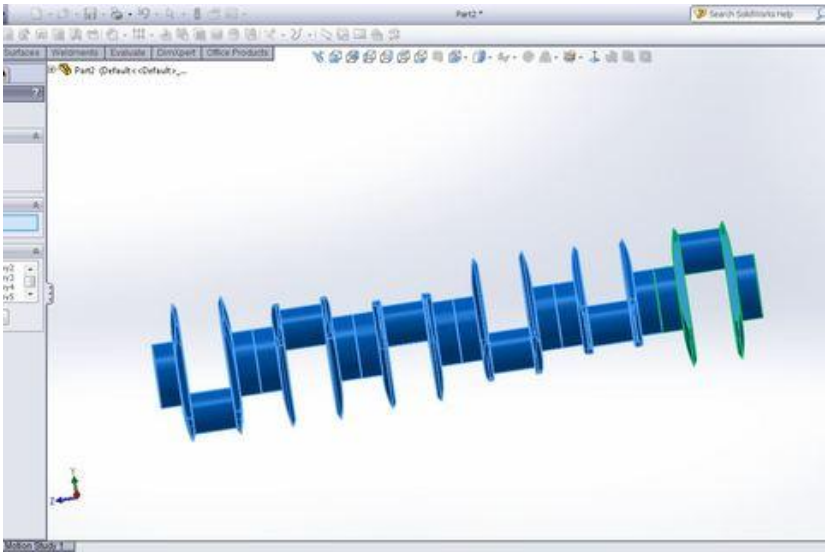
**Step 12**-Rotate second body 240 degree



**Step 13-**Rotate 5th body 300 degree



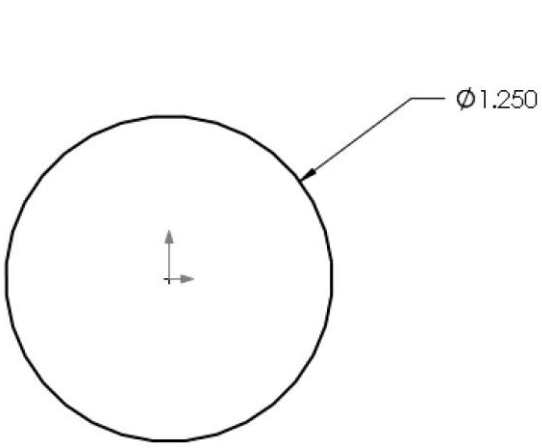
**Step 14-**Use combine command. Crankshaft is ready

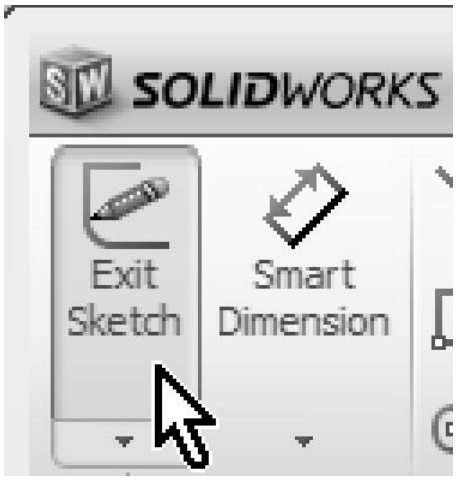


**EXPERIMENT:6**

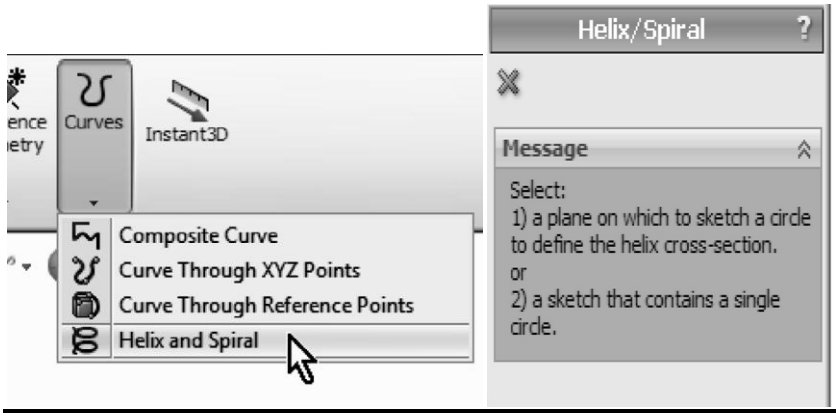
**Objective- Draw a Spring using helix command**

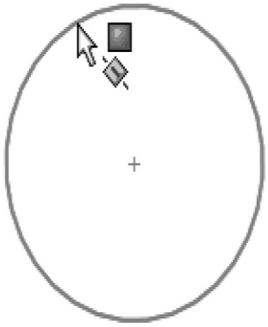
**Step 1-**In order to make a helix, first we need to make a sketch with a circle. This Circle is going to be the helix's diameter. Select the Front Plane and make a sketch using the following dimensions. Exit the sketch when done.



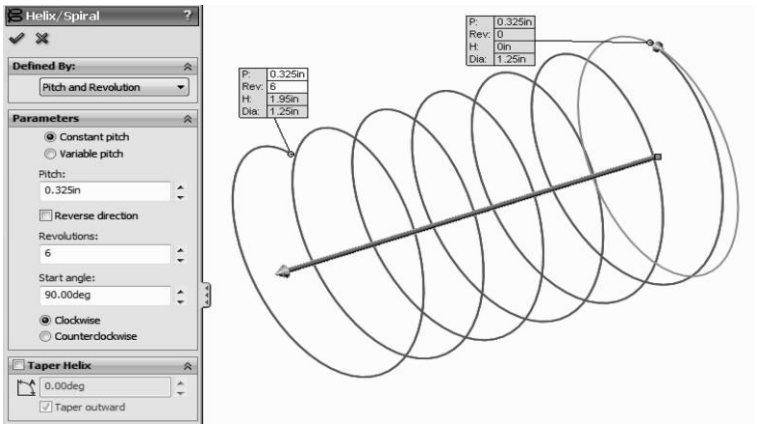




**Step 2-**In the Features tab select "**Curves**, **Helix and Spiral**" from the drop-down menu. When asked to select a plane or a sketch, select the sketch we just drew

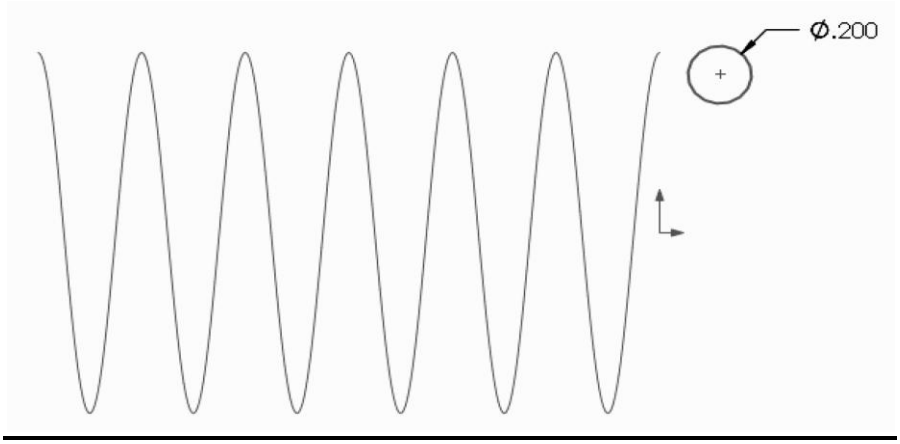


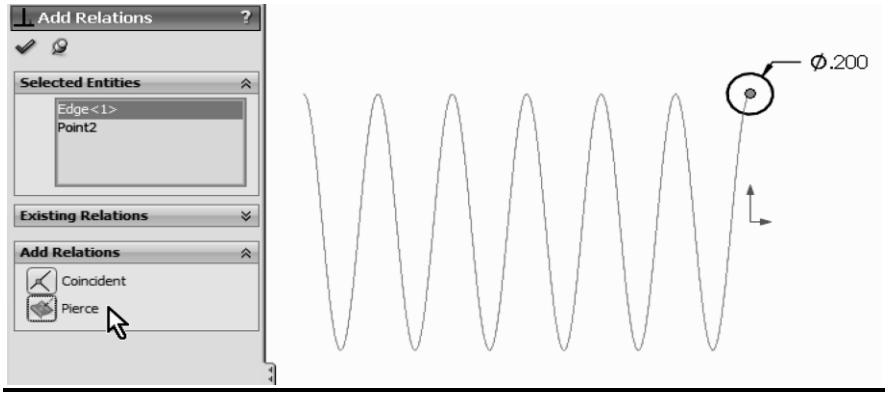
**Step 3-**The helix can be defined by the combination of any two parameters Between Pitch, Revolutions and Height. For this example select "Pitch andRevolution" from the "**Defined By:**" drop down menu, and make the pitch 0.325"and 6 Revolutions. The "Start Angle" value will define where the helix will start.By making it 90 degrees it will start coincident with the Front Plane.




**Step 4-**Once the Helix is done, we need to make the Profile sketch for the sweep.Switch to a Right view, and add a new sketch in the Right Plane as shown. Make the circle close to the Helix .

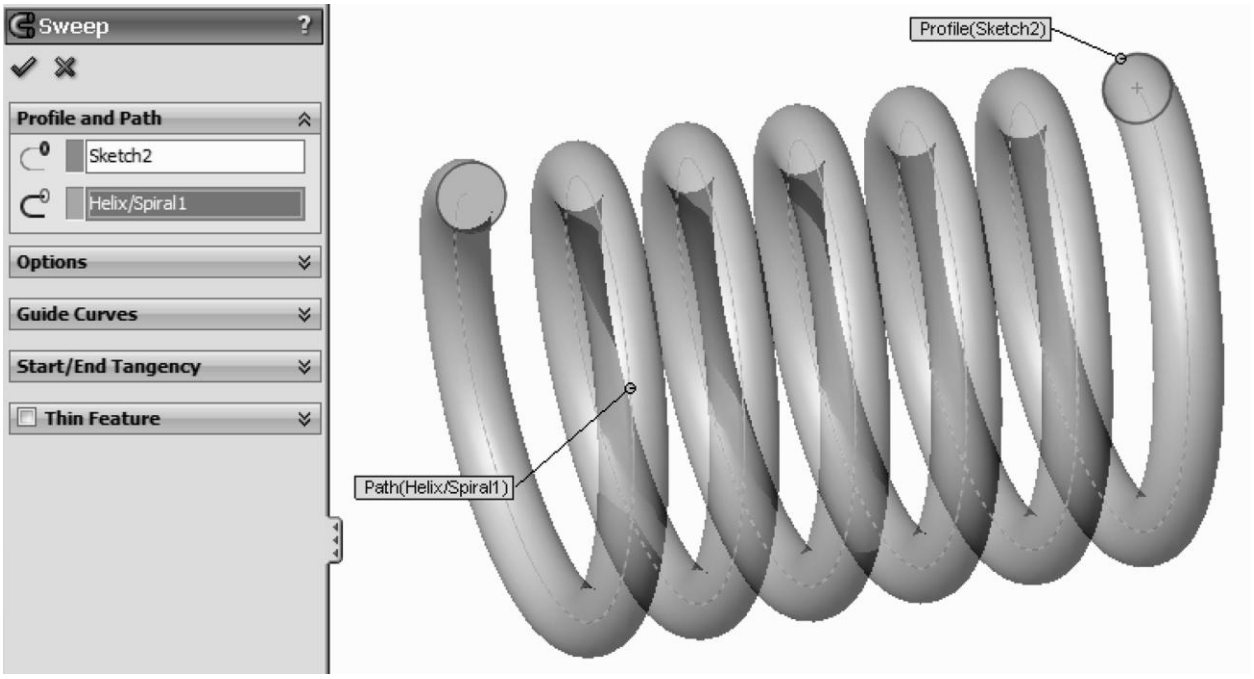



**Step 5-** Add a "**Pierce**" geometric relation between the center of the circle and the helix. This way the path will start at the beginning of the helix. This relation will make the sketch fully defined. Exit the Sketch when done.



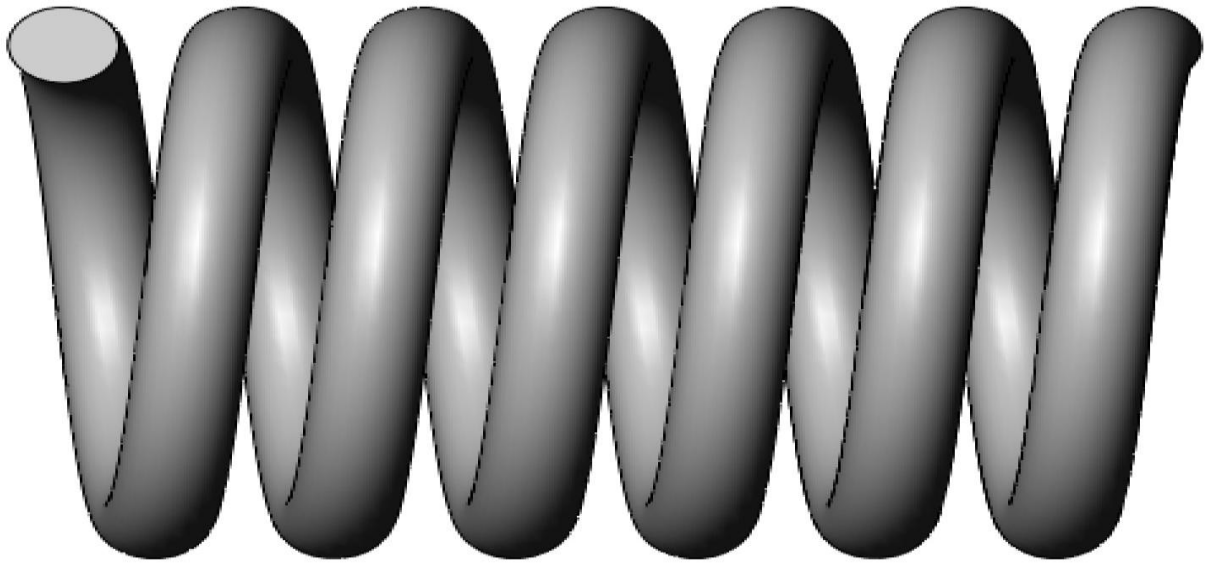
## Step 6

Select the Sweep command and make the sweep using the last sketch as a profile and the Helix as a Path. Notice the Preview.




