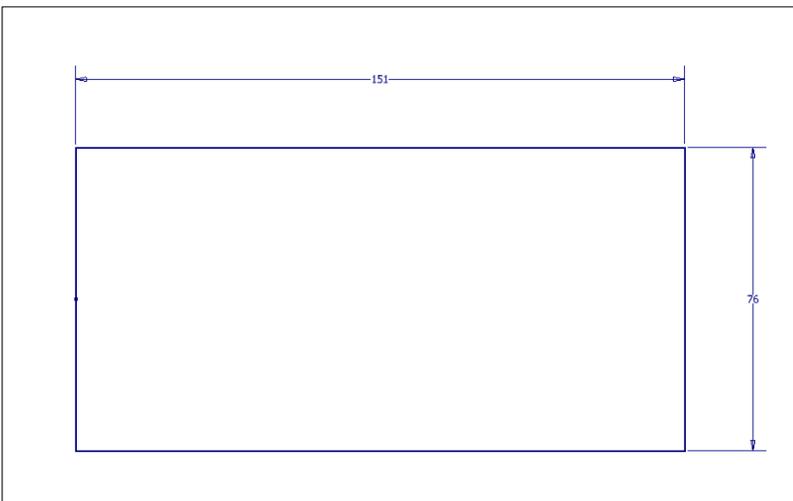
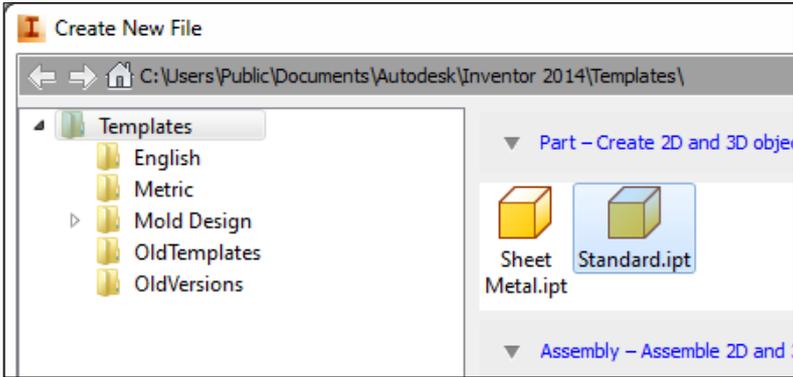
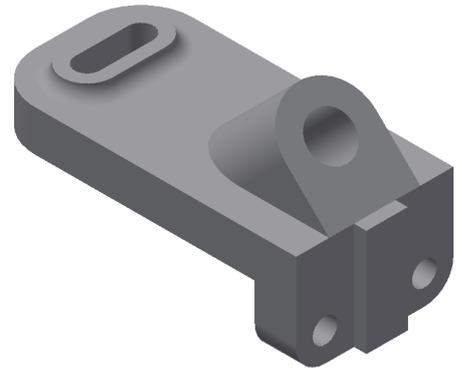


Corner Support Exercise.

Step # 1. From the Create New File dialog box , select the *Standard.ipt* Template.

(Refer to the attached drawing for all dimensions)



Step # 2. On the Sketch tab under the Draw panel select the *Rectangle* command , and sketch a rectangle 151 mm long by 76 mm high.

Finish the Sketch.

FIG 1.0



Step # 3. From the Create Tab on the ribbon, select the *Extrude* Command. Extrude the rectangular sketch 30 mm.

