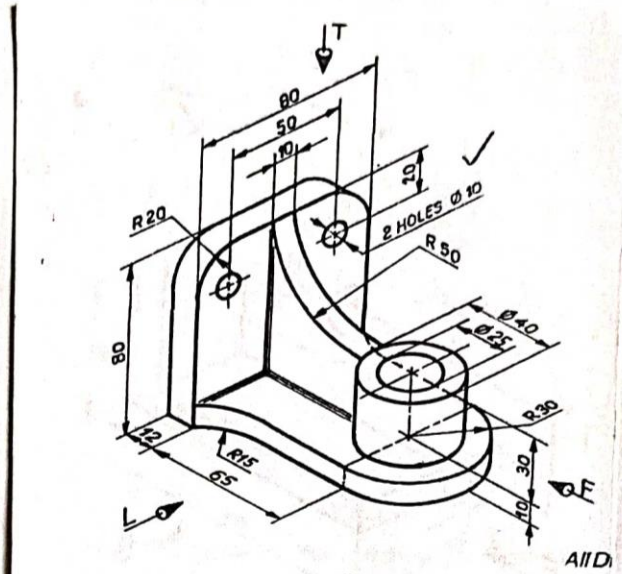


Fig.2.23

Features used are :

* XC - YC datum plane
* YC - ZC datum plane
* Extrude - rectangle, line
  - circle
  - arc
  - quick trim, mirror curve
* Unite
* Subtract
* Edge blend

27/02/2020

Procedure

1. Select the datum plane option and select the YC - ZC plane.

	Let the distance be 0. Select apply.
2.	Click on extrude and the created plane. From the origin draw a rectangle (80 x 80). Draw 2 circles with distance between centres as 50 mm. Click finish sketch.
3.	In the extrude dialog, give distance as 12 and select apply. The boolean should be given 'None'.
4.	Select the datum plane option and click on XC - YC plane and apply. Select this plane for extrusion.
5.	From the edge of the first extrusion draw a 65 x 80 mm rectangle. At the other end, from the midpoint of the rectangle, draw a circle of 60 ϕ. Draw lines from the endpoints of the diameter and extend.
6.	From the ~~two~~ endpoint of the previously drawn rectangle, select arc option and give the radius as 15. Let the arc meet the line drawn from the diameter endpoint tangentially. Join the 2 arc ends.
7.	Draw a centerline from the midpoint of the rectangle and mirror this arc along it.
8.	Quick trim all extra lines to get the profile shown.
9.	Extrude with distance 10 mm. The boolean must be 'unite' with the previous extrusion as the body selected.
10.	Select extrusion and the top face of the extrusion just created.

11. From the centre of the 60 ϕ semicircle, draw a 40 ϕ circle and extrude it to 30 mm., unite with previous

12. Select the top face of this extrusion, draw a 25 ϕ circle from the circle centre and extrude. During extrusion, the boolean must be 'subtract' from the body selected (previously extruded) with distance greater than 40 mm. Select the vector carefully or reverse direction such that the extrusion passes through the body and does not go on top of it.

13. Select extrude and the ~~for~~ top face of the extrusion (base). Draw a centerline from the midpoint of the rectangle to the centre of the circle. Draw the same line with a 5mm distance to the centre line on either side, extend. Draw a 10 mm line end to end.

14. Draw a circle of 40 ϕ to intersect the lines drawn The centre of the circle is same as the centre of the other circles drawn earlier during other extrusions.

15. Using quick trim, trim the centreline completely and the arc, so that we get a profile like 

16. Extrude and unite to 80 mm.

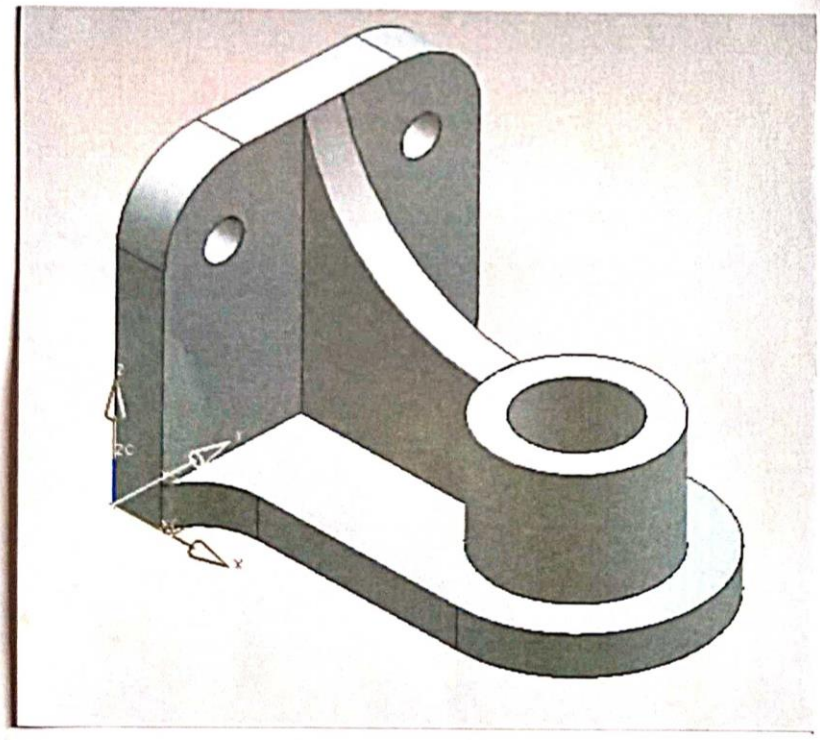
17. Select any one side of this extrusion and click extrude ~~Select~~ arc and click on the topmost point of the rectangular extrusion (seen in side view) and the upper end point of the cylindrical extrusion. give radius 50 mm. Draw 2 lines to form a quarter