## Step 7

Finished spring.

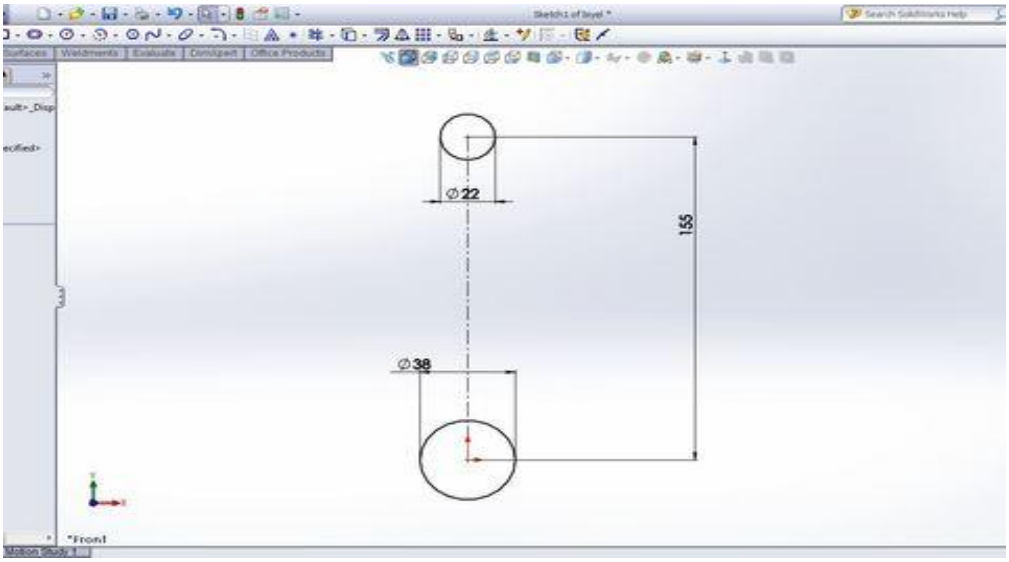


## EXPERIMENT: 7

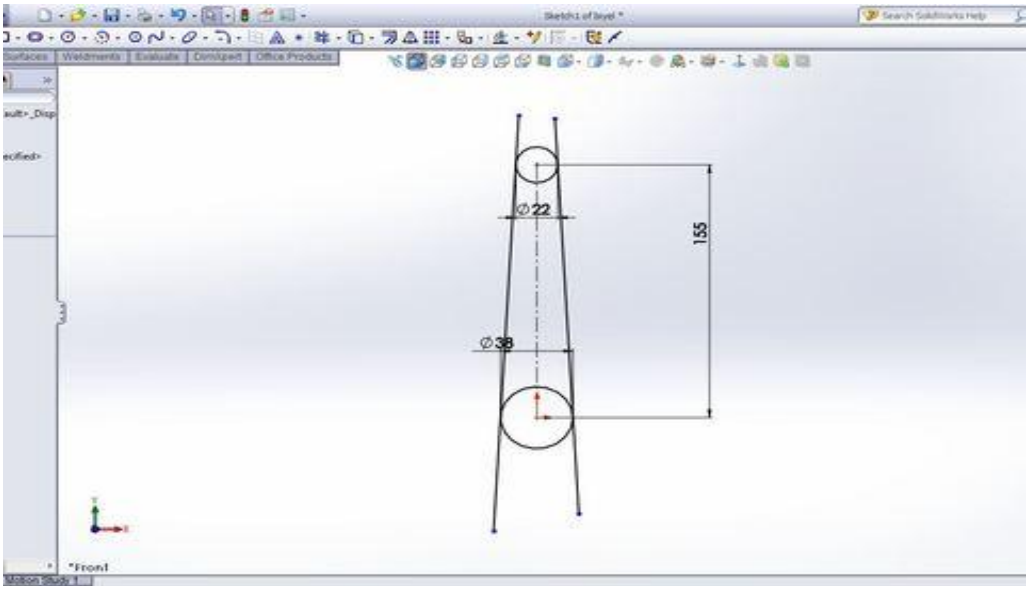
**Objective- Draw 3-D model of Connecting Rod**

THEORY- A *connecting rod* is an engine component that transfers motion from the piston to the crankshaft and functions as a lever arm. Connecting rods are commonly made from cast aluminum alloy and are designed to withstand dynamic stresses from combustion and piston movement

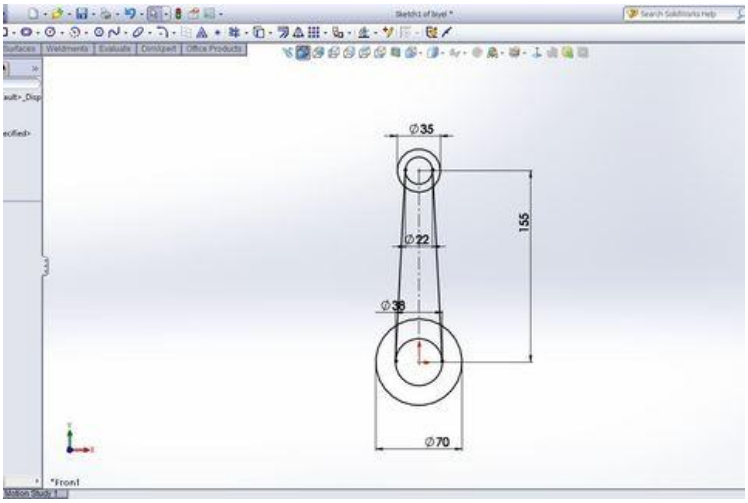
**Step 1-**Draw two circles like shown on the front plane



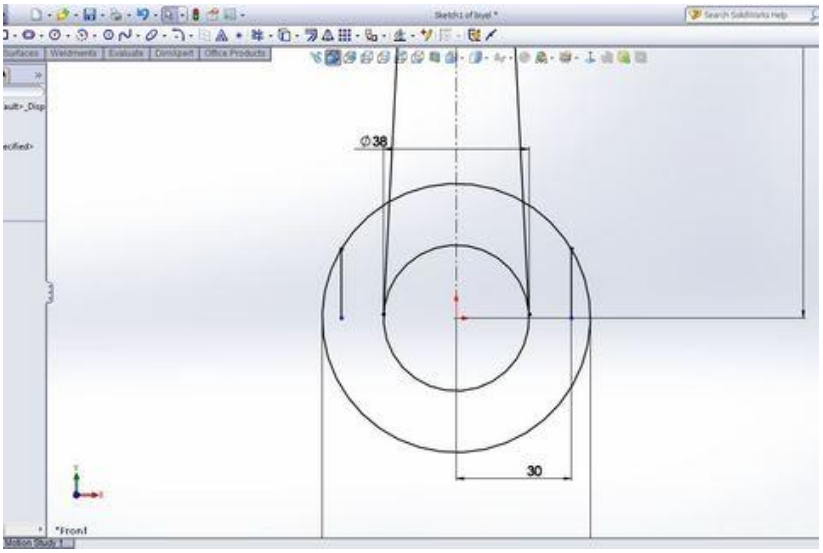

**Step 2-**Add two tangent lines



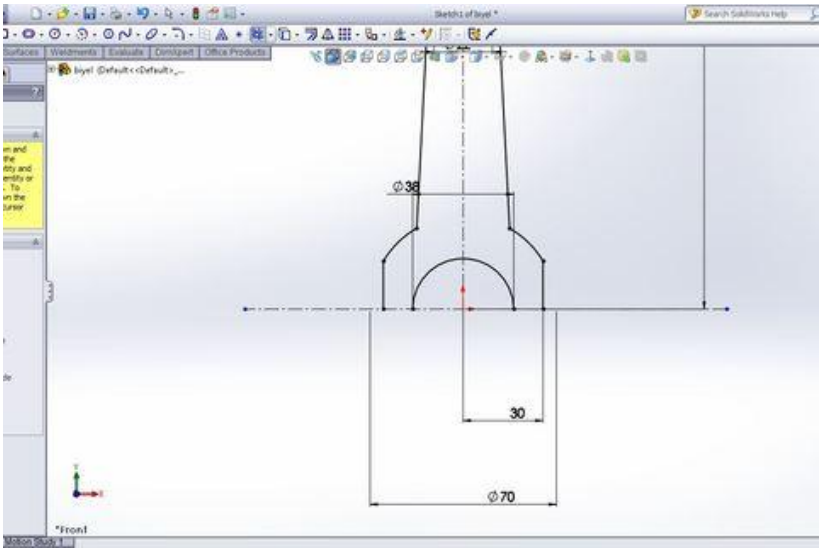
**Step 3-**Add two more circles



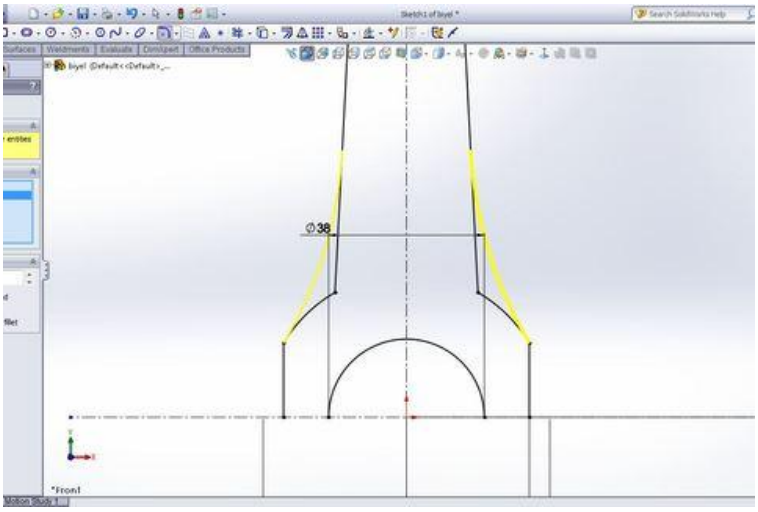

**Step 4-**Draw a line and mirror it



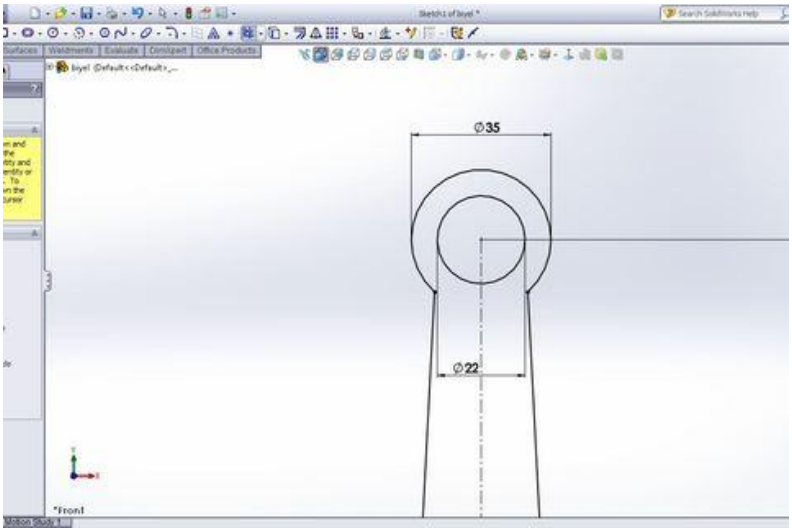

**Step 5-**Draw a centerline and use trim command



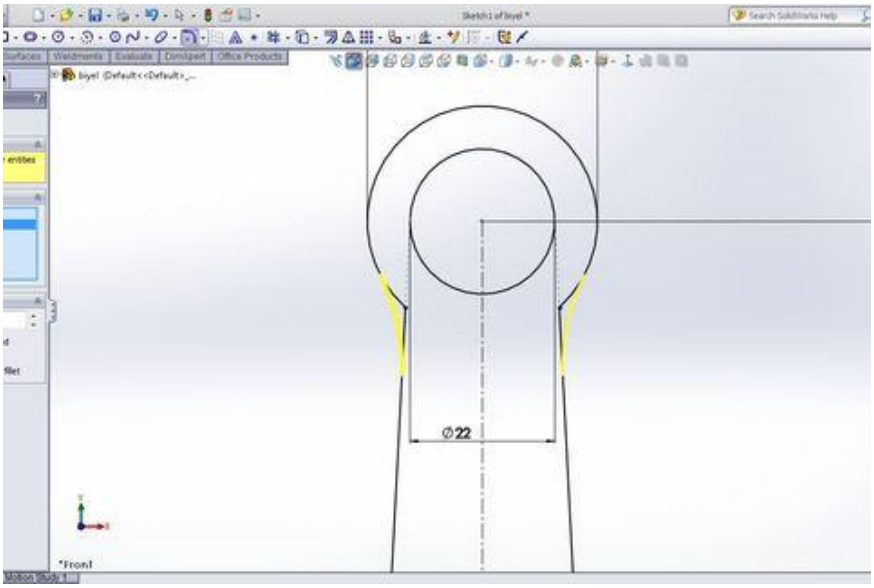

**Step 6-**Use fillet command like shown



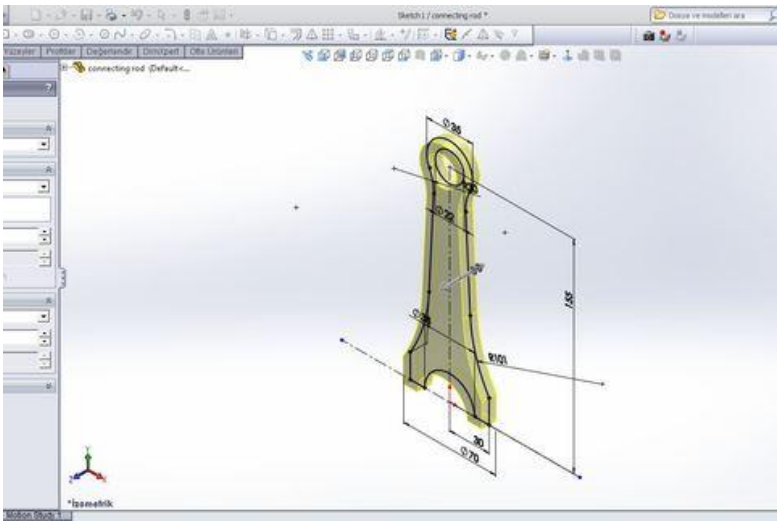

**Step 7-**Go up side and use trim command



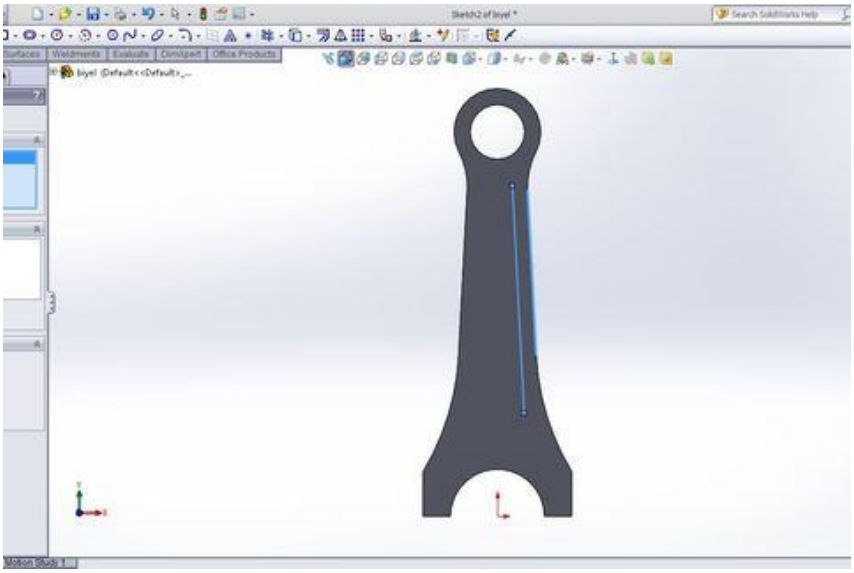

**Step 8-**Use fillet command again



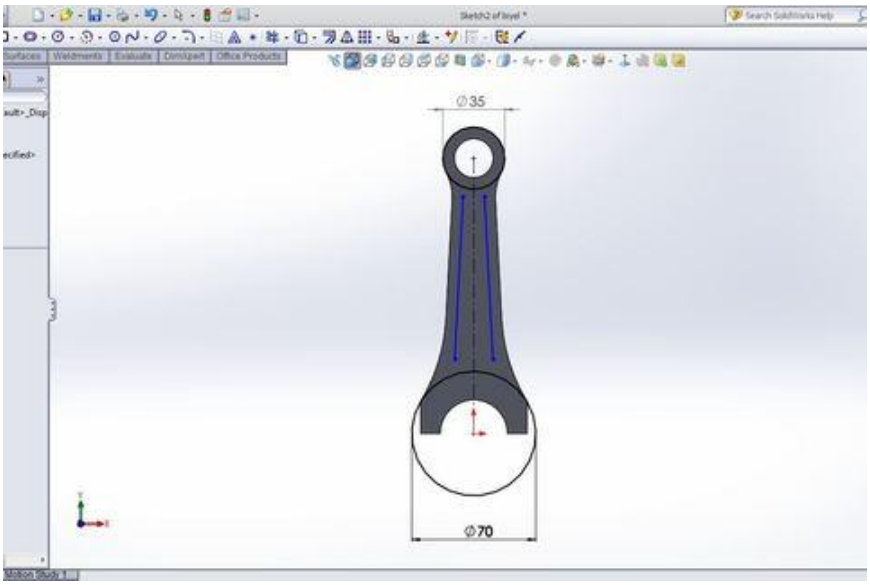

**Step 9-**Open a sketch on the body and draw a line. Make parallel the line with the shown edge



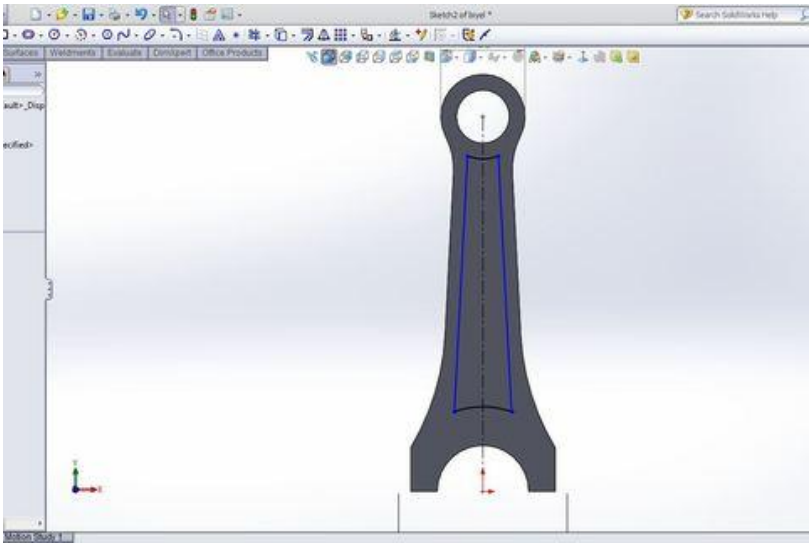
**Step 10-**Open a sketch on the body and draw a line. Make parallel the line with the shown edge



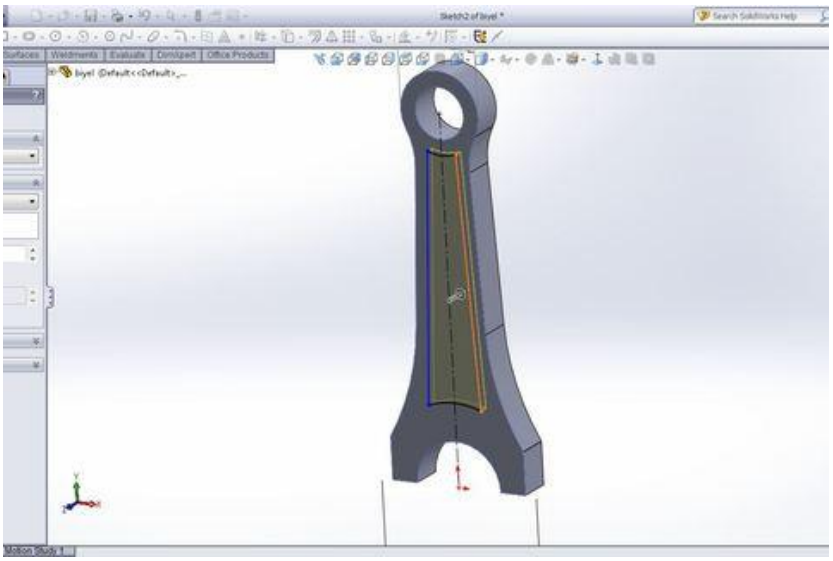
**Step 11-**Mirror the line and add these two circles



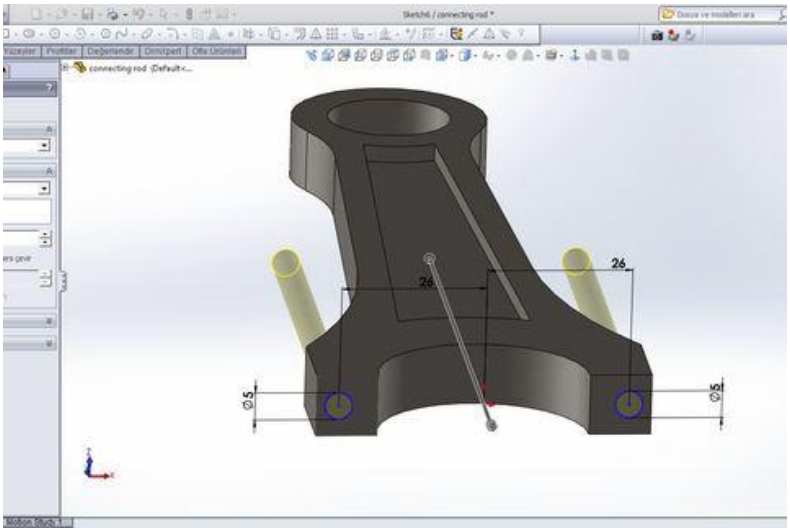

**Step 12-**Intersect the lines and use trim command



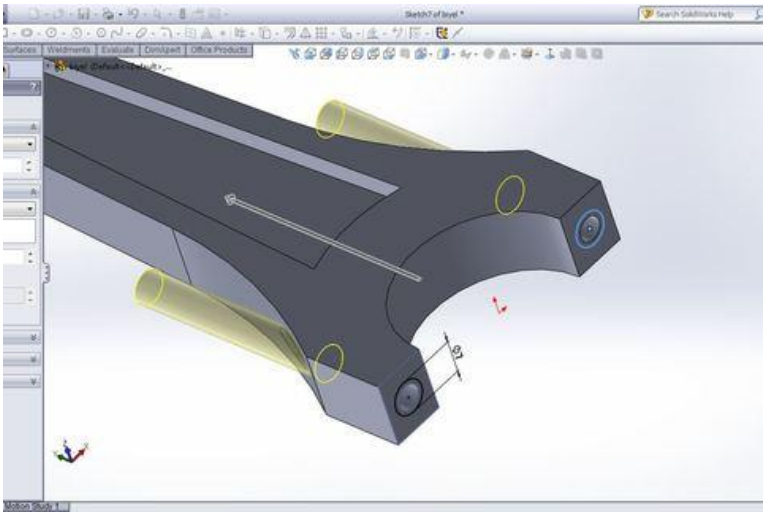
**Step 13-**Use trim command and mirror it



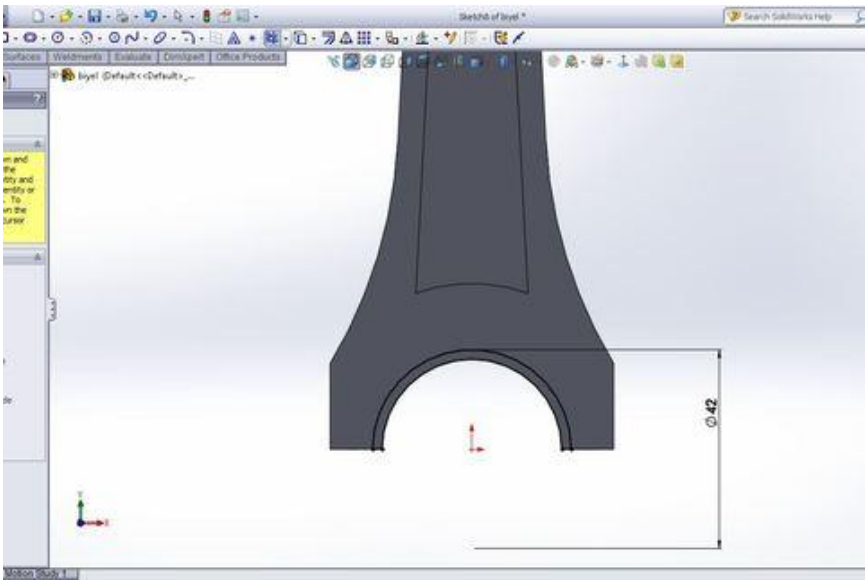
**Step 14-**Draw two circles and cut the body



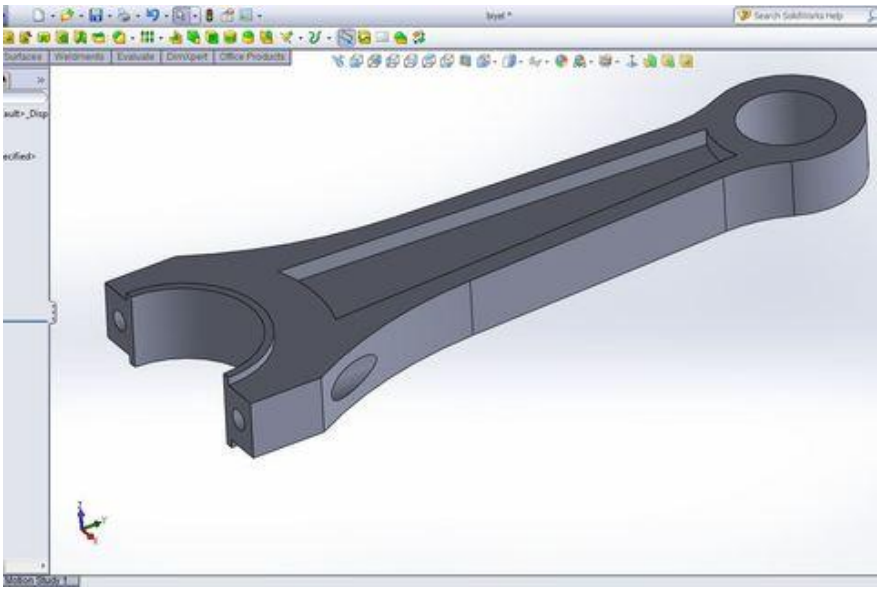

**Step 15-**Draw two circles more and cut the body again. Use offset feature



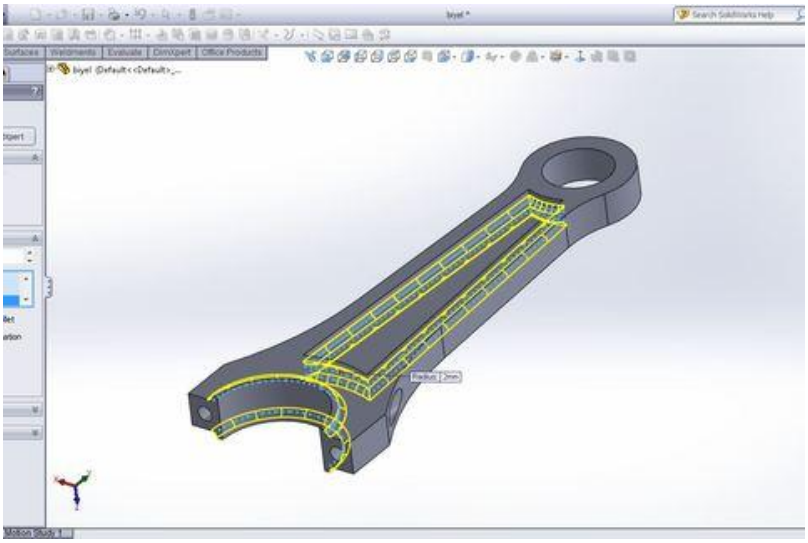
**Step 16-**Draw a sketch on the body like shown



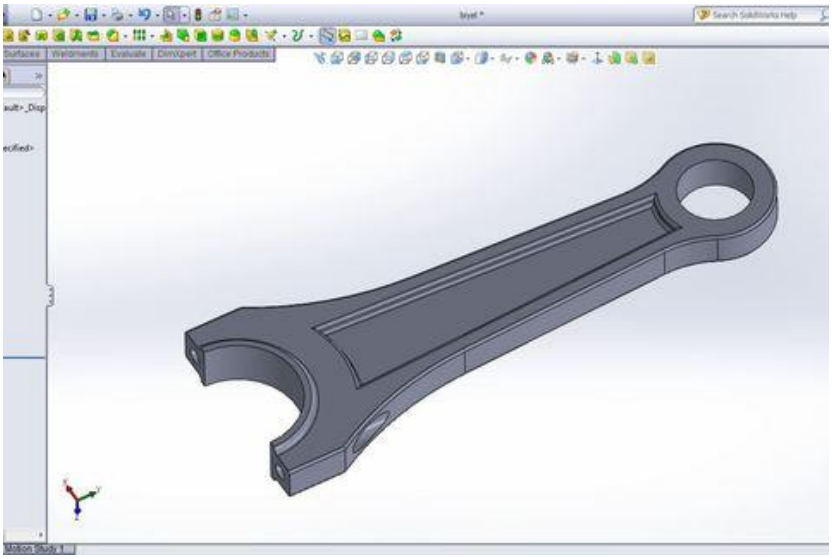

**Step 17-**Extrude it 2mm and use mirror command



**Step 18-**Now use fillet command



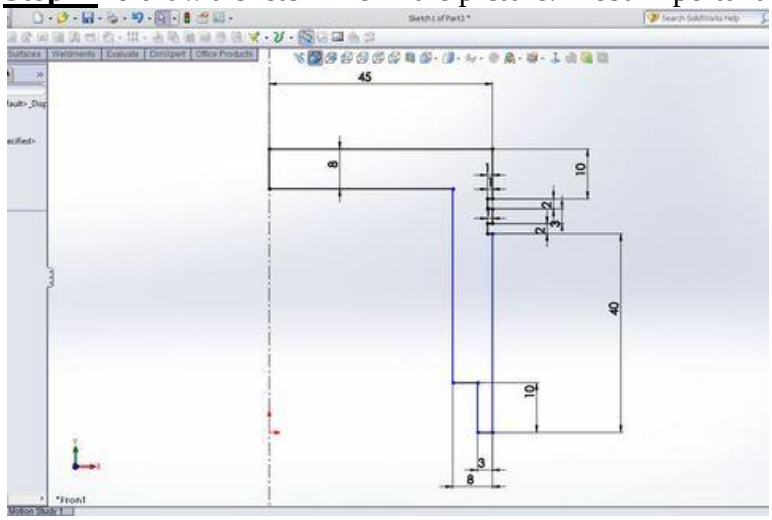
**Step 19-** one part of Connecting rod is ready



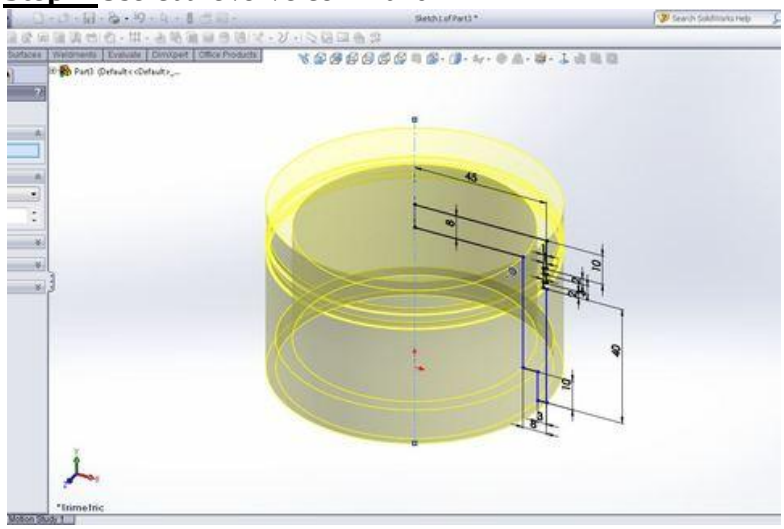
## EXPERIMENT NO:-8

**OBJECTIVE- Draw 3-d model of Piston**

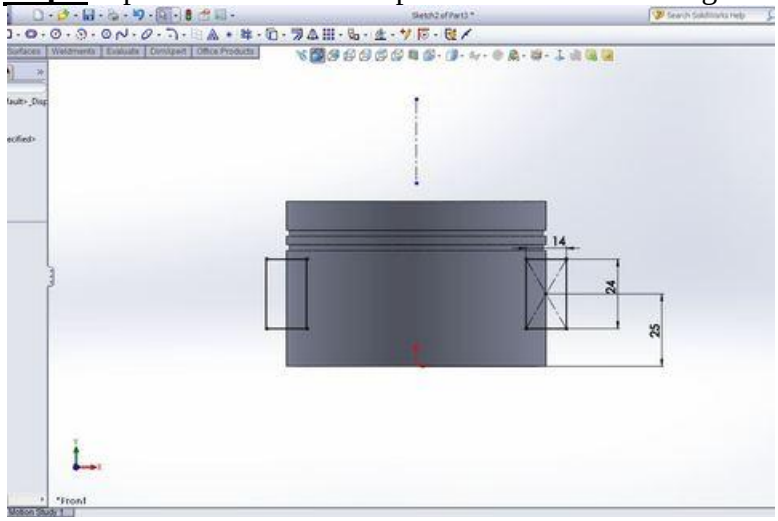
**Step 1**-To draw a sketch like in the picture. Most important dimension is piston radius.



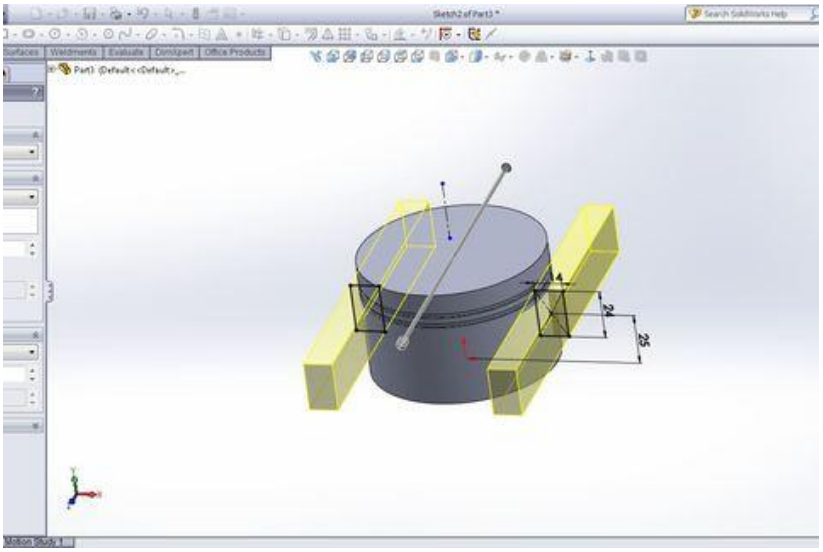

**Step 2**-Use cut revolve command



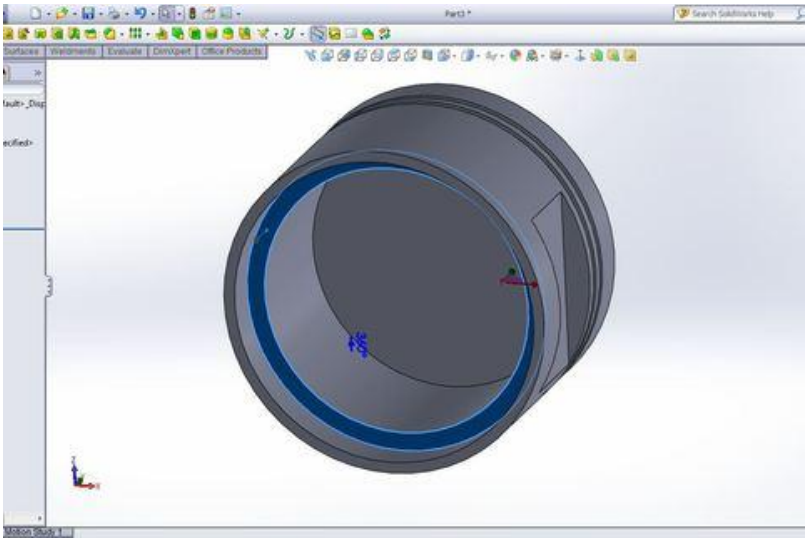

**Step 3**-Open a sketch on front plane and draw a rectangle



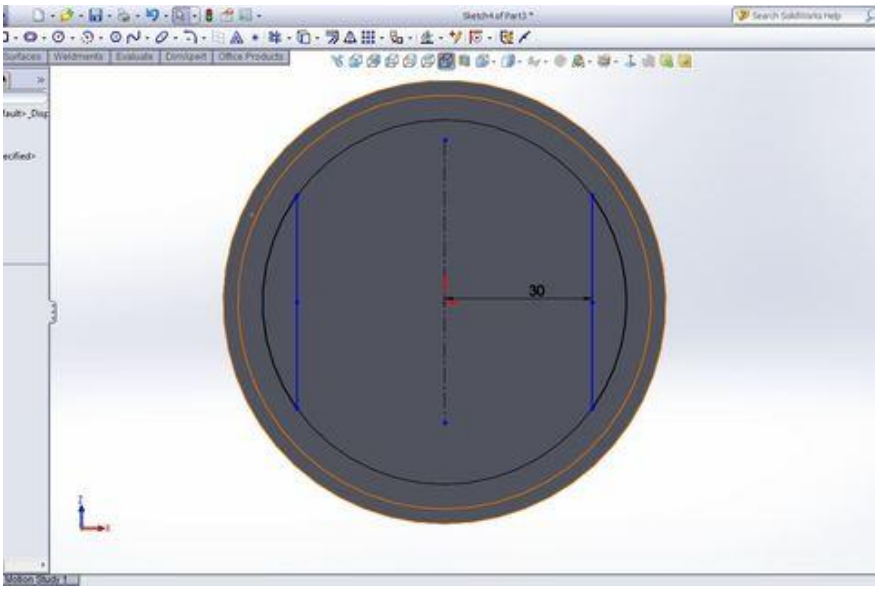
**Step 4-**Use cut extrude command and cut the body



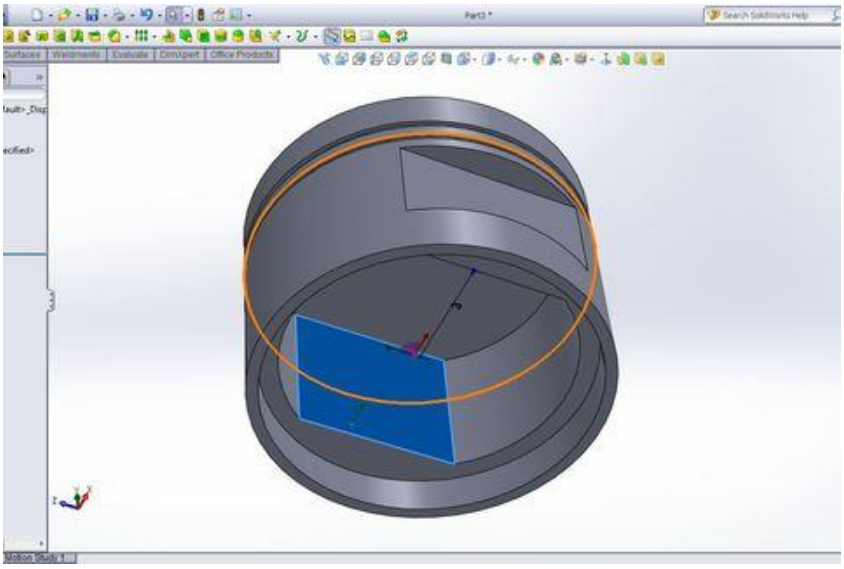

**Step 5-**Open a sketch on this surface



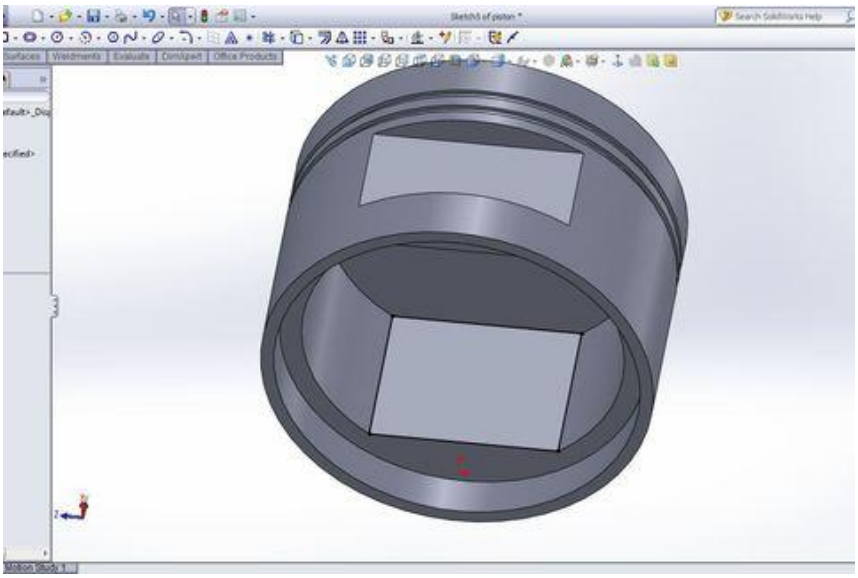
**Step 6-**Draw this sketch with using mirror command



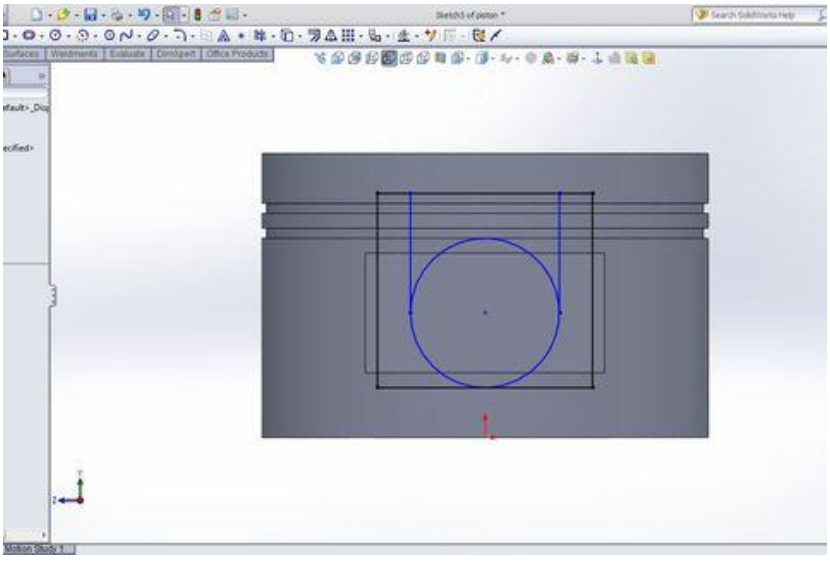

**Step 7-**Extrude it 40mm. then open a sketch on this surface



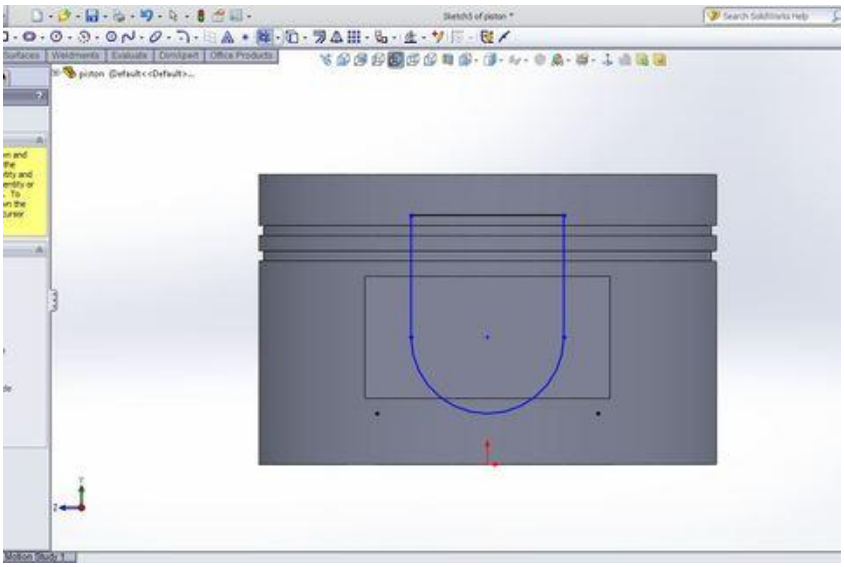
**Step 8-**Use convert entities command and select this surface



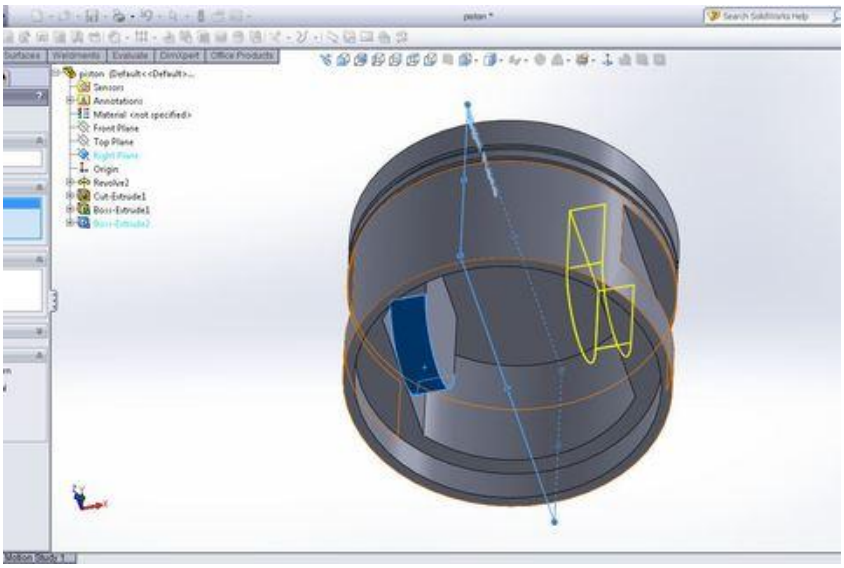
**Step 9-**Draw the shown sketch



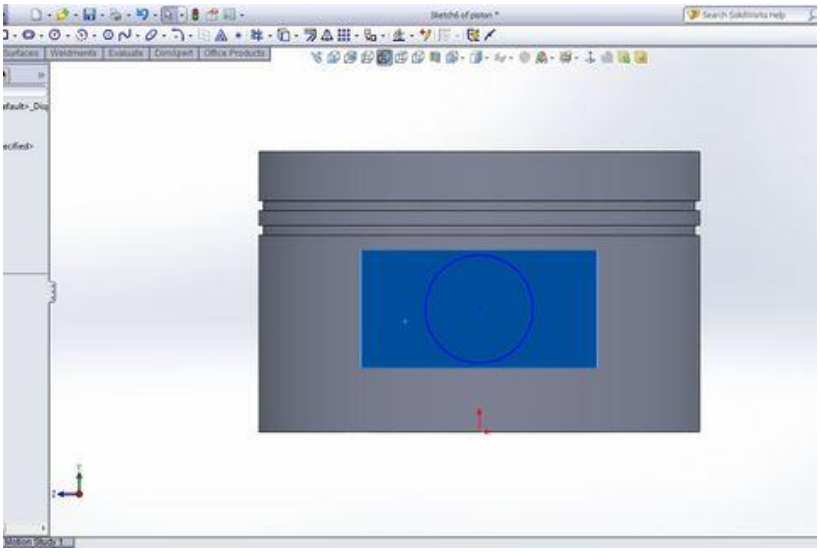
**Step 10-**Use trim command



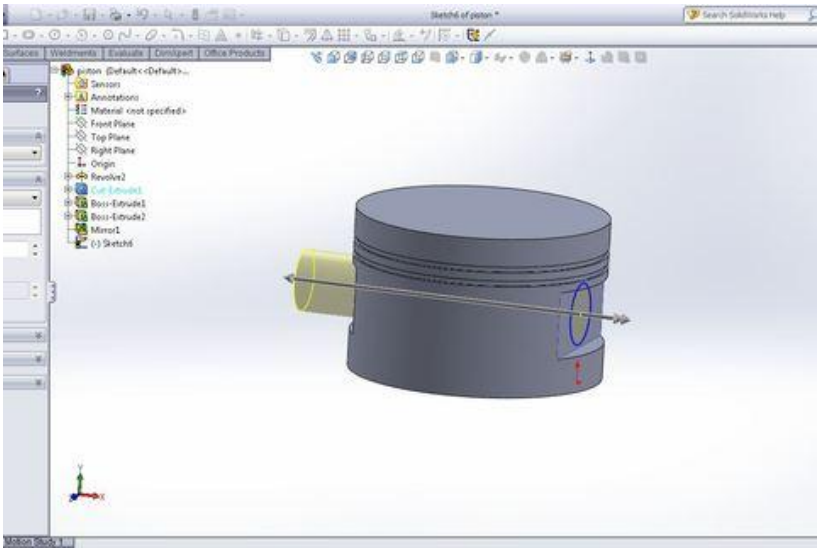
**Step 11-**Extrude 10mm and mirror it



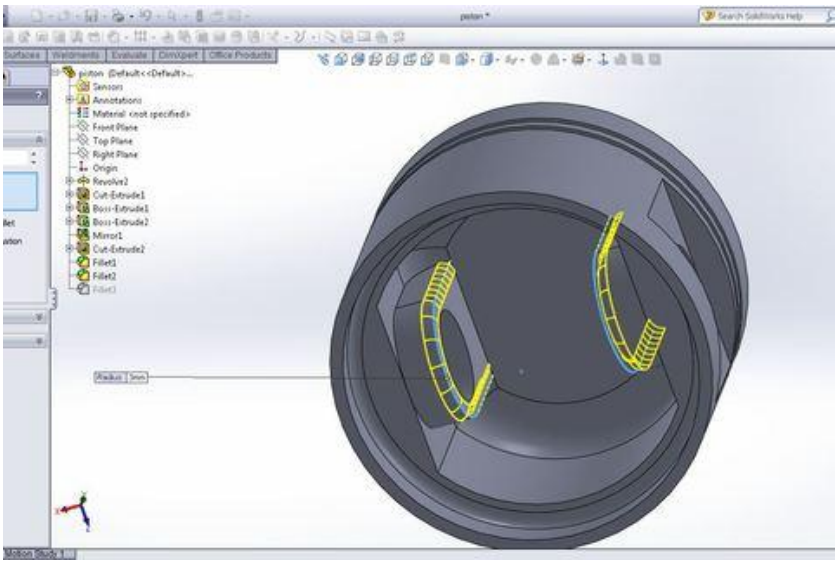
**Step 12-**Open a sketch on blue surface and draw a 11mm circle center of it



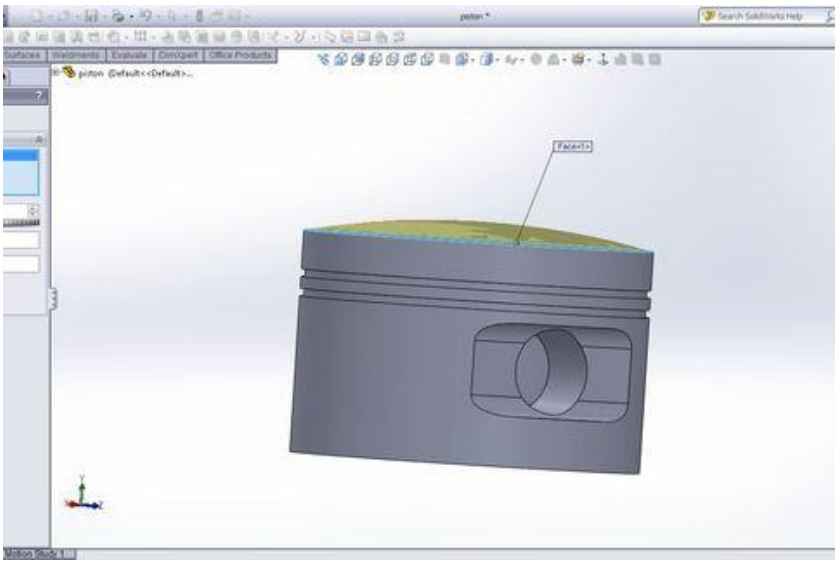
**Step 13-**Cut the body



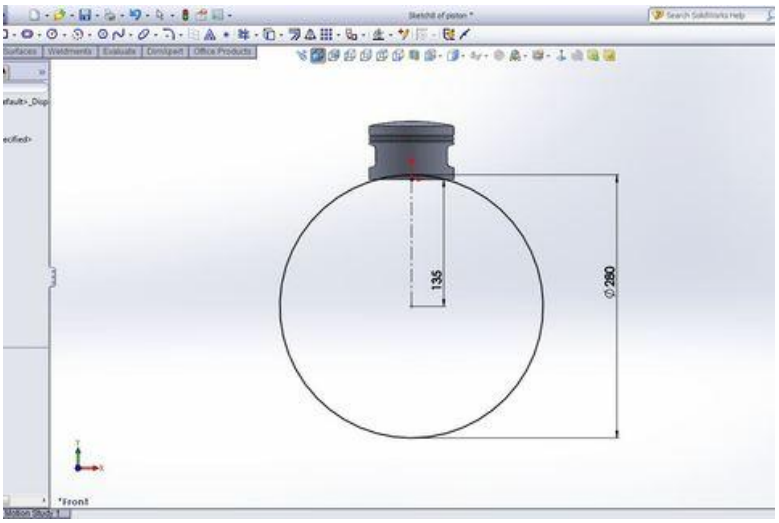
**Step 14-**Now you should use fillet command.



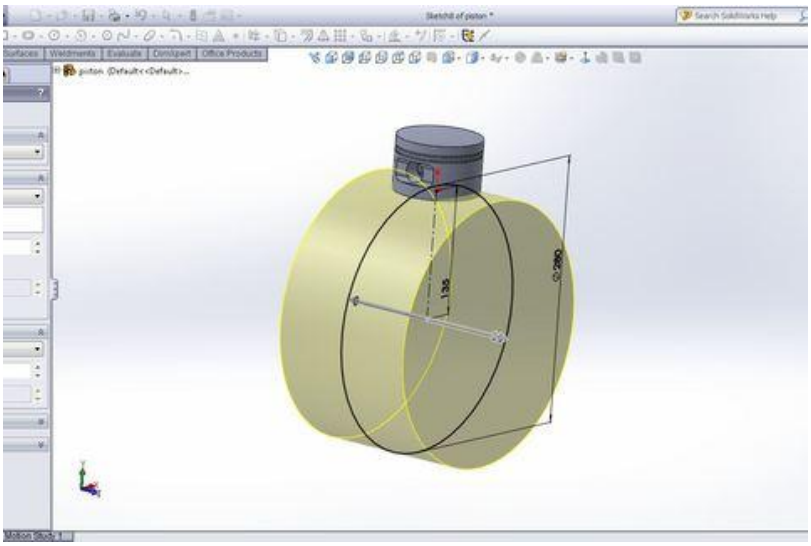
**Step 15-**Use dome command



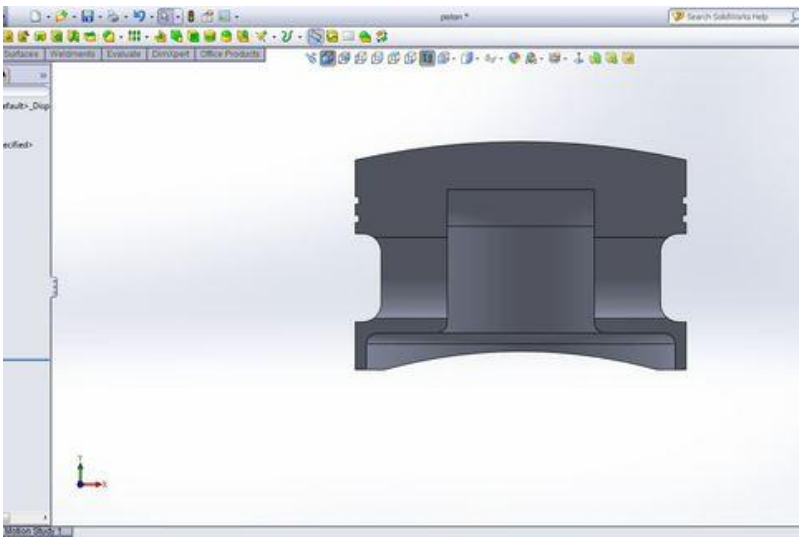
**Step 17-**Draw shown sketch on the front plane



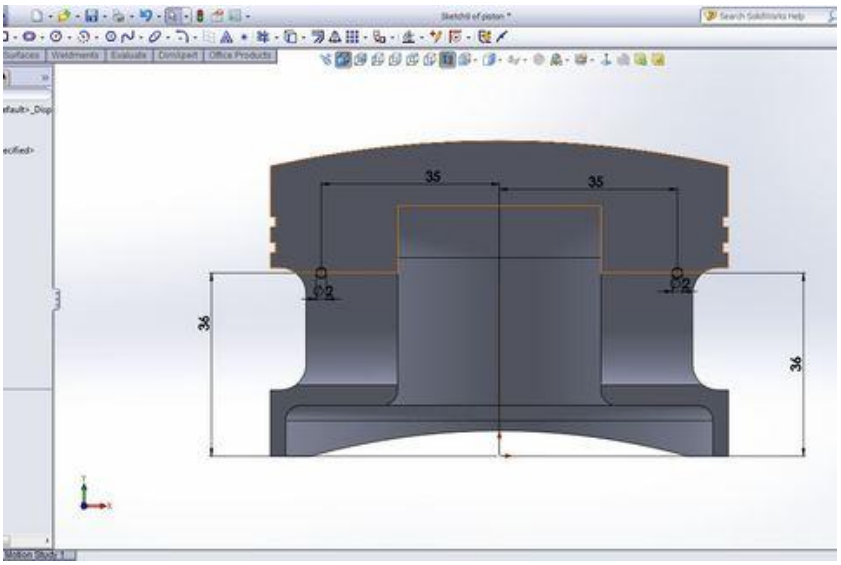

**Step 18-**Use cut extrude command



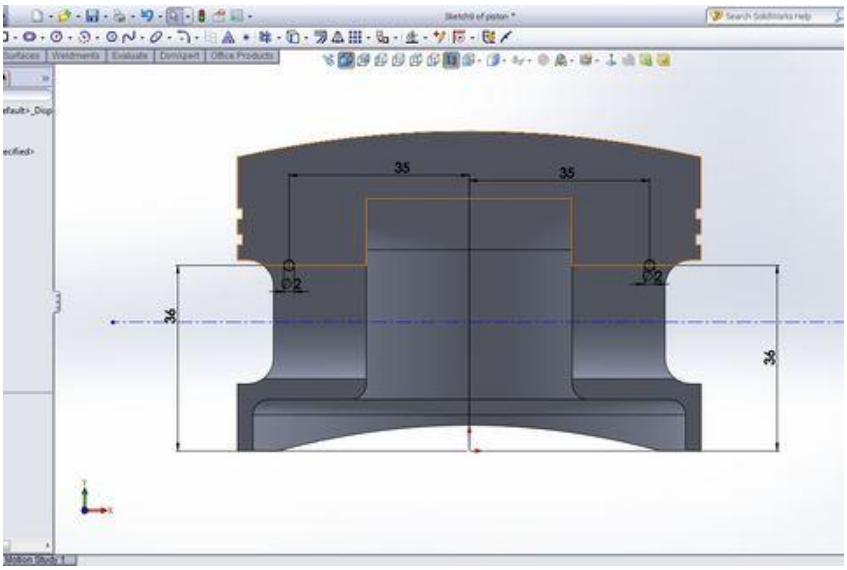
**Step 19-**Use section view button



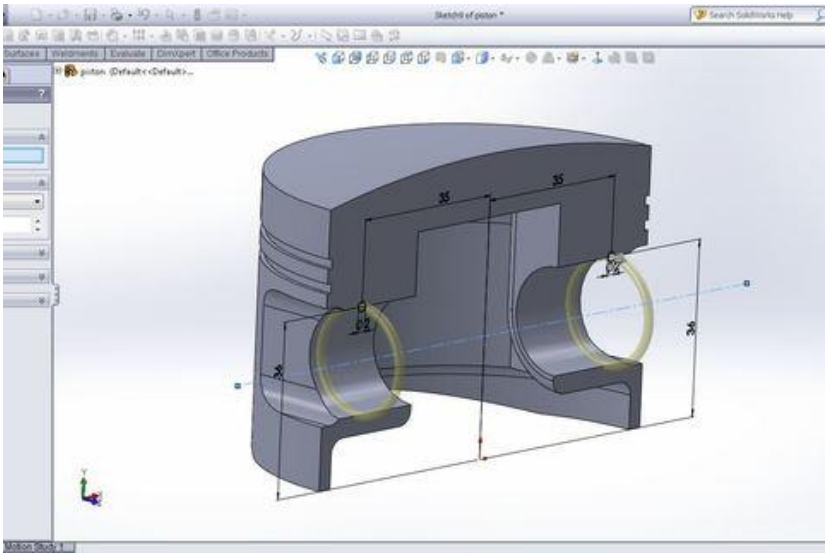
**Step 20-**Open a sketch on the front plane and draw shown sketch




**Step 21-**Add an axis




**Step 22-**Use cut revolve command

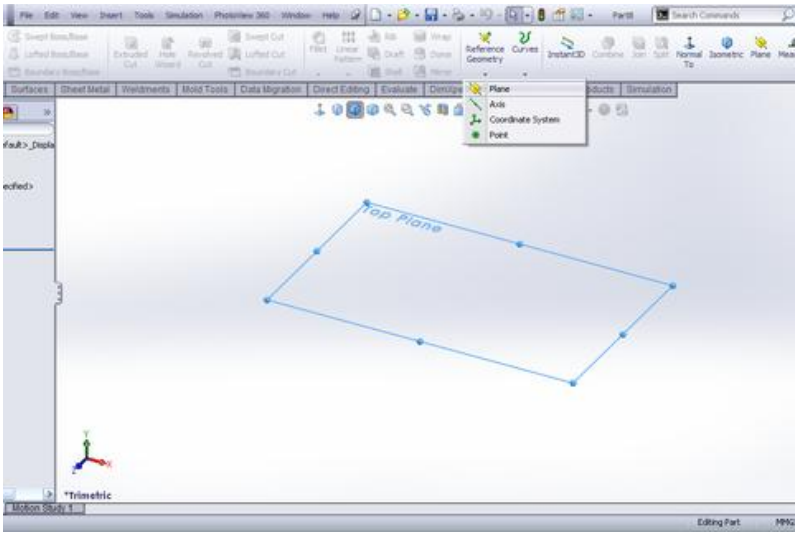



**Step 23-** Disable section view icon & Piston is ready

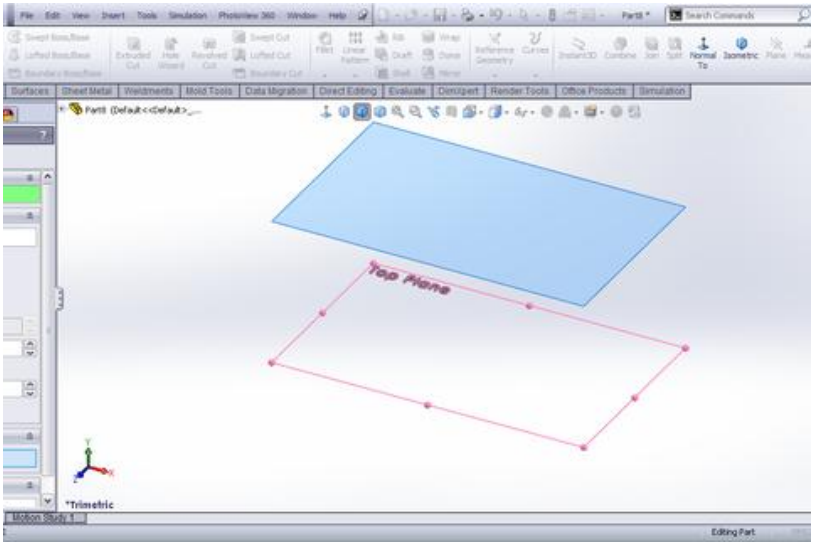
EXPERIMENT: 9

Objective- To make an assembly of Universal Coupling

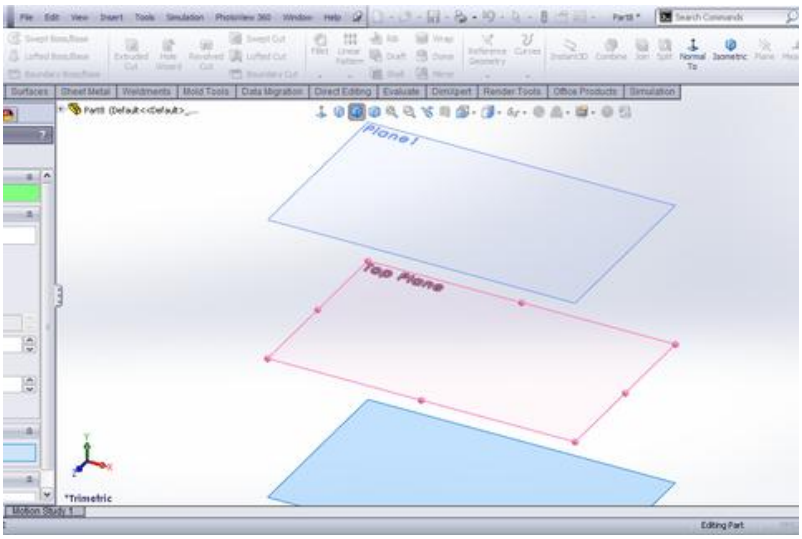
Step 1-Select top plane and then click Reference geometry >> plane.



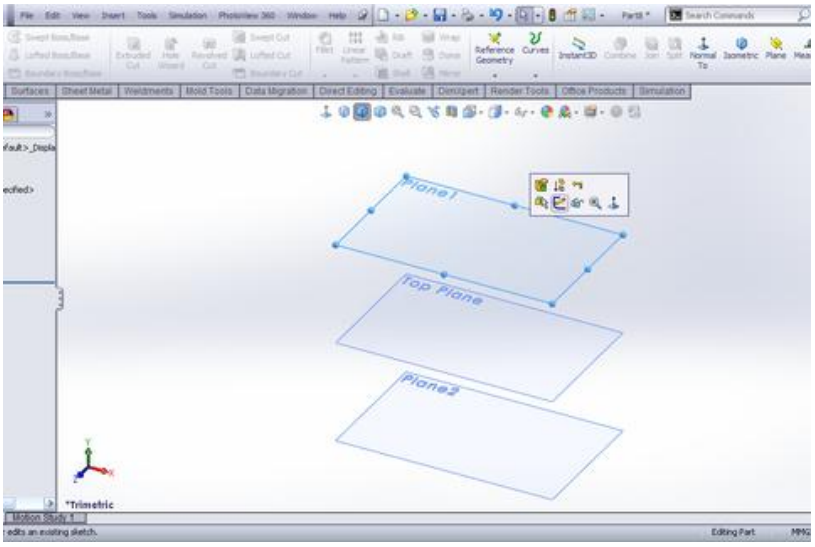

Step 2-Offset the top plane by 83mm.



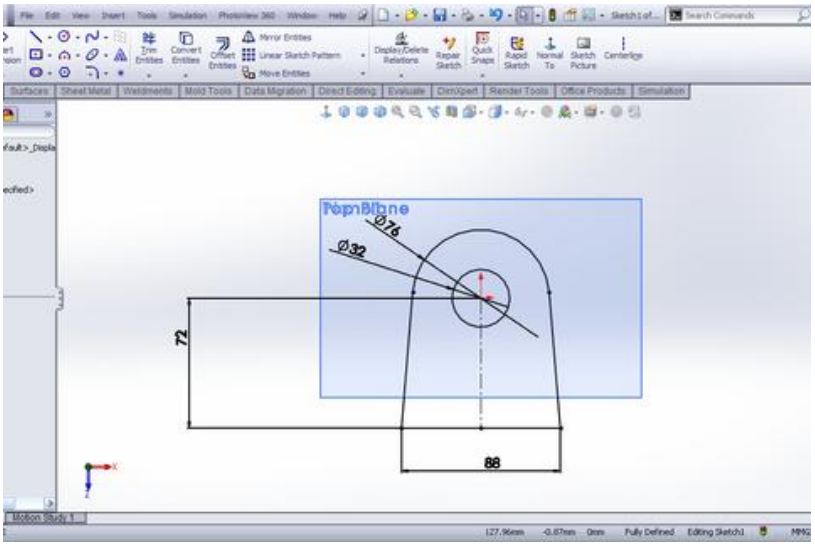

Step 3-Again offset the top plane by 83mm in opposite direction.



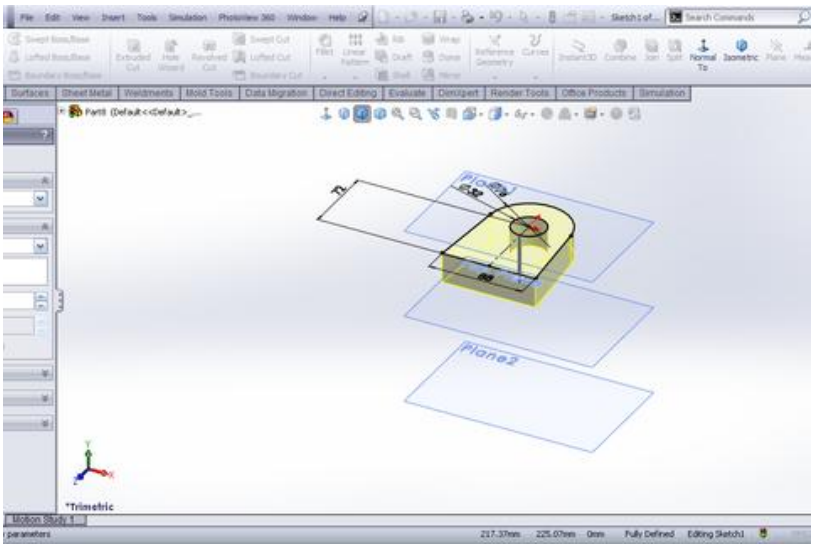

Step 4-Plane1>>Sketch.



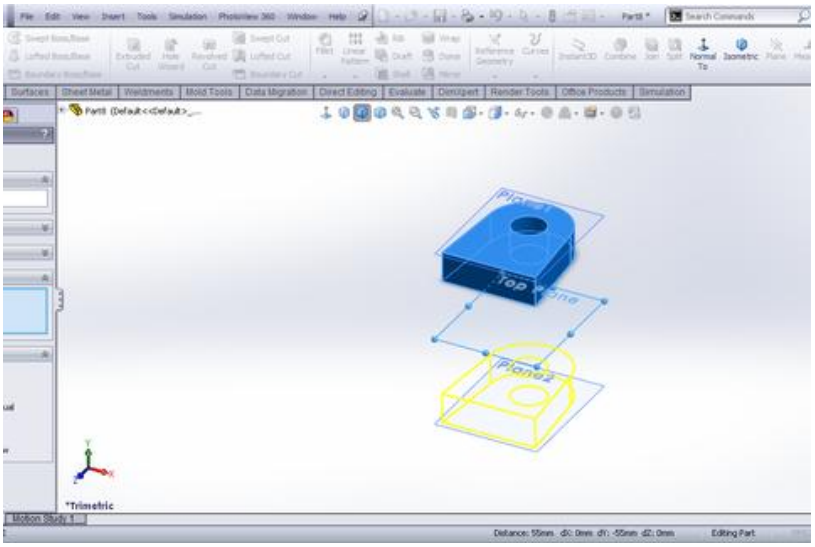

Step 5-Draw this profile.



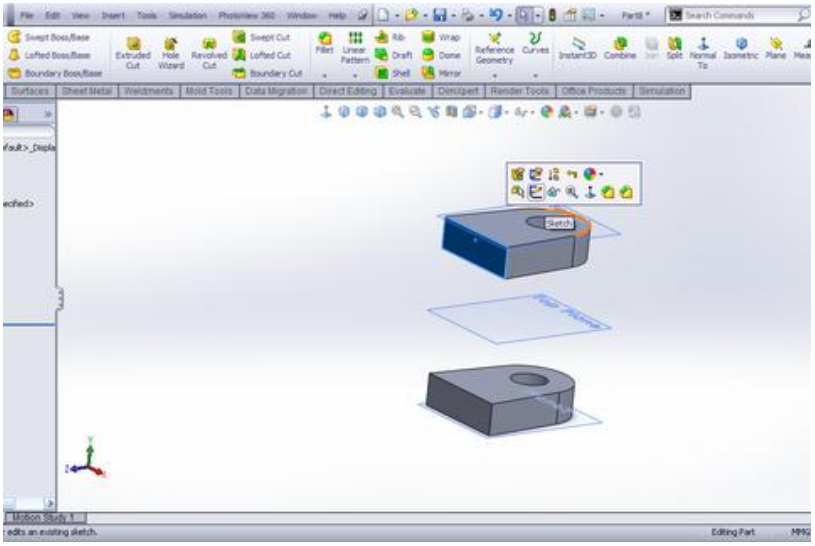

Step 6-Extrude it in downward direction by 28mm.



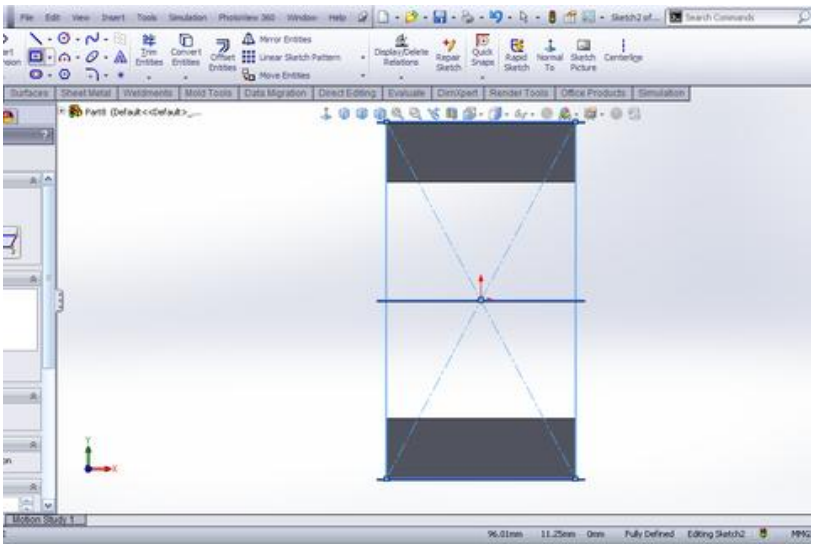

Step 7-Mirror this body about top plane. Uncheck Merge solids.



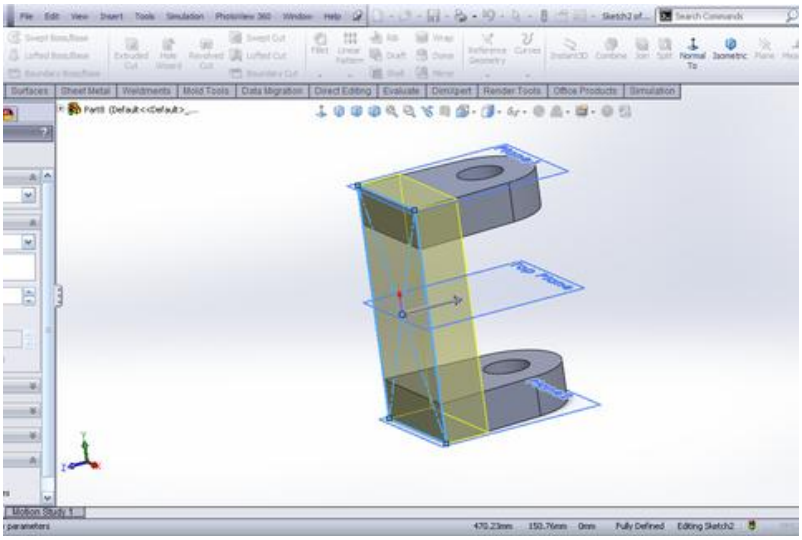

Step 8-Front face >> Sketch.



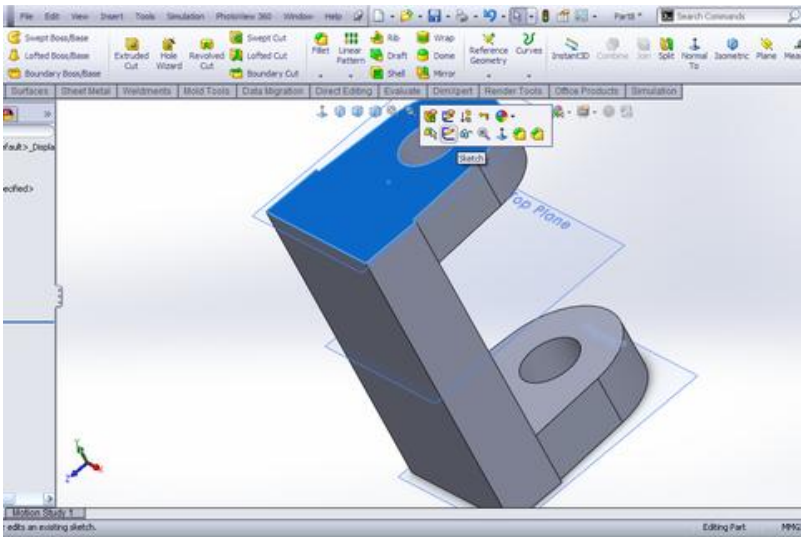

Step 9-Draw a rectangle with center at origin and coincident to the corners



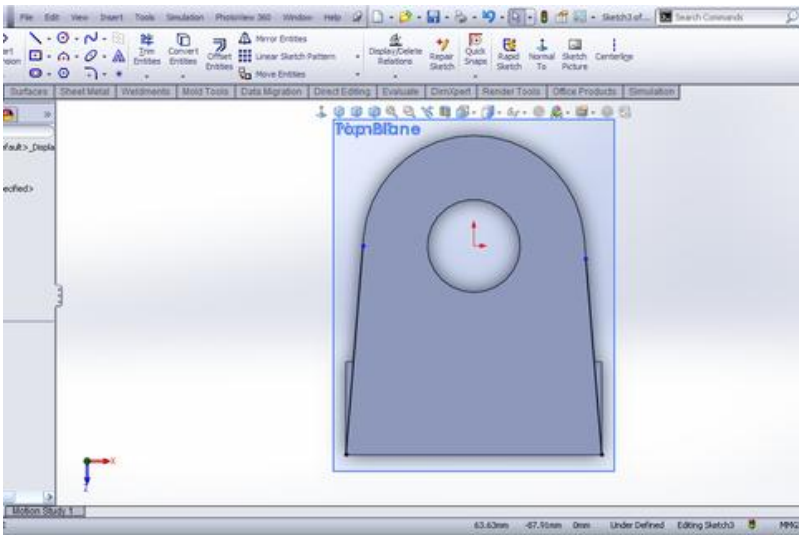
Step 10-Extrude it in backward direction by 32mm



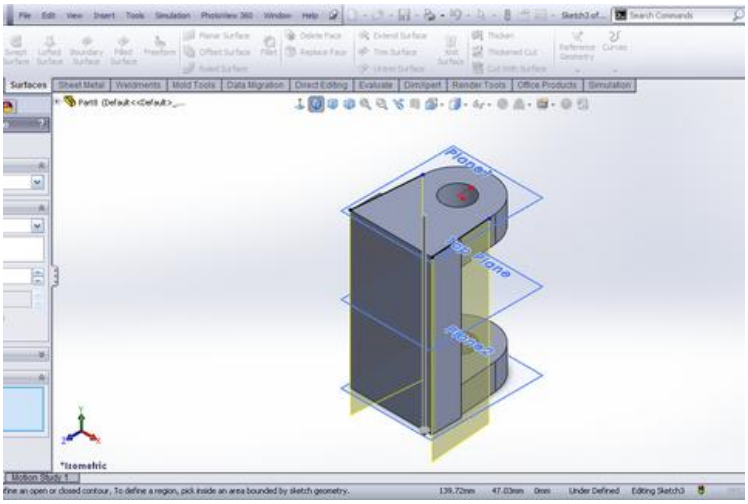
Step 11-Top face>>Sketch.



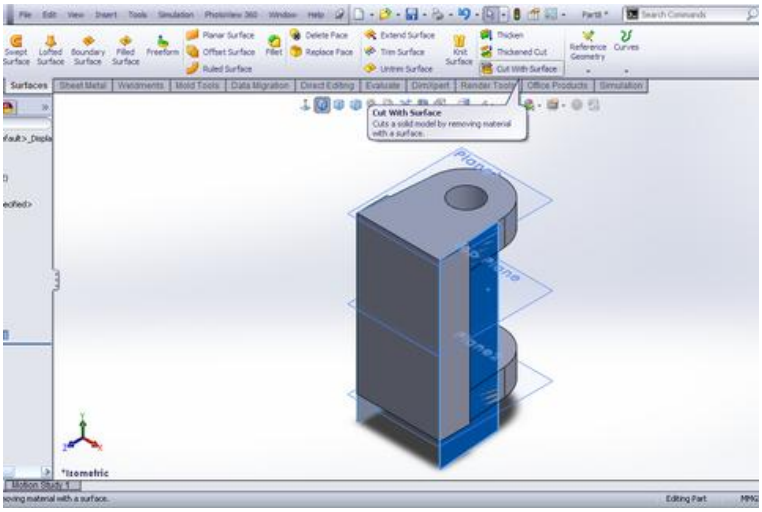
Step 12-Draw two lines.



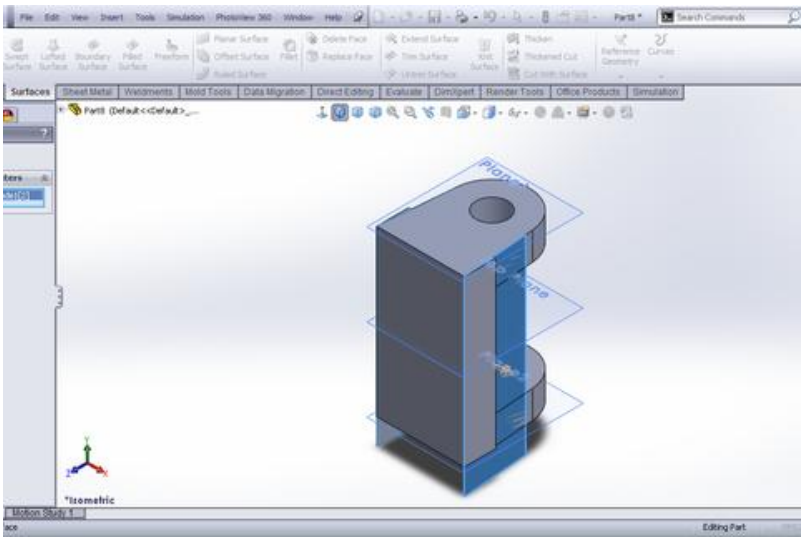

Step 13-Extrude surface by length larger than fork height.



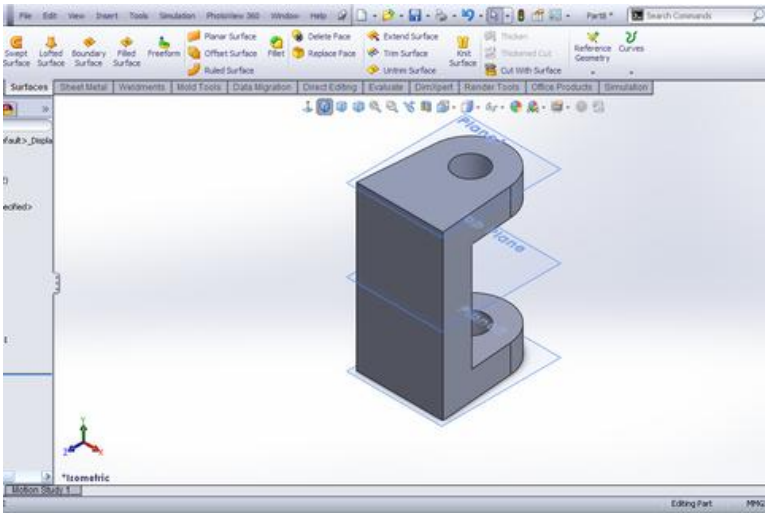

Step 14-Cut with surface.



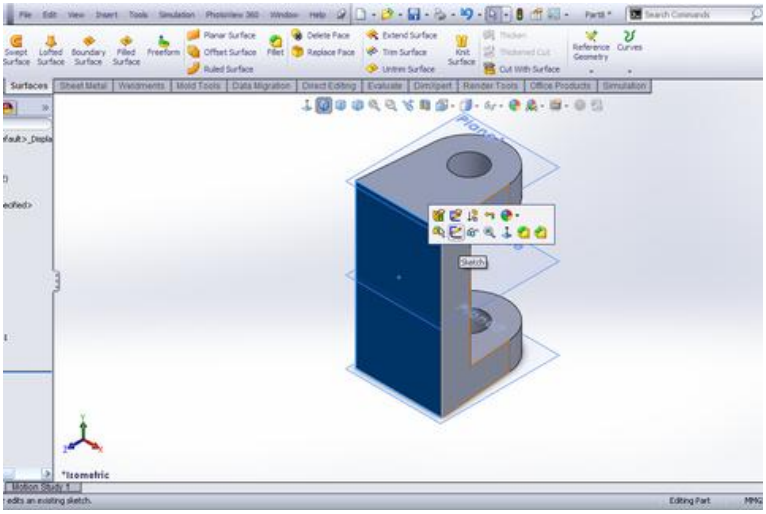

Step 15Select the right surface and direction be outwards the part. Click OK.



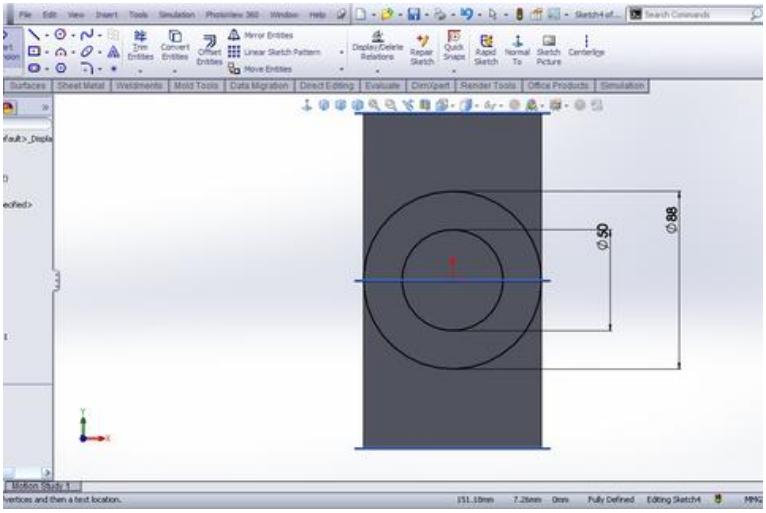
Step 16-Repeat the same step for other side and hide the surfaces.



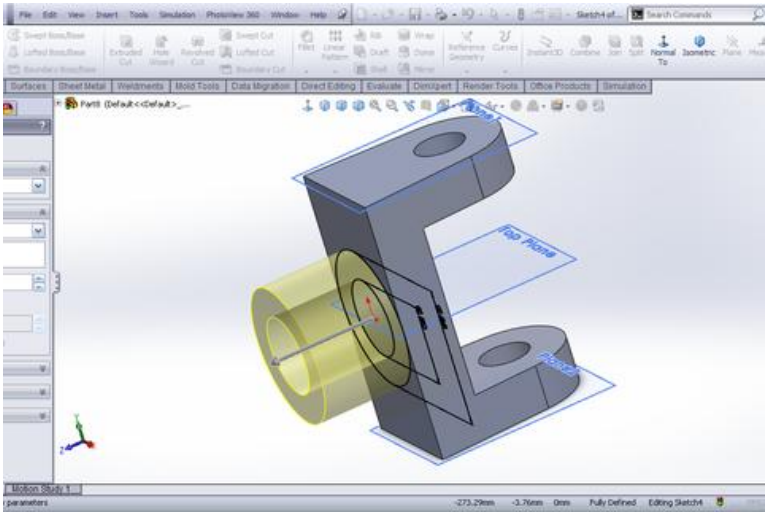
Step 17-Front face>>Sketch.



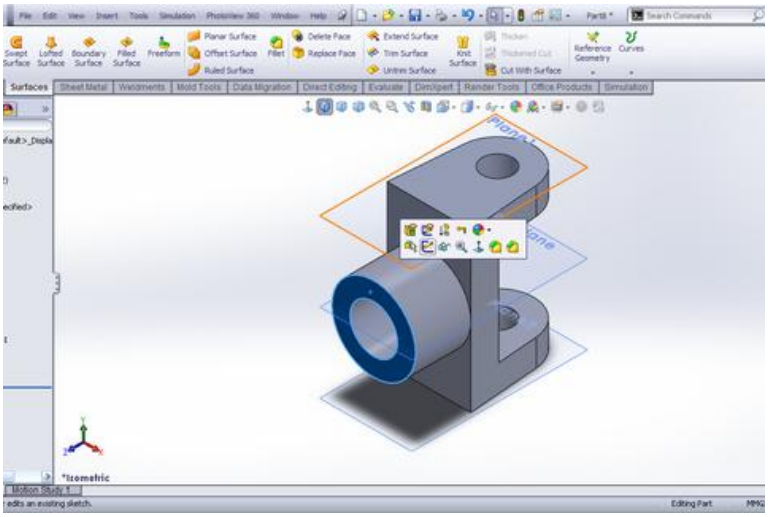
Step 18-Draw two circles of 88mm and 50mm dia.



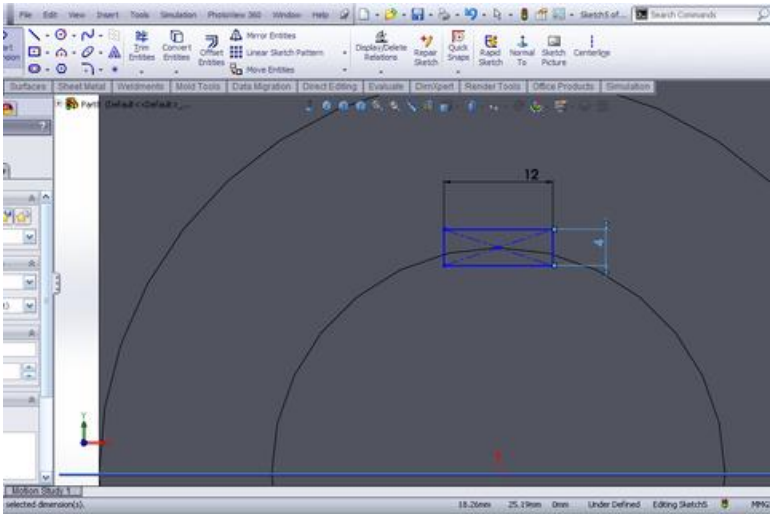
Step 19-Extrude it by 58mm.



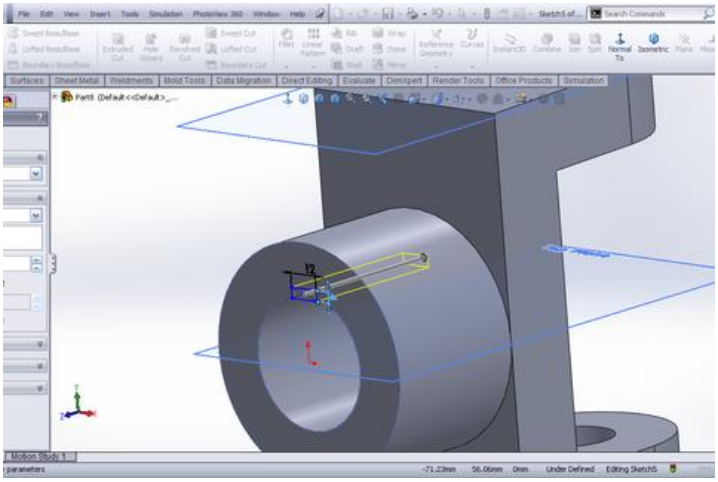
Step 20-Front face>>Sketch.



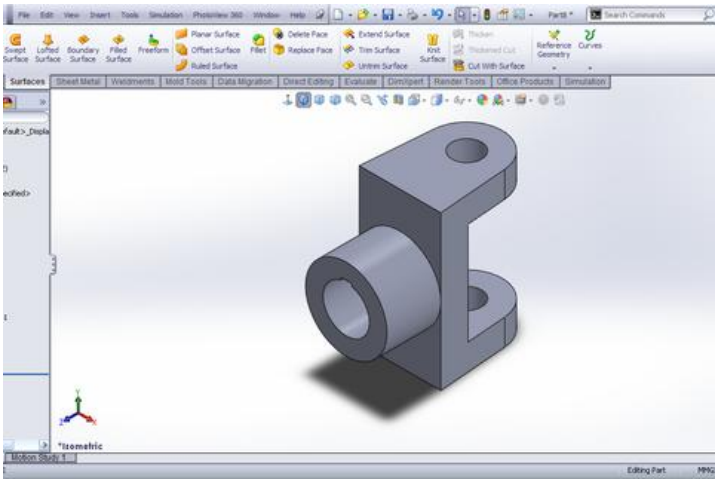
Step 21-Draw this rectangle of 12mm x 4mm.




Step 22-Extrude cut it by 58mm.




Step 23We have the fork.

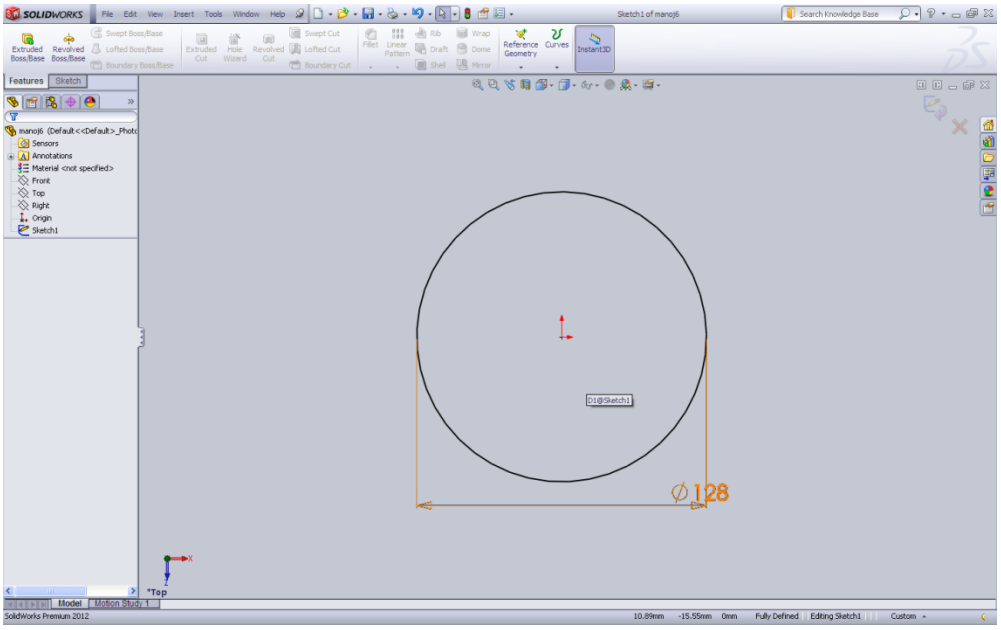


## EXPERIMENT NO :-10

**AIM**: **TO DRAW A 3D MODEL OF A GEAR**

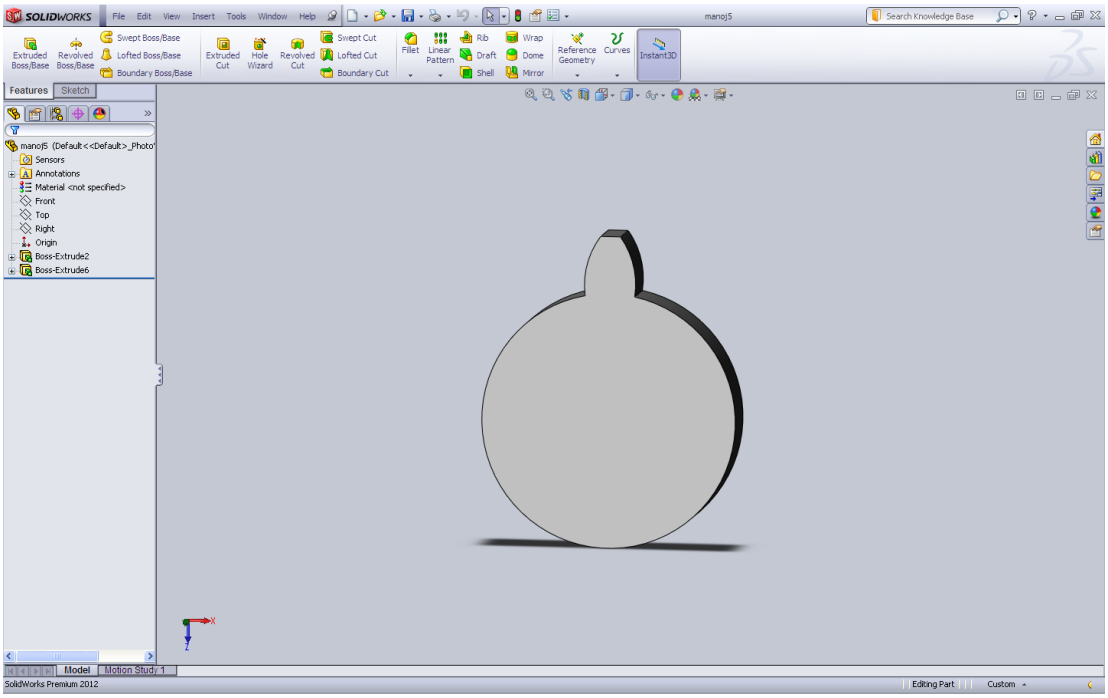
STEP : 1 Open a new sketch on front plane

STEP : 2 Take a circle of diameter 128mm.




STEP : 3 Now go to freature >Extrude Boss > Extrude with 30mm.

STEP : 4 Now open the sketch on front plane and convert entities , and than take a centre line after that a take a 3point arc and mirror about a centre line and join the arc by a line and give a proper dimension.A gear teeth profile generated.



STEP : 5 Now ,Extrude it.

STEP : 6 Now go to freature >Circular Pattern >select the face on which the profile is generated and fill the number of profile 10 and OK.