Hit OK

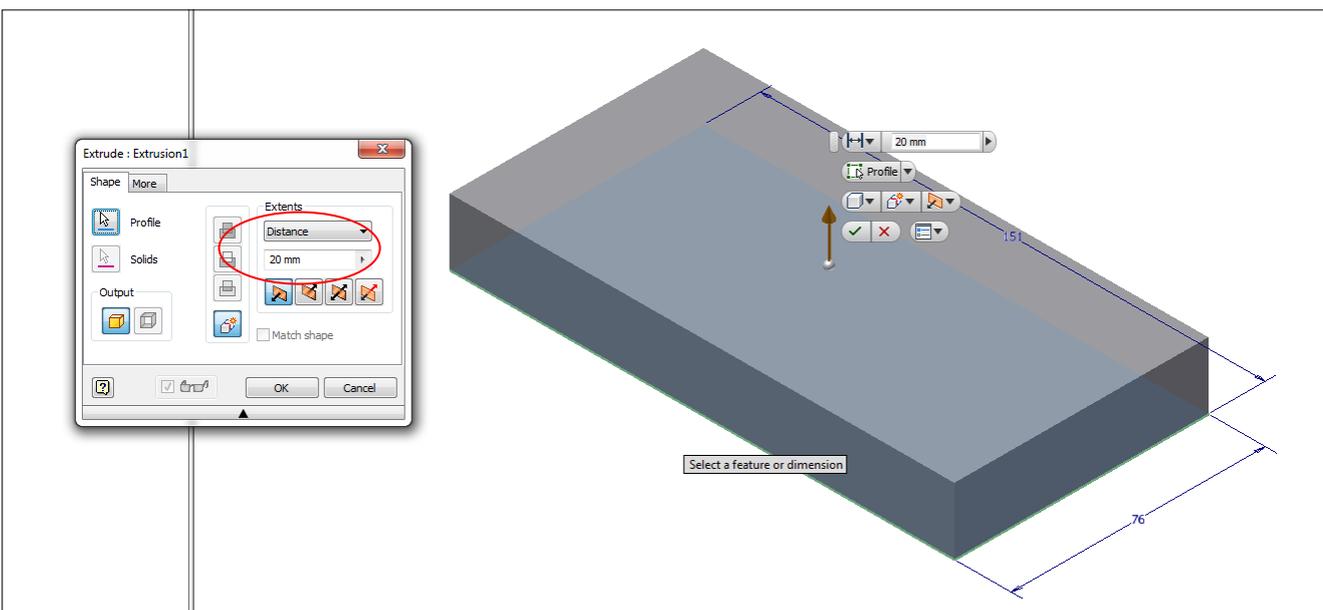


FIG 2.0

Step # 4. Create a second sketch.

Right Click in the graphics area and select *New Sketch*.

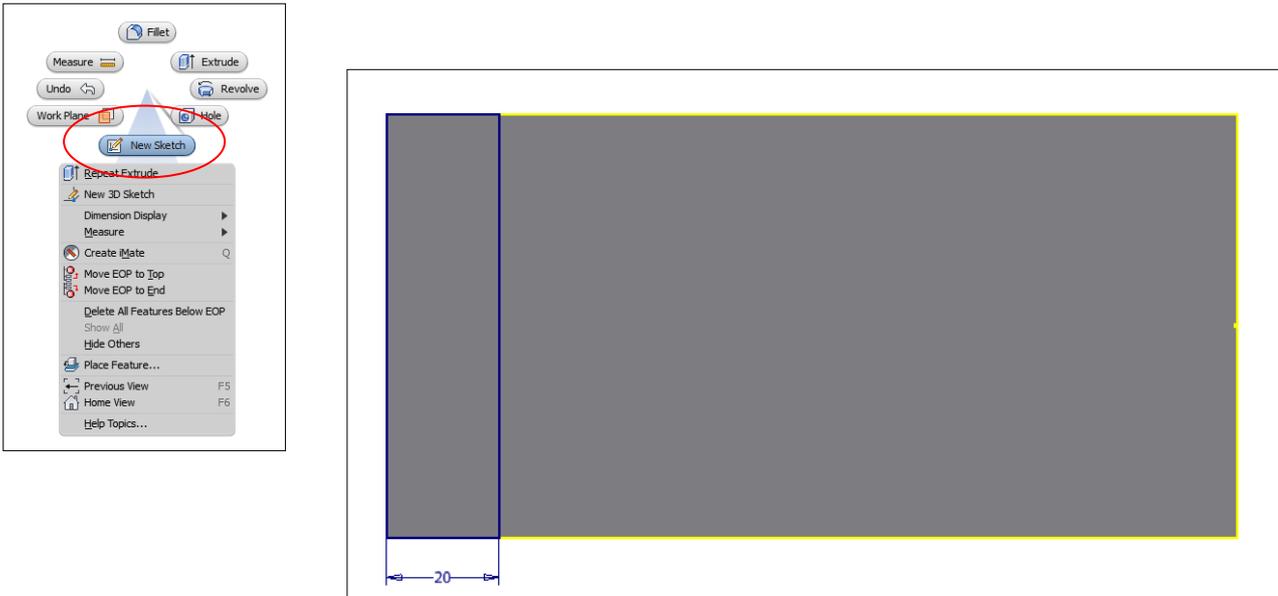


FIG 2.0

Step # 5. As shown in Fig 2.0

Left click on the underside face of the previously extruded rectangular shape. The sketch plane should automatically rotate to face the XY plane.

Using the rectangle command, sketch a rectangle 20 mm wide. Object snaps will appear to position the rectangle at the corner vertices. Finish the sketch

Step # 6. Select the *Extrude Command*.

Select the new 20 mm wide rectangle and extrude it 32 mm. Results are shown in Fig 3.0

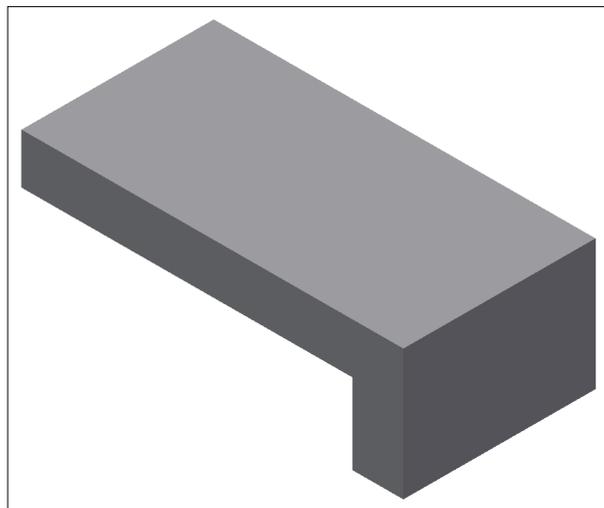


FIG 3.0

Step # 7. From the Modify Tab, select the *Fillet Command*.

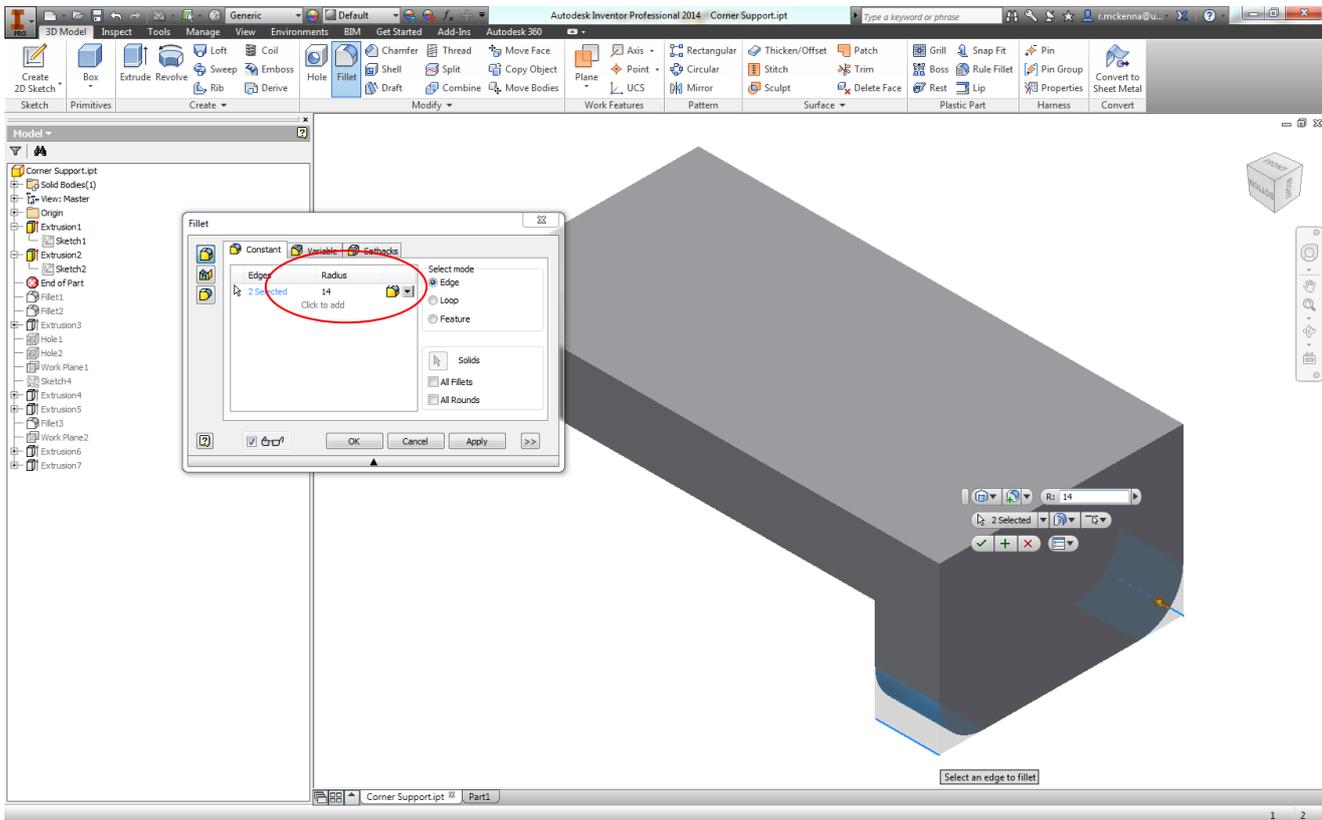


Step # 7. continued.

Select the 2 lower edges indicated and then change the Radius value to 14 mm.

Hit the Apply button

FIG 4.0



Step # 7. continued.

Select the 2 remaining edges and then change the Radius value to 25 mm. Hit OK

FIG 5.0 A

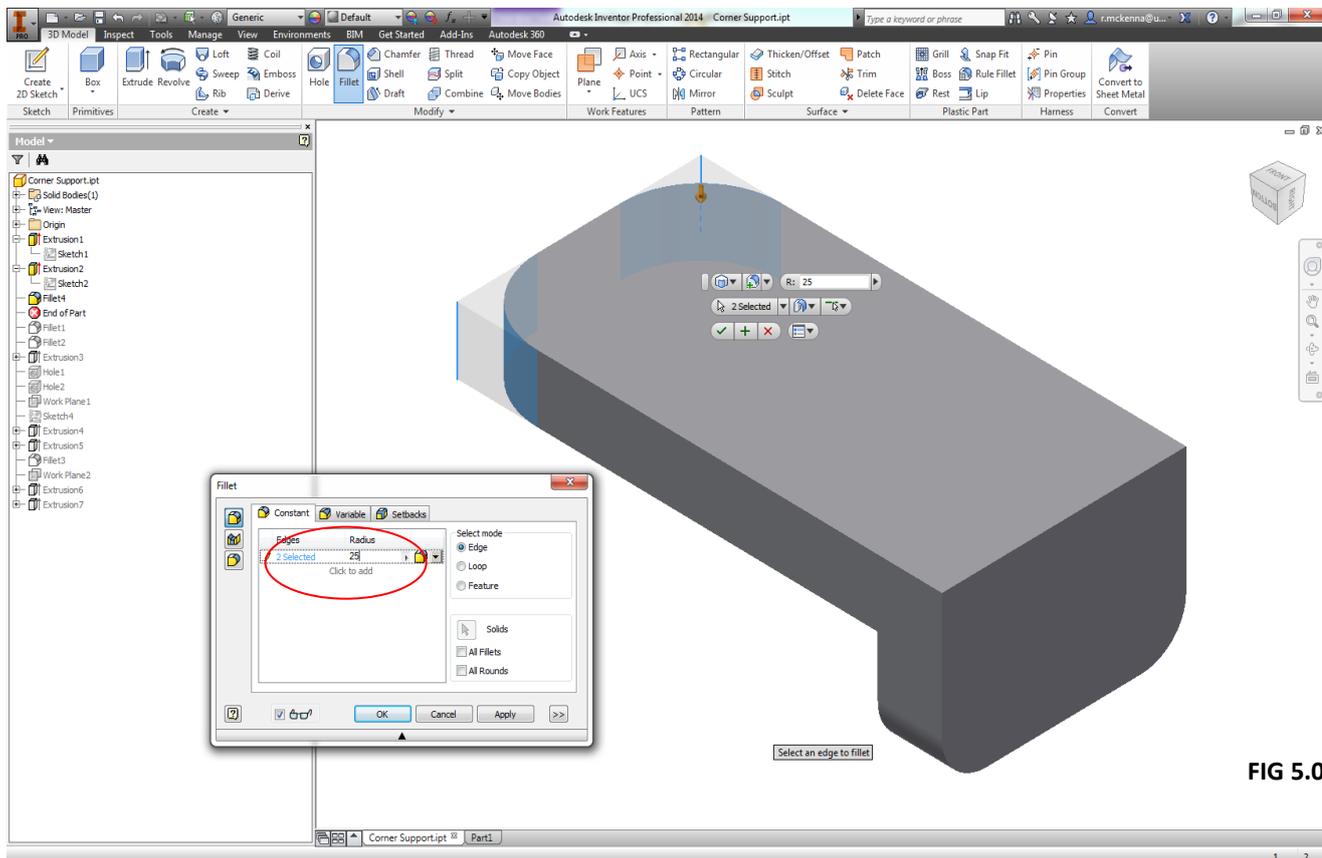


FIG 5.0 B

Step # 8. Right Click on the graphics area and select *New Sketch*.

Use the *Rectangle Command* create another sketch on the model 20 mm wide.

Use object snaps to position the rectangle accurately with respect to projected geometry.

Using the *Horizontal Constraint Tool* from the Constraint Tab, select the Mid-Point Snap of Rectangle and the Mid-Point Snap of the edge of the model. This will centre the sketch geometry

As shown in Fig 6a and 6b

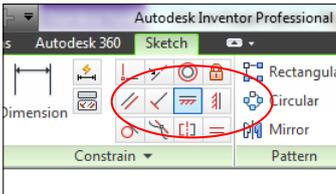
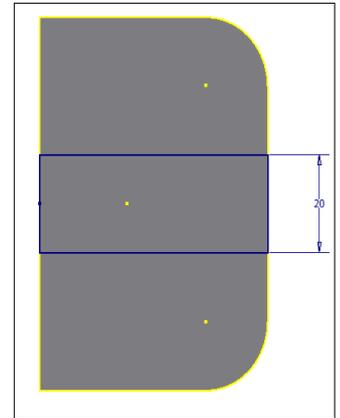


FIG 6.0 A

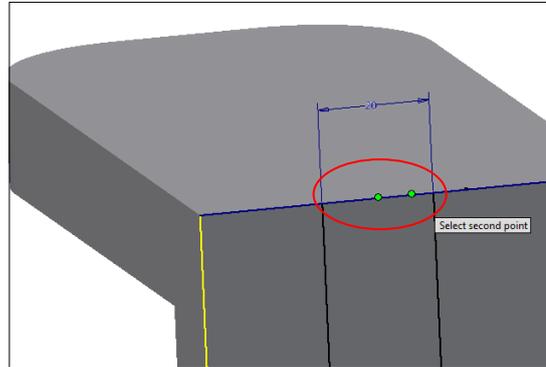


FIG 6.0 B

Step # 9. On the Modify Tab , select the *Hole Command*.

Set the Placement method to “Concentric”, and the hole Diameter to 12 mm.

Select the “Plane Button” and left click on the face as shown in Fig 7.0.

For the “Concentric Reference” , select the edge of fillet feature. A preview of the hole will be displayed. Hit Apply.

Repeat the process for the other 12 mm hole. Hit OK.

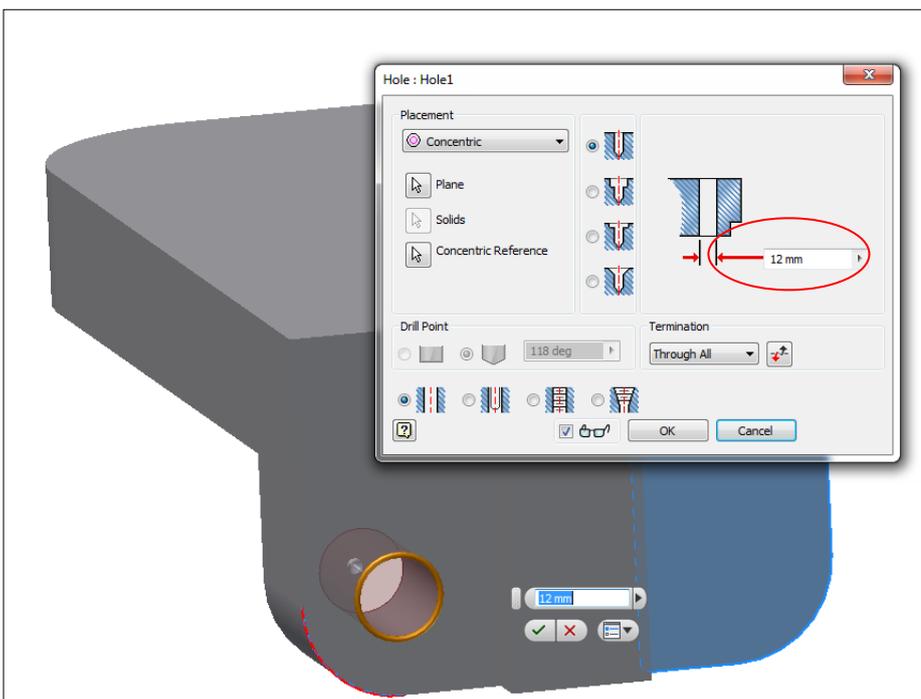


FIG 7.0

Step # 10. Right click in the graphics area and select "New Sketch"

Left click on the top face of the model as shown in Fig 8.0

From the Draw Tab, use the New *Slot Command* to create the sketch.

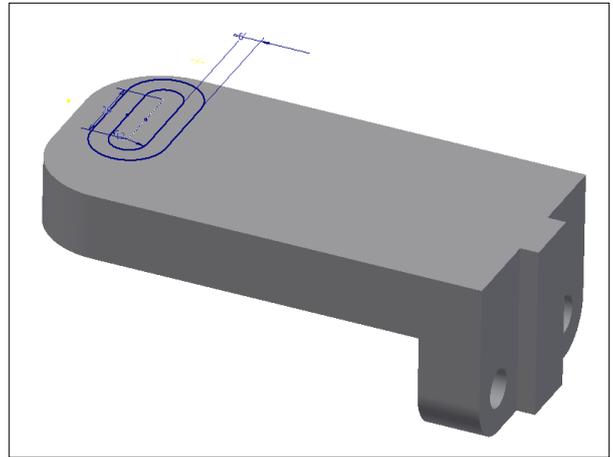
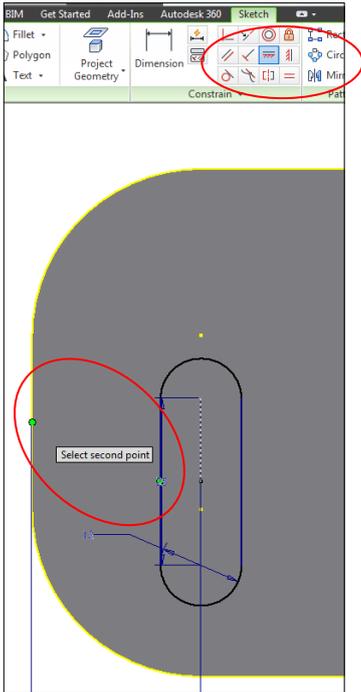
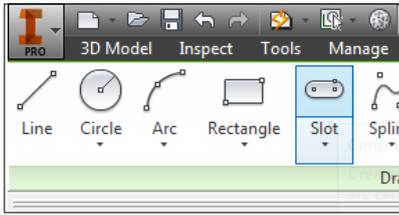


FIG 8.0



To position the slot, place a 25 mm dimension from the edge of the part to the centre of the slot . To ensure the slot is centred on the part, use a horizontal constraint and select the part Mid-Point snap and the Slot Mid-Point snap as shown in Fig 9.0

To complete the sketch, use the *Offset Command* to offset the slot , then dimension the offset 6mm. As shown in Fig 10.0

Finish the sketch

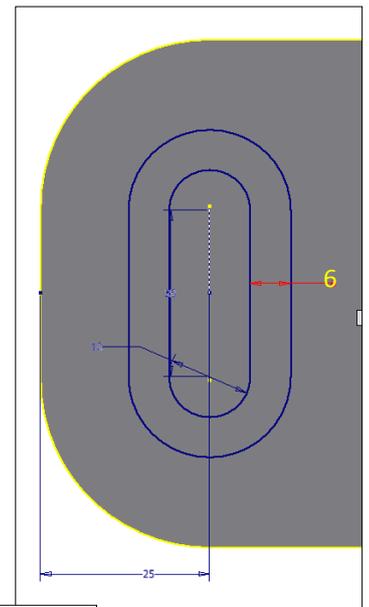


FIG 10.0

Step # 11. From the Create Tab select the *Extrude Command*.

Select the region between the first and second slot geometries and extrude upwards 6 mm, as shown in Fig 11.0

Hit OK

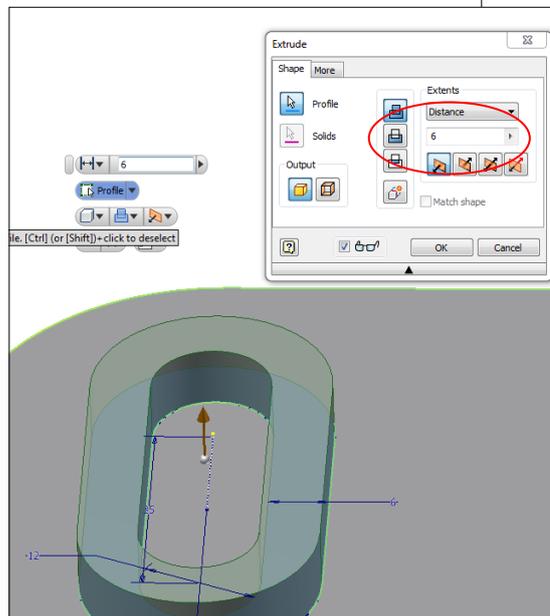
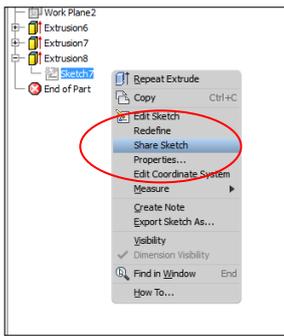


FIG 11.0

Step # 12. Right click in the browser , expand the previous extrusion by pressing the + sign



Right click on the consumed sketch and select Share Sketch.

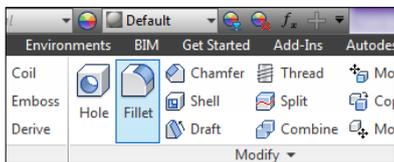
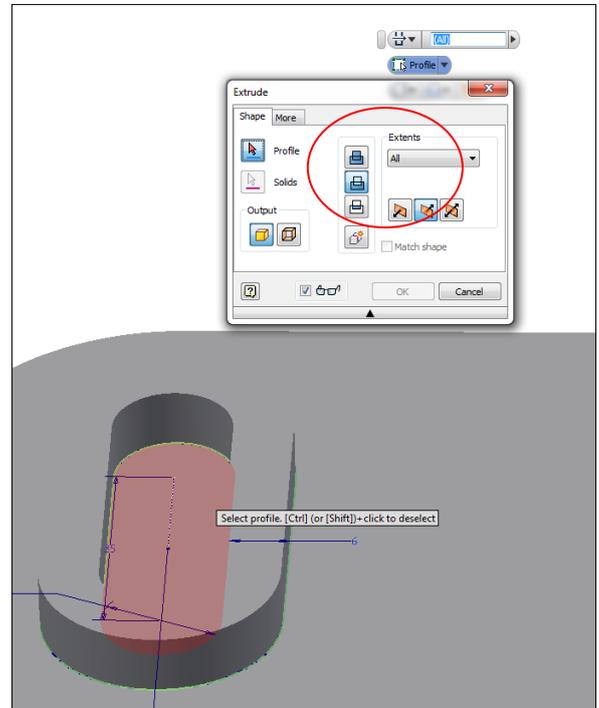
The previously sketched slot geometry is now available to extrude the through slot in the part

FIG 12.0

Step # 13. From the Create Tab select the *Extrude Command*.

Select the centre region of the slot geometry and set the extrude method set as a “Cut”, and Extents set to “All”

As shown in Fig 12.0



Step # 14. From the Modify Tab , Select the *Fillet Command*.

Select the intersecting edge between the slot extrusion and the upper face. Set the Radius to 6 mm. As shown in Fig 13.0

Hit OK

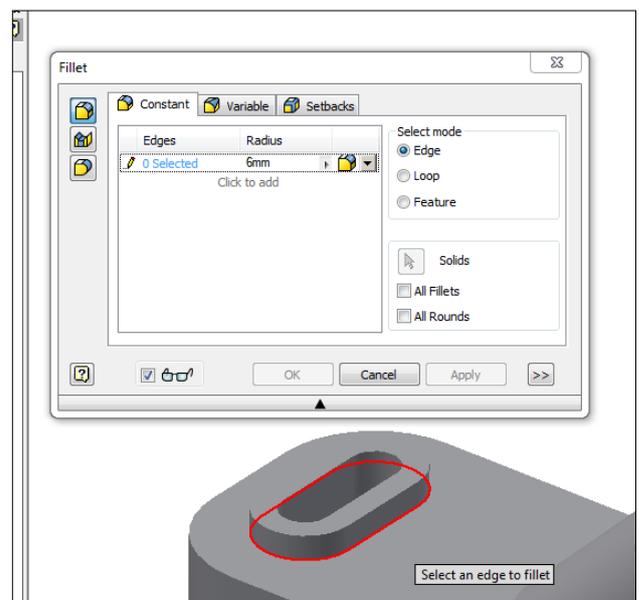


FIG 13.0

Step # 15. Create a work plane

From the Work Features Tab, select the *Plane Command*.

Select the top face, as shown in Fig 14.0 below. Then select the edge as shown in Fig 15.0

After the edge is selected, a prompt will be displayed asking to input an angle value.

Enter 45 degrees. As shown in Fig 16.0

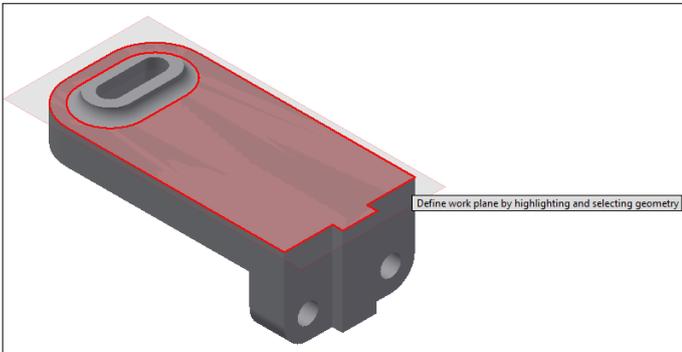
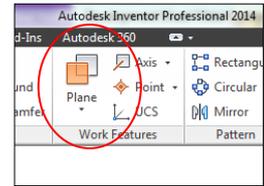


FIG 14.0

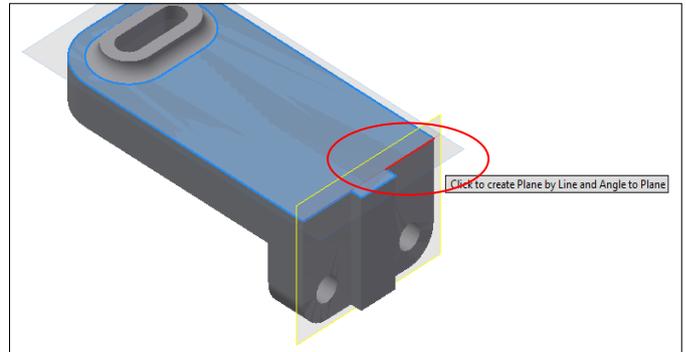


FIG 15.0

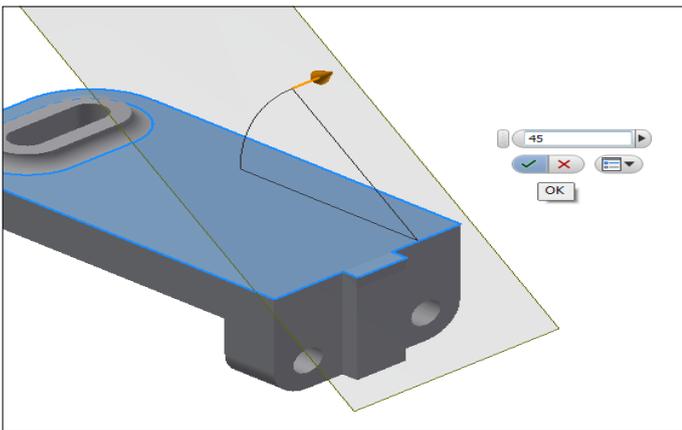


FIG 16.0

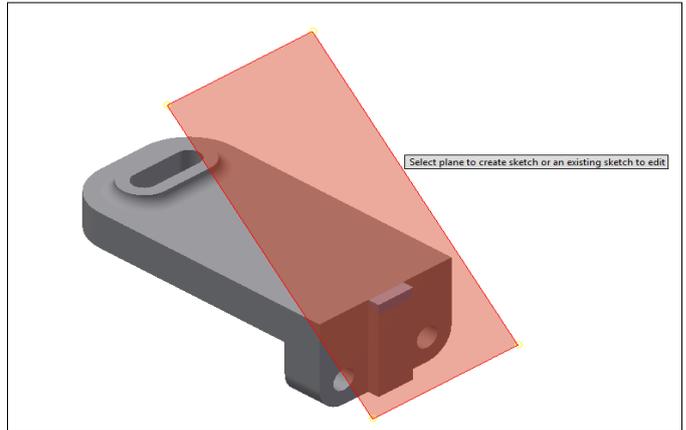


FIG 17.0

Step # 16. Right click in the graphics area and select “New Sketch”

Left click on the border of the of the newly created work plane as shown in Fig 17.0

From the Draw Tab, use the Line Command to create the sketch.

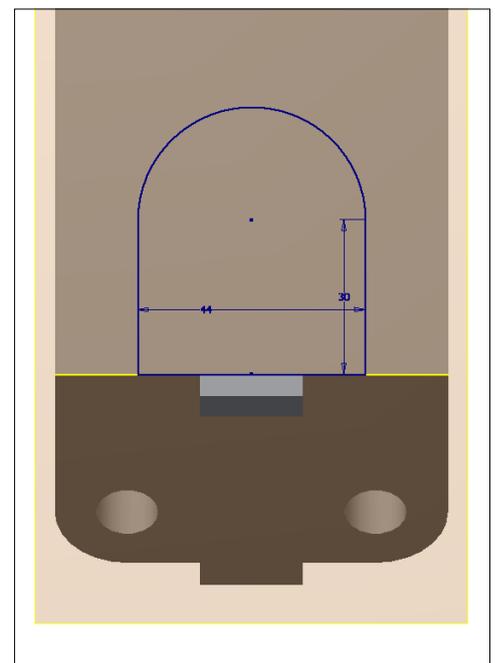


FIG 18.0

Step # 17. From the Create Tab select the *Extrude Command*.

As seen in Fig 19.0 , the region suitable for Extrusion should automatically select.

Set the “Extents” value to “To Next” and OK the dialog box.

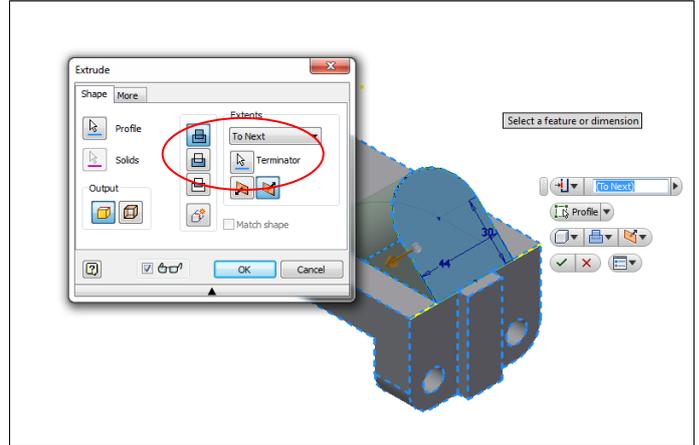


FIG 19.0

Step # 18. Right click in the graphics area and select “New Sketch”

Left click on the top face of the model as shown in Fig 20.0

From the Draw Tab, use the Circle Command create the sketch.

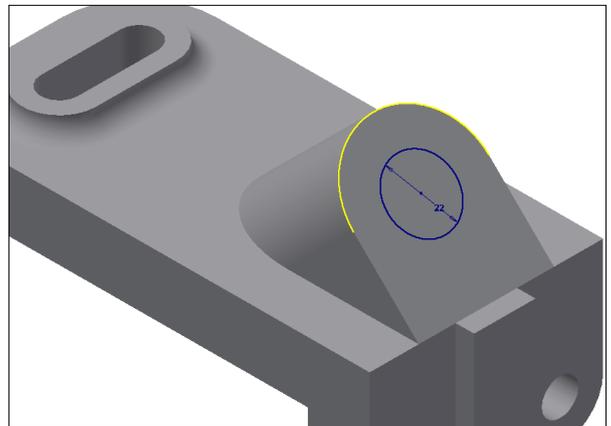


FIG 20.0

Step # 19. From the Create Tab select the *Extrude Command*.

As seen in Fig 21 .0 , select the region defined by the circle.

The Extrusion type should be set to “Cut”

Set the “Extents” value to “Distance”, and the distance value to 32 mm.

OK the dialog box.

The exercise is now complete.

Save your work.

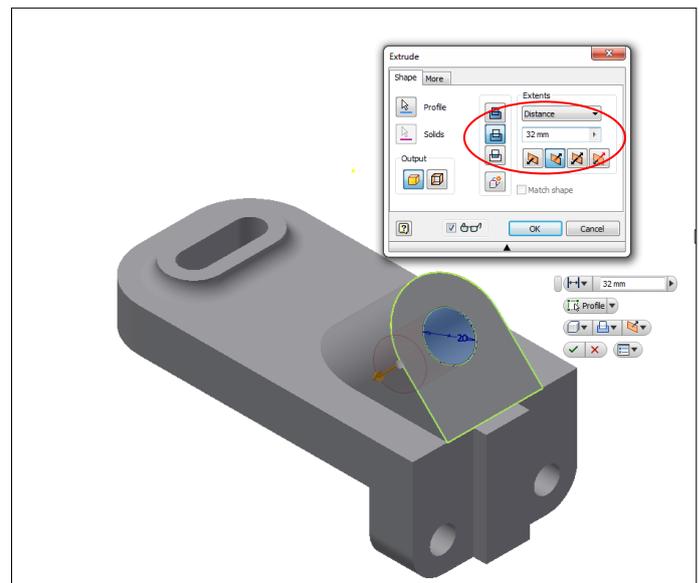


FIG 21.0