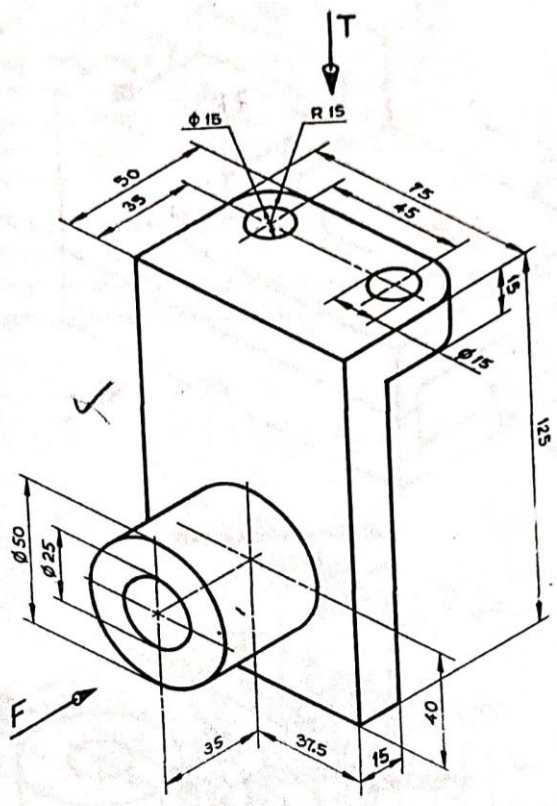
circle which looks like :

18. Select finish sketch and subtract from parent body with distance greater than 10 mm.

19. Select edge blend and the two edges of the first rectangular extrusion and give radius 20 to complete the model's profile.

Output :





All Dimensions i

Fig.2.27

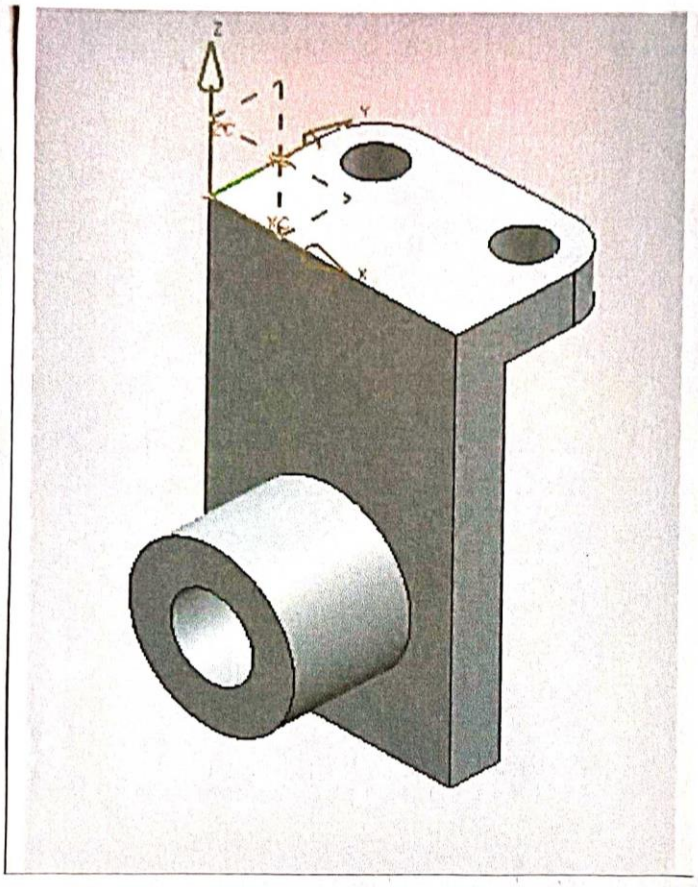
Features used are:

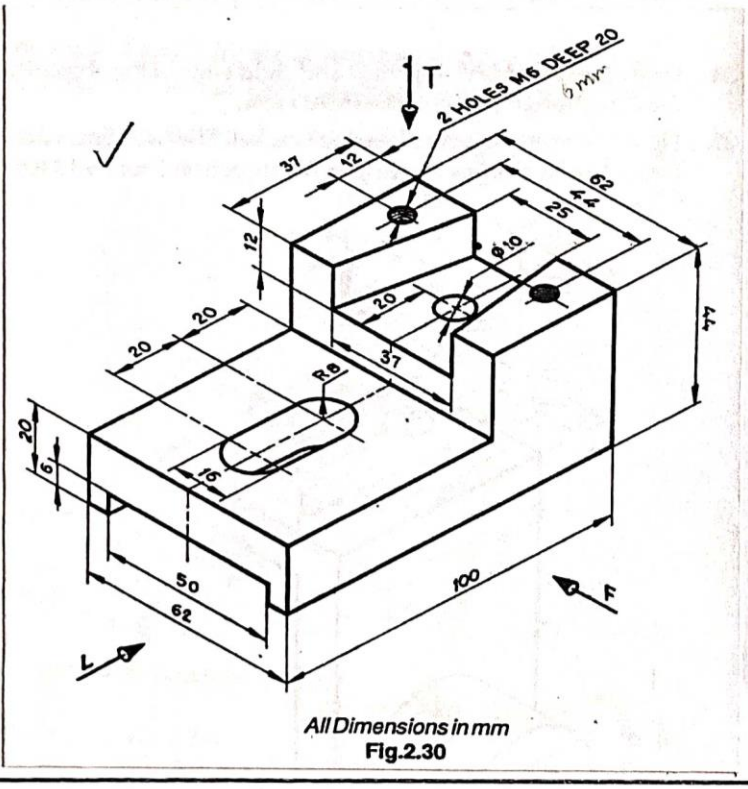
* XC - ZC datum plane
* Extrude - line
  - circle
  - quick trim
* Unite
* Subtract
* Edge blend.

Procedure

1. Select datum plane, XC - ZC plane and apply.
2. Select extrude, click this plane. Draw an L - shaped profile from the origin with dimensions 125 x 15 x 50 x 15 x 35 x 110 mm. (As seen from the right side view of the given figure)
3. Extrude with boolean as 'none', upto distance = 75 mm
4. Click on the top face and extrude. Draw two circles of 15 ϕ with their centres 45 mm apart. Select finish sketch (with dimensioning done) and using 'subtract' boolean, give distance > 15 mm
5. Select extrude and click on the front face of the L shaped extrusion. Draw a circle of 50 ϕ with centre located at 40 mm above and 37.5 mm to the side of the base.
6. Extrude with distance 35 mm and 'unite' boolean.
7. Click the front face of this, extrude a 25 ϕ thole and subtract from parent with distance > 35 + 15 mm
8. Select edge blend and the side edges of the L shape extrusion, with radius 15 mm to complete the model.

Output:





All Dimensions in mm
Fig.2.30

Features used are :

* XC - YC datum plane
* Extrude - rectangle
  - circle
  - line
  - quick trim, mirror curve
  - inferred dimensioning
* Sketch - point
* Unite
* Subtract
* Hole
* Insert design feature - thread

Procedure.

1. Create an XC - YC datum plane.

2. Select extrude and this plane. From the origin, draw a 100 x 62 mm rectangle. Extrude with boolean 'None', and distance 20 mm.

3. Select extrude, left face of this extrusion (from figure) Draw a rectangle 6 x 50mm, 6 ~~on~~ mm from one edge. & Dimension using inferred dimensioning, finish sketch, subtract from parent with distance > 100 mm

4. Select the top face of the parent extrusion, draw a 62 x 37 rectangle from the top edge and extrude with boolean unite with distance 24 mm.

5. Select the top face of this extrusion. Draw a 25 mm line at a distance of 18.5 mm from one edge & at a 35 7 mm at a distance of 12.5 mm on the opposite edge. This is to get that trapezoidal cutout. Subtract from parent with distance 12 mm.

6. Select the top face of this cutout portion. Draw a circle aligned with the midpoint, 20 mm from edge. Draw a rectangle, with its midpoint aligned to the centre of the circle at 20 mm from the same edge. (Rectangle = 20 x 16 mm). Draw circles of 16 ϕ from midpoints of opposite edges of the rectangle. Trim extra curves to get the profile as shown in figure.

7. Subtract from parent with distance > 44 mm.

8. Select sketch option and place a point 12mm from the the top edge & 22 mm from the midpoint of the side edge. Draw a line through the midpoint & mirror this point such that the line acts as ~~a~~ centre line & the distance between the points is 44 mm. Finish sketch.

9. Click on hole, select the 2 points, with general hole of 6mm dia & depth 20 mm. click apply. Select insert → design feature → thread and select detailed thread, selecting one of the holes created. The software automatically adjusts radius, pitch etc, click apply. Repeat with other hole. This completes the model.

Output: