Parala Maharaja Engineering College

A CONSTITUENT COLLEGE OF B.P.U.T GOVT. OF ODISHA

Department of

Automobile Engineering

Lab Manual

Machine Drawing and Solid Modelling

(PC) MEPC 2201

Laboratory Location: Mechanical Engineering Dept. 2nd Floor, Room no- 326

Parala Maharaja Engineering College, Berhampur

A Government Engineering College affiliated to
Biju Patnaik University of Technology, Odisha, Rourkela, India

ପାରଳା ମହାରାଜା ଯାନ୍ତ୍ରିକ ମହାବିଦ୍ୟାଳୟ, ବ୍ରହ୍ମପୁର
(ସରକାରୀ ଯାନ୍ତ୍ରିକ ମହାବିଦ୍ୟାଳୟ)

Safety in the Lab

- You are only allowed in the laboratory when there is a 'responsible person' present such as a demonstrator or the laboratory staff.

- Do not touch any equipment or machines kept in the lab unless you are asked to do so.

- A tidy laboratory is generally safer than an untidy one, so make sure that you do not have a confused tangle of electrical cables. Electrical equipment is legally required to be regularly checked, which means it should be safe and reasonably reliable: do not tamper or attempt to repair any electrical equipment (in particular, do not rewire a mains plug or change a fuse - ask one of the laboratory staff to do it). Never switch off the mains using the master switches mounted on the walls. Please make yourself aware of the fire exits when you first come into the lab. When the alarm sounds please leave whatever you are doing and make your way quickly, calmly and quietly out of the lab. You must always follow instructions from your demonstrators and the laboratory staff.

- You must keep walkways clear at all times and in particular coats and bags must be stowed away safely and must not pose a trip hazard.

- It is important that you make a point of reading the "Risk Assessment" sheet included in the manuscript of each experiment before you start work on the experiment.

- Please take notice of any safety information given in your scripts. If an experiment or project requires you to wear PPE (personal protective equipment) such as gloves and safety glasses, then wear them.

- Always enter the lab wearing your shoes. It is strictly prohibited to enter the lab without shoes.

- There must be NO smoking, eating, drinking, use of mobile phones or using personal headphones in the laboratory. This last point is not because we dislike your choice of music but because you must remain aware of all activity around you and be able to hear people trying to warn you of problems.

- Keep the lab neat and clean.

List of Experiment

1. Sketcher workbench:

a. Creating sketches
b. Selecting & Editing of Geometry, Features, Models
c. Creating Sketcher Geometry & Using Sketcher Tools
d. Using Sketches & Datum Features

2. Basic Solid part modelling

a. Creating Extrudes, Revolves, and Ribs
b. Creating Holes, Shells, Draft & Patterns
c. Creating Rounds, Chamfers & Using Layers

3. Advance Solid Part Modeling

a. Advanced Selection, Creating Sweeps and Blends
b. Sweeps with Variable Sections
c. Helical Sweeps & Swept Blends
d. Relations, Parameters & Family Tables
e. Measuring and Inspecting Models.

4. Assembly design:

a. Creating assembly with top-down approach and bottom- up approach
b. Assembling with Constraints, Exploding, Replacing Components,
c. Cross- Sections in Assemblies

5. Drafting workbench:

a. Introduction, creating new drawings and drawing views,
b. Adding model details and tolerance information to drawings.
c. Adding notes, symbols, tables, balloons and layers in drawings.

Experiment-1

1. Sketcher workbench :

Create a new part.

Click New from the Menu bar. The New SOLIDWORKS Document dialog box is displayed.

Select the Advanced mode.

- If needed, click the Advanced tab. The below New SOLIDWORKS Document box is displayed.
- Click the Templates tab.
- Click Part. Part is the default template from the New SOLIDWORKS Document dialog box.
- Click OK from the New SOLIDWORKS Document dialog box.

The Advanced mode remains selected for all new documents in the current SOLIDWORKS session. When you exit SOLIDWORKS, the advanced mode setting is saved.

The default SOLIDWORKS installation contains three tabs in the New SOLIDWORKS Document dialog box: Templates, MBD, and Tutorial. The Templates tab corresponds to the default SOLIDWORKS templates. The MBD tab corresponds to the templates utilized in the SOLIDWORKS (Model Based Definition). The Tutorial tab corresponds to the templates utilized in the SOLIDWORKS Tutorials.

Part1 is displayed in the Feature Manager and is the name of the document. Part1 is the default part window name.

The Part Origin is displayed in blue in the centre of the Graphics window. The Origin represents the intersection of the three default reference planes: Front Plane, Top Plane and Right Plane. The positive X-axis is horizontal and points to the right of the Origin in the Front view. The positive Y-axis is vertical and points upward in the Front view.

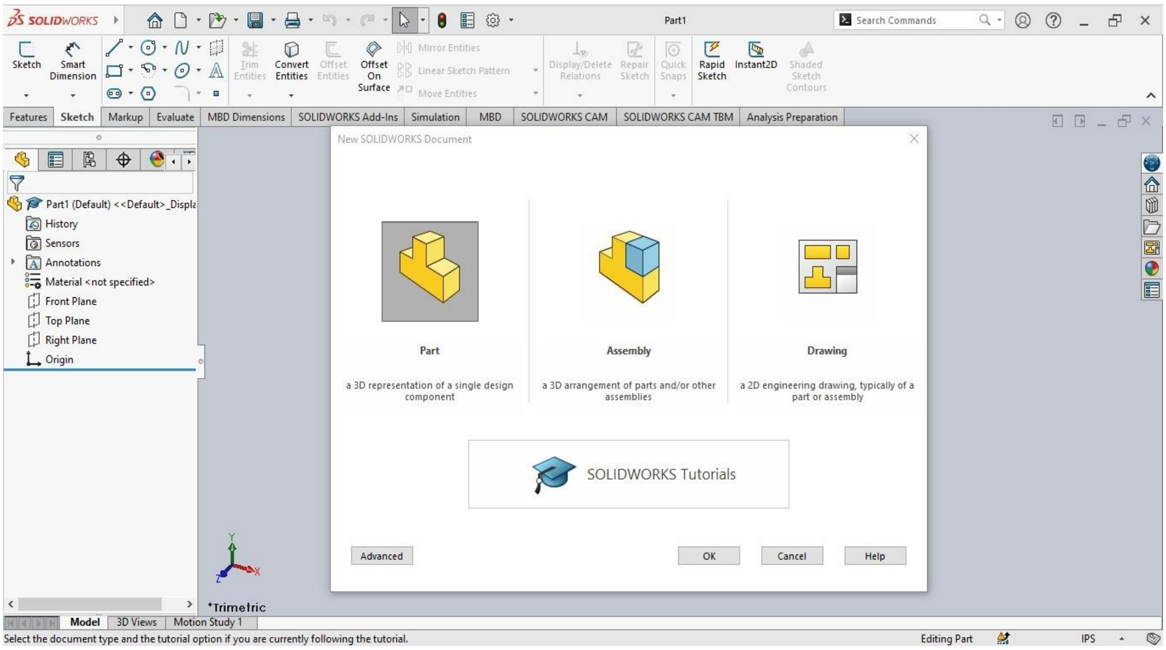
The Feature Manager contains a list of features, reference geometry, and settings utilized in the part.

Edit the document units directly from the Graphics window as illustrated.

Command Manager tabs, Feature Manager tab, Task Pane tabs will vary depending on system setup, version, and Add-ins.

View the Default Sketch Planes.

- Click the Front Plane from the Feature Manager.
- Click the Top Plane from the Feature Manager.
- Click the Right Plane from the Feature Manager.
- Click the Origin from the Feature Manager. The Origin is the intersection of the Front, Top, and Right Planes. The Origin point is displayed.
- Click inside the Graphics window.



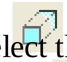
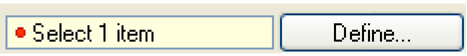
Experiment-2

Basic Solid part modelling Creating Extrudes, Revolves, and Ribs Creating Holes, Shells, Draft & Patterns Creating Rounds, Chamfers & Using Layers

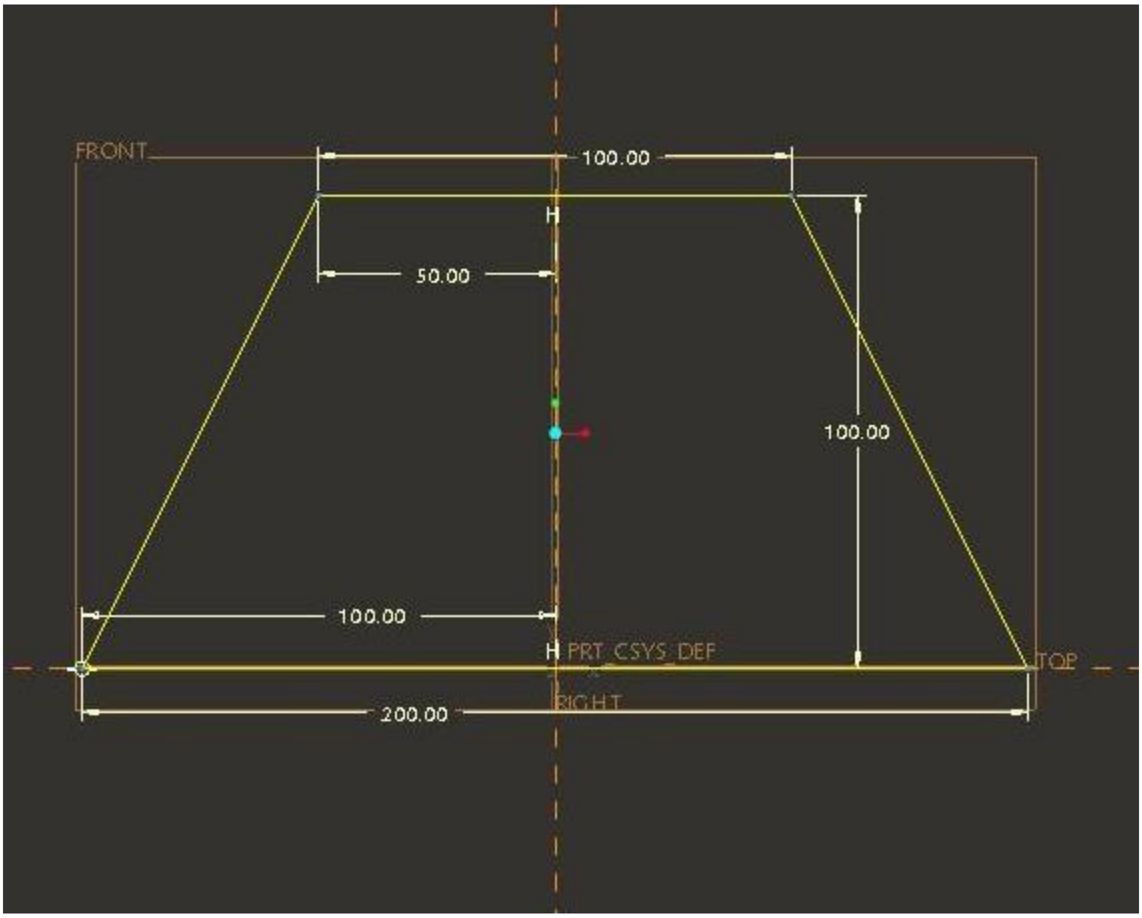
Introduction:

In this tutorial you will build up on what you have learned in the previous tutorials by creating a bit more complex part using extrude, revolve and the hole tool. In addition to the hole tool which was introduced previously, Pro/E has several other tools that can be readily used for modifying existing solids. Some of these tools, such as shell, rip, round and chamfer, will be introduced. Datum planes are used for drawing sketches in order to create solid features. When there is no surface available for drawing a sketch at the desired location, additional datum planes can be created at that location. In this tutorial you will see how additional datum planes can be created.

Creating Extrusion:

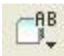
1. Start Pro/E Wildfire.
2. Choose [**File**] -> [**Set Working Directory…**], and select a folder to save your work in.
3. Select [**File**] -> [**New**], and type the part name [**Tutorial4**] in Text Box.
4. Click the [**OK**] Button.
5. Select the **Extrude**  icon from the *Feature Creation Toolbar* at the right of the screen.
6. Select the **Placement**  button then the **Define** 

button from the Dashboard.
7. Select the plane marked as **FRONT** to be your sketching plane then click the [**Sketch**] button.
8. [**Close**] the References dialog box.
9. Select the **Line**  icon from the Sketcher Toolbar and sketch the shape shown in Figure 4.1 and dimension it as shown in the figure. Note that you will need to use the Create dimension  icon to define the same set of dimensions shown in the figure. Remember that you can add the dimension between any two entities by clicking both entities then clicking the middle mouse button.

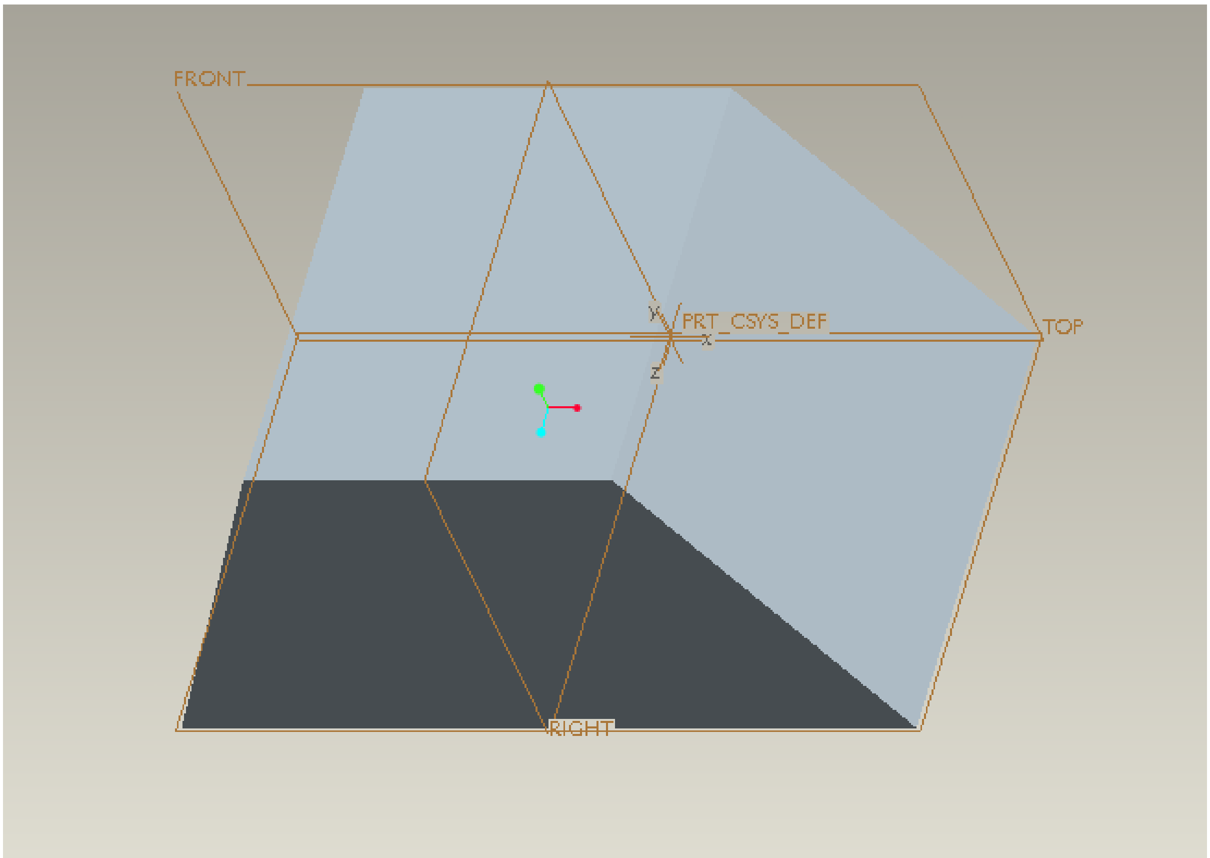


10. Click on the **check**  button to exit the Sketcher.

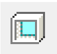
11. From the Dashboard set the extrusion depth to be **125**, and click the **Check**  button to complete the extrusion.

Click on **Saved view list**  icon; choose [**Standard Orientation**]. Your part should look similar to that shown in Figure.

12. Select [**File**] -> [**Save**] from Menu Bar then click [**Ok**] to save the part.



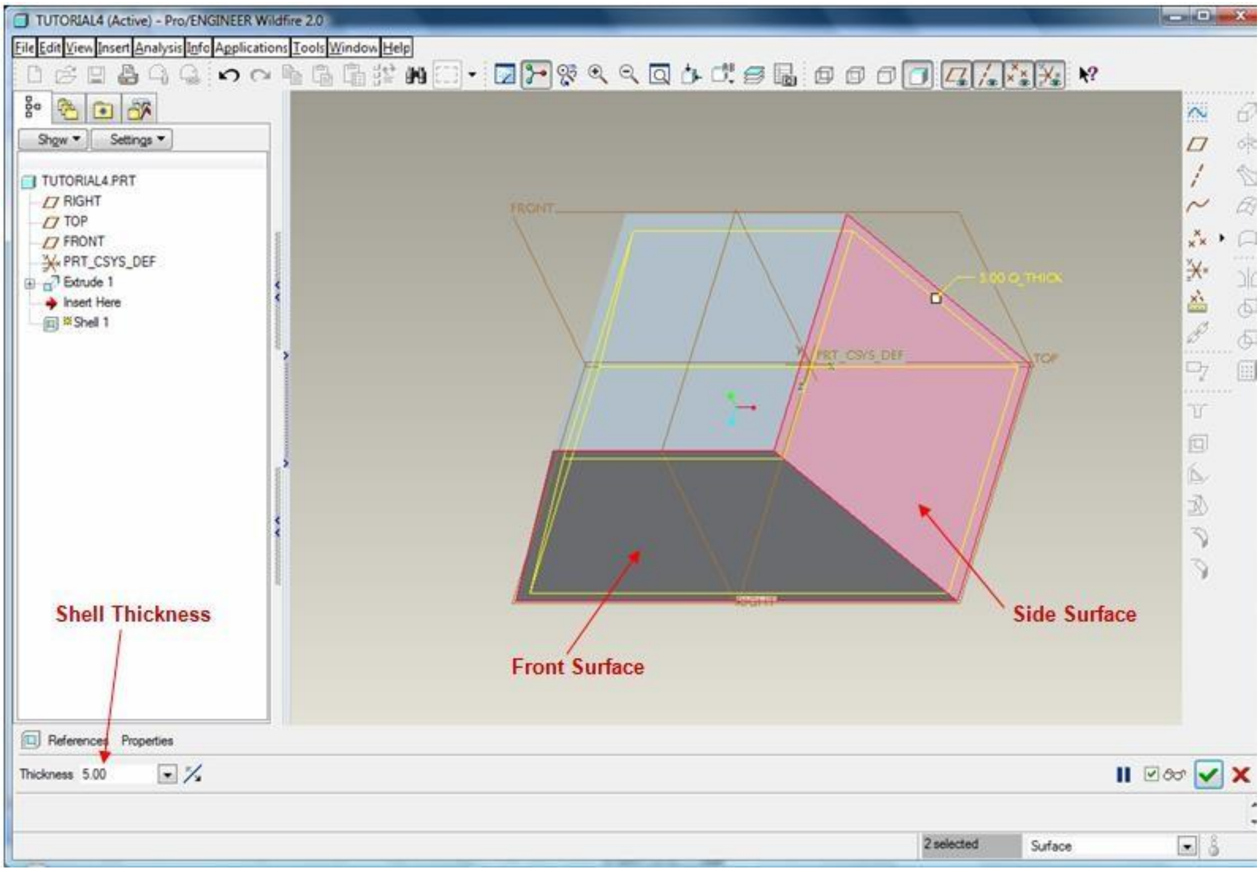
Creating shell and another extrusion:

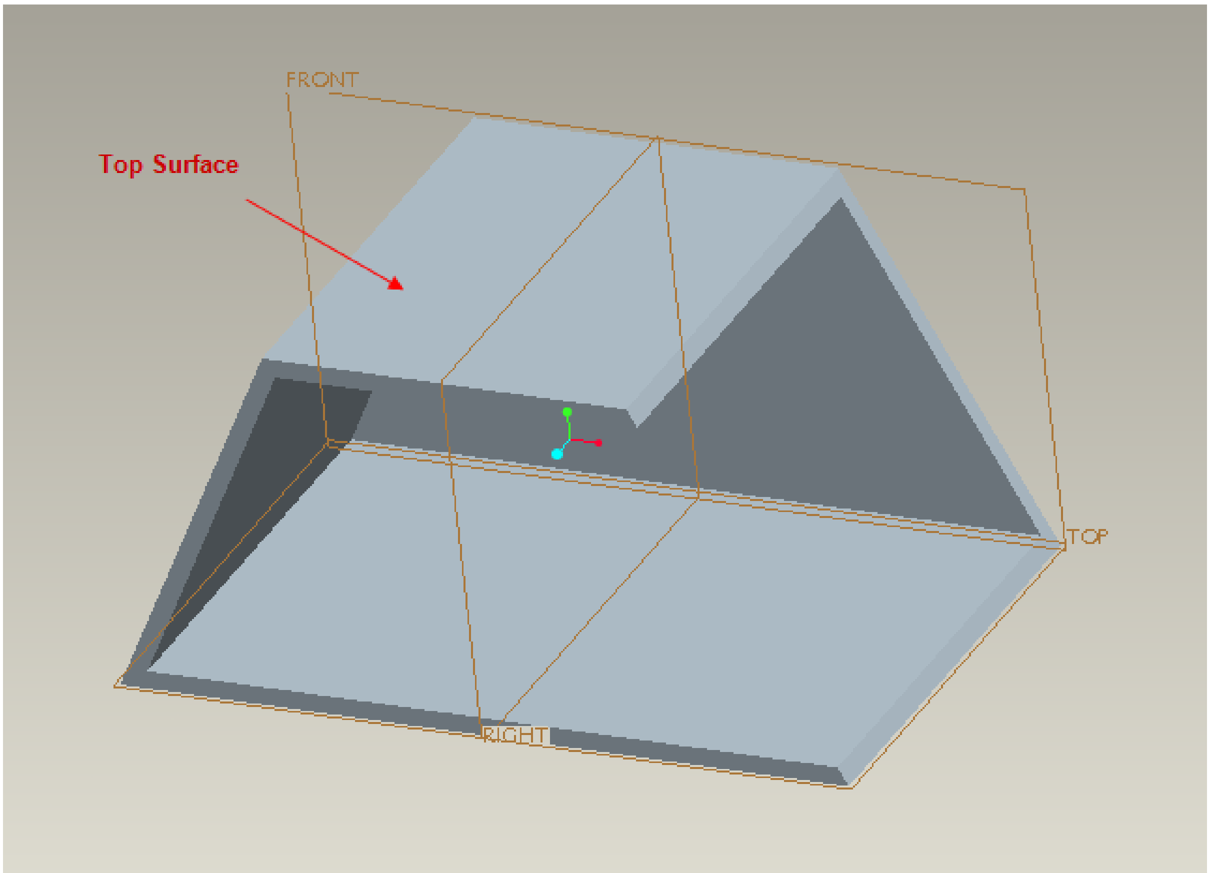
Select the **Shell**  icon from the Toolbar at the right of the screen (*or choose [**Insert**] -> [**Shell**] from the Menu Bar)*. The options of the Shell tool will be displayed in Dashboard at the bottom of the Pro/E main window.

Form the Dashboard set the **Thickness** of the shell to be **5** and hit Enter.

Using the **left mouse button** select the ***Front Surface*** as shown in Figure (it should become highlighted in pink as seen in the figure), then while holding down the [**Ctrl**] button on the keyboard click on the **Side Surface** as shown in the figure (both surfaces should become highlighted as seen in the figure).

Click the **check**  button to complete the shell. Your part should look similar to that shown in Figure.


TUTORIAL4 (Active) - Pro/ENGINEER Wildfire 2.0
File Edit View Insert Analysis Info Applications Tools Window Help
Show
Settings
TUTORIAL4.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Extrude 1
Insert Here
Shell 1
Shell Thickness
Front Surface
Side Surface
References
Properties
Thickness 5.00
2 selected
Surface


FRONT
Top Surface
TOP
RIGHT

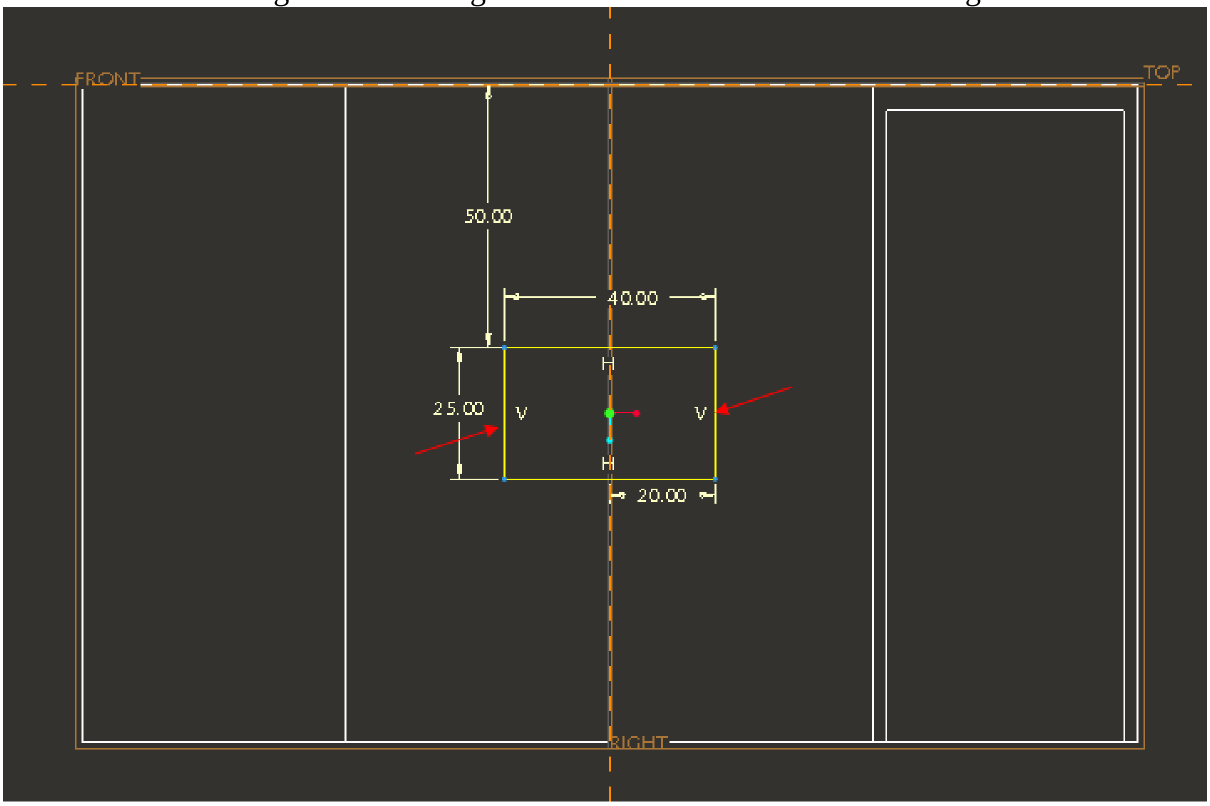
1. Select the **Extrude** icon from the Feature Creation Toolbar.

2. Select the [**Placement**] button then the [**Define**] button from the Dashboard.

3. Select the ***Top Surface*** of the part *(indicated in Figure)* as your sketching plane, then click the [**Sk** button.]

4. [**Close**] the References dialog box.

5. Select the **Rectangle** icon from Sketcher Toolbar and create the rectangle shown in Figure an dimension it as shown in the figure.

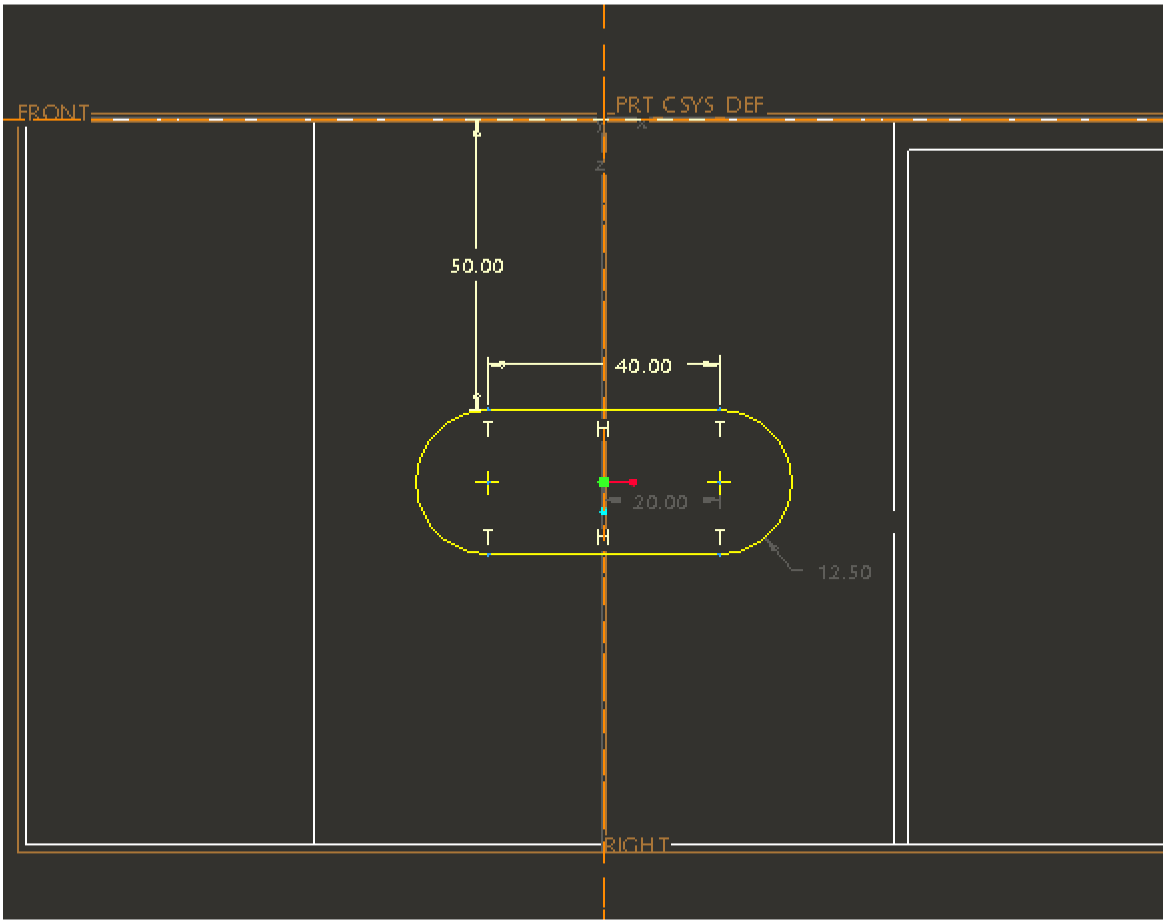


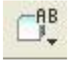
1. Now select the two vertical lines *(as indicated in Figure)* and delete them using the [**Delete**] button on your keyboard.

2. After deleting the two lines, select the **Arc** icon from the Sketcher Toolbar and create the two tangent arcs as shown in Figure. Make sure that the dimensions are same as what is shown in the figure.

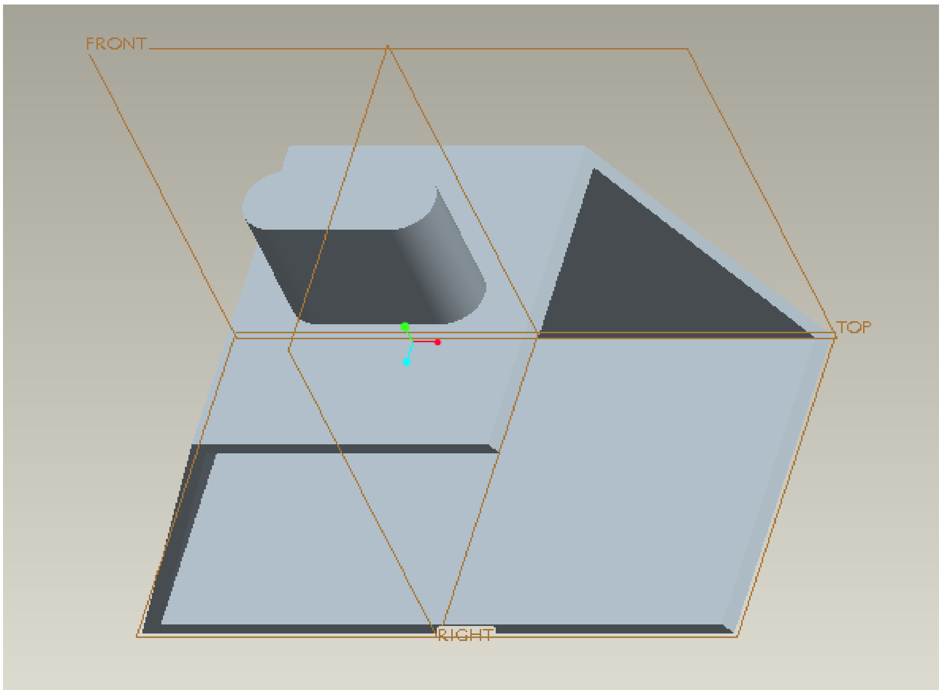
Note: if the arcs you created are not tangent then you can make them tangent using the **Impose sketcher constraint** icon then choosing the **Make two**

entities tangent icon. By clicking the line first then the arc, they will become tangent.



1. Click on the **Check**  button to exit Sketcher.
2. Click on **Saved view list**  icon; choose [**Standard Orientation**].

From the Dashboard set the *Extrusion depth* to be **50**, and click **Check**  button. Your part should look similar to that shown in Figure.



Select [**File**] -> [**Save**] from Menu Bar then click [**Ok**] to save the part.

Important Note: The order in which you define the **shell** is important**.** Note that we created the second extrusion after we defined the shell, this way the second extrusion will not be shelled. However, if we defined the shell after creating the second extrusion, the two extrusions (i.e., the entire part) will be shelled. If you want to test that you can choose the **Shell 1** from the model tree and drag it using the mouse to place it after Extrude 2, then rotate the part and see how Extrusion 2 become empty from the inside.

Creating Revolves:

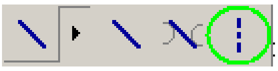
Select the **Revolve**  icon from the Feature Creation Toolbar.

Select the [**Placement**] button then the [**Define**] button from the Dashboard.

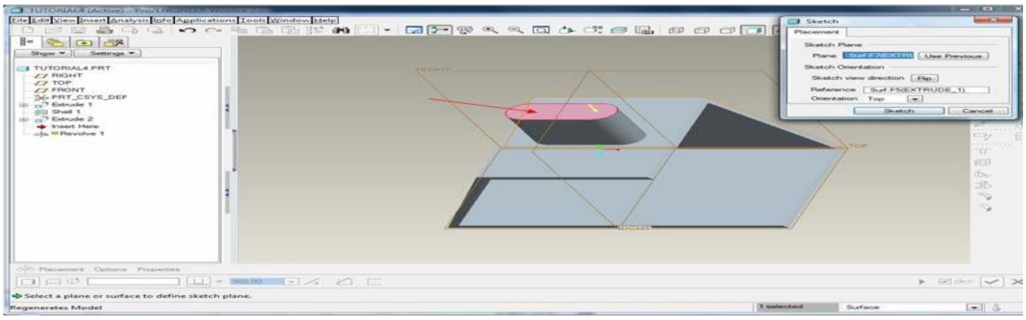
Select the ***Top Surface*** of *Extrude 2* to be your sketching plane, as shown in Figure 4.8, then select the [**Sketch**] button.

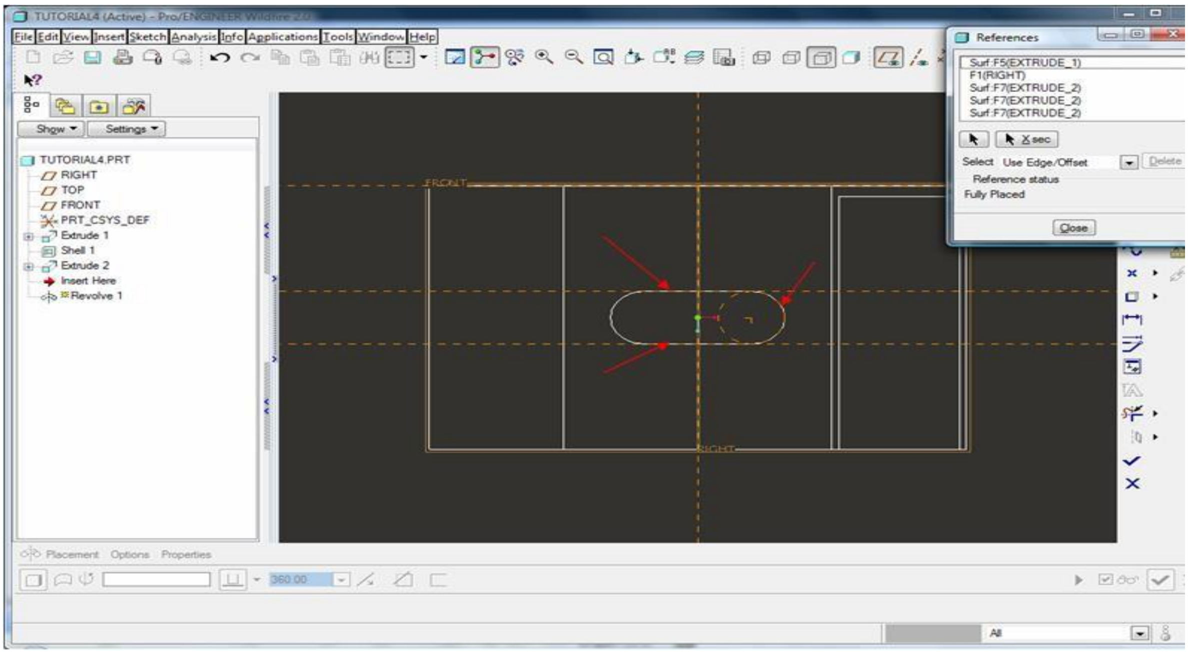
Do not close the References window, and make sure to change the view mode to wireframe  such that you can see the outline of Extrude 2.

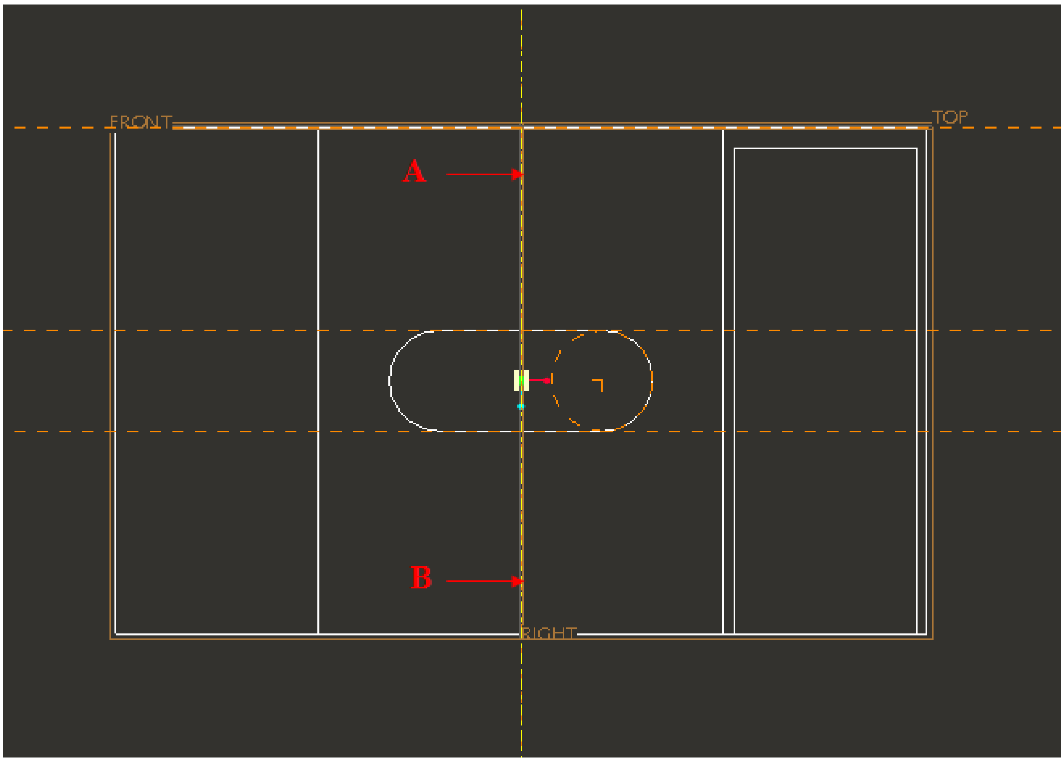
Now we will add additional references that will help us in drawing the sketch. Click on the two horizontal edges of *Extrude 2* and the edge of the arc as shown in Figure. After selecting the additional references [**Close**] the pop-up window.

Select the Centerline  icon from Sketcher Toolbar.

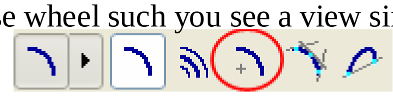
Click point A and point B as shown in Figure 4.10 to create a vertical center line.







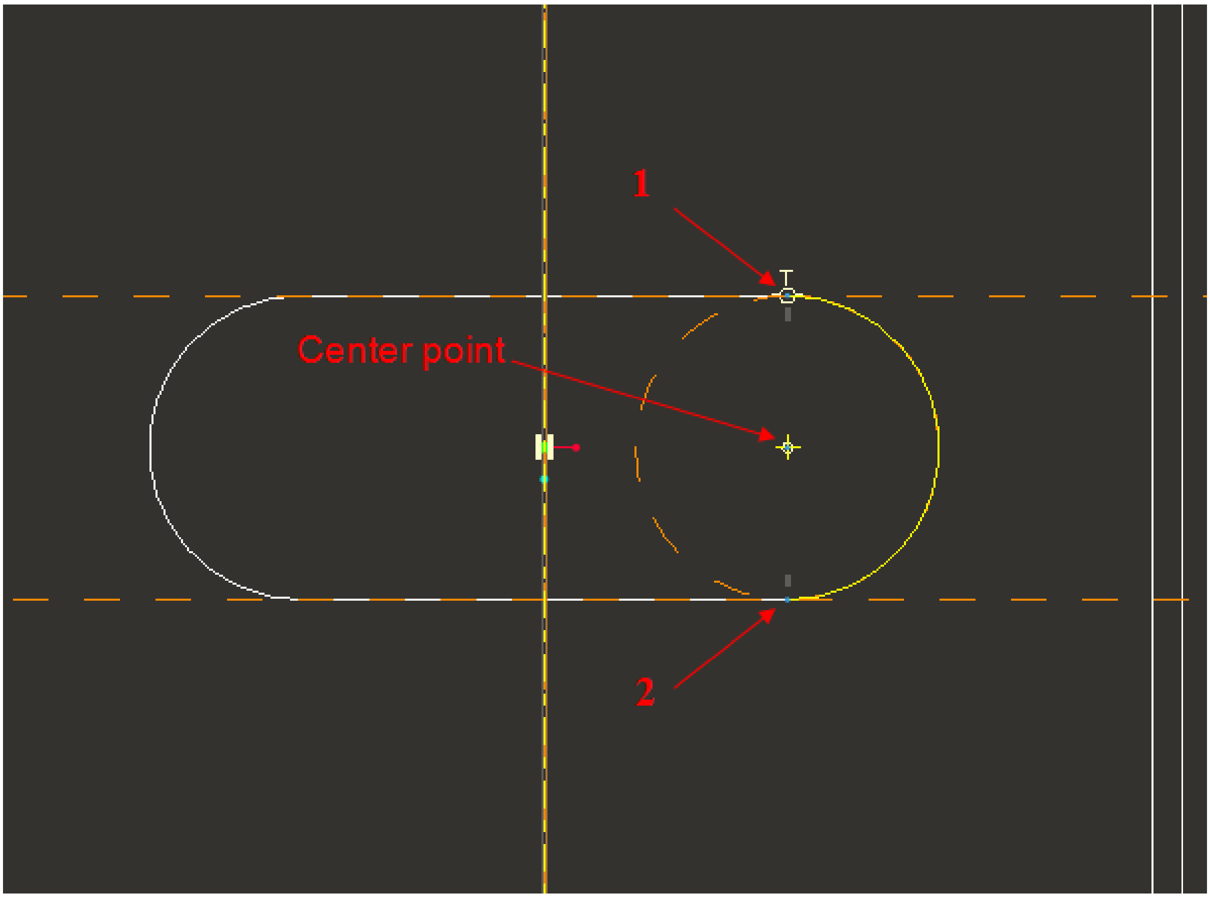
Zoom in on *Extrude 2* using the mouse wheel such you see a view similar to that in Figure.



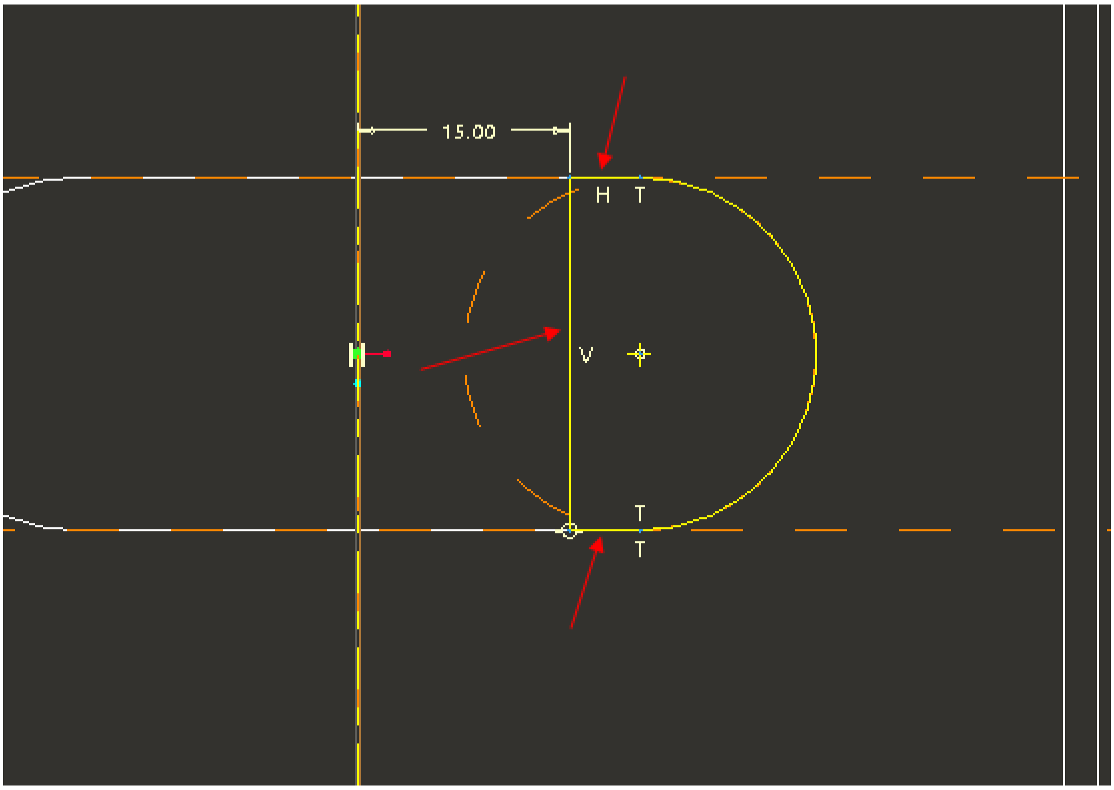
Click on the arrow next to the arc icon  *then choose the Create arc by picking its center and endpoints icon.*

Using the left mouse button <u>first click the center point</u> (as shown in Figure 4.11) to be the center of the arc then clicking on <u>point 1</u> to be the start point then clicking <u>point 2</u> to be the end point of the arc.

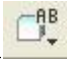
Note: make sure that the T (tangent) appears next to the ends of the arc. If for any reason the auto constraints did not work and dimensions are displayed on the figure, then just make sure that the radius of the arc is 12.5 and the horizontal distance from the end points to the centerline is set to 20.



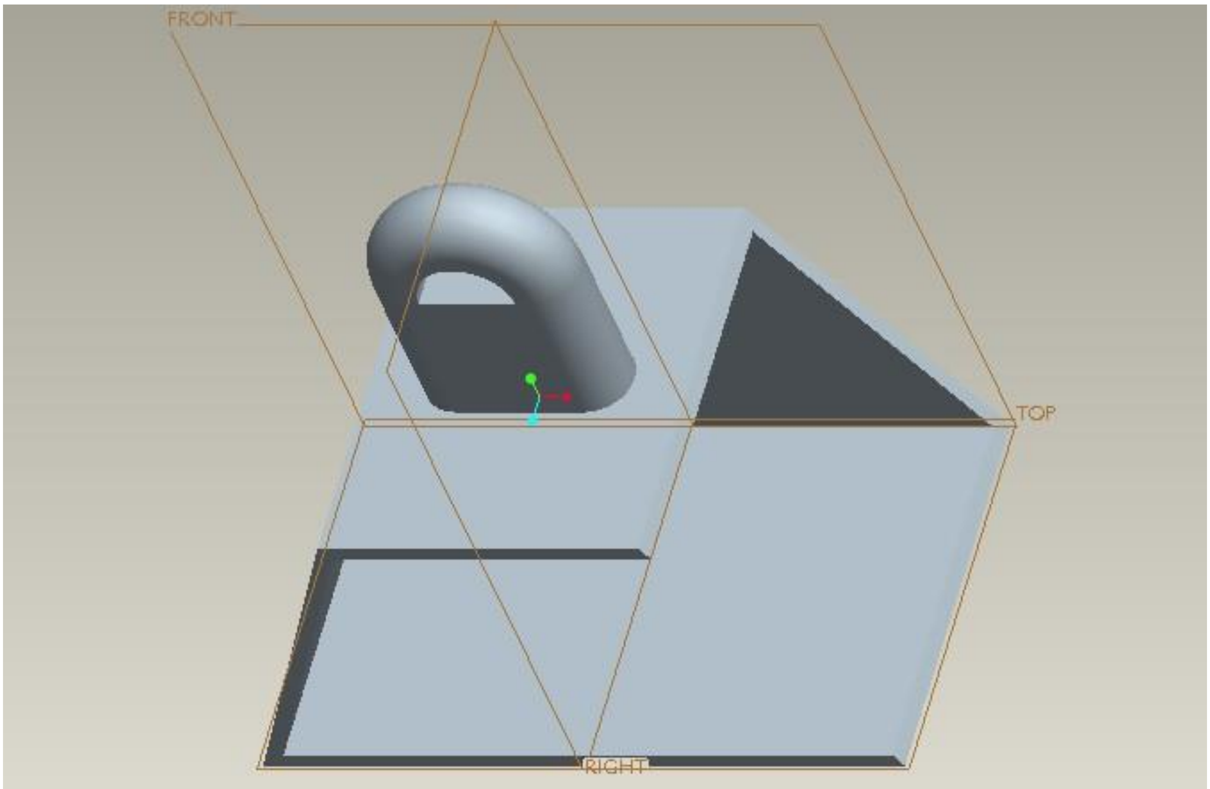
Select the **line**  icon then starting from the endpoint of the arc draw a horizontal line then a vertical line then a horizontal line again ending at the other endpoint of the arc, as shown in Figure 4.12. Set the distance between the vertical line and the centerline to be **15** as seen in the figure.



Click on the **check**  button to complete the sketch.

Click on **Saved view list**  icon; choose [**Standard Orientation**].
From the Dashboard set the **Revolve angle** to be **180**, make sure that the revolve direction is correct if not flip it using the *arrow*  icon.

Click the **check**  button to complete the revolve. Your part should now look similar to that shown in Figure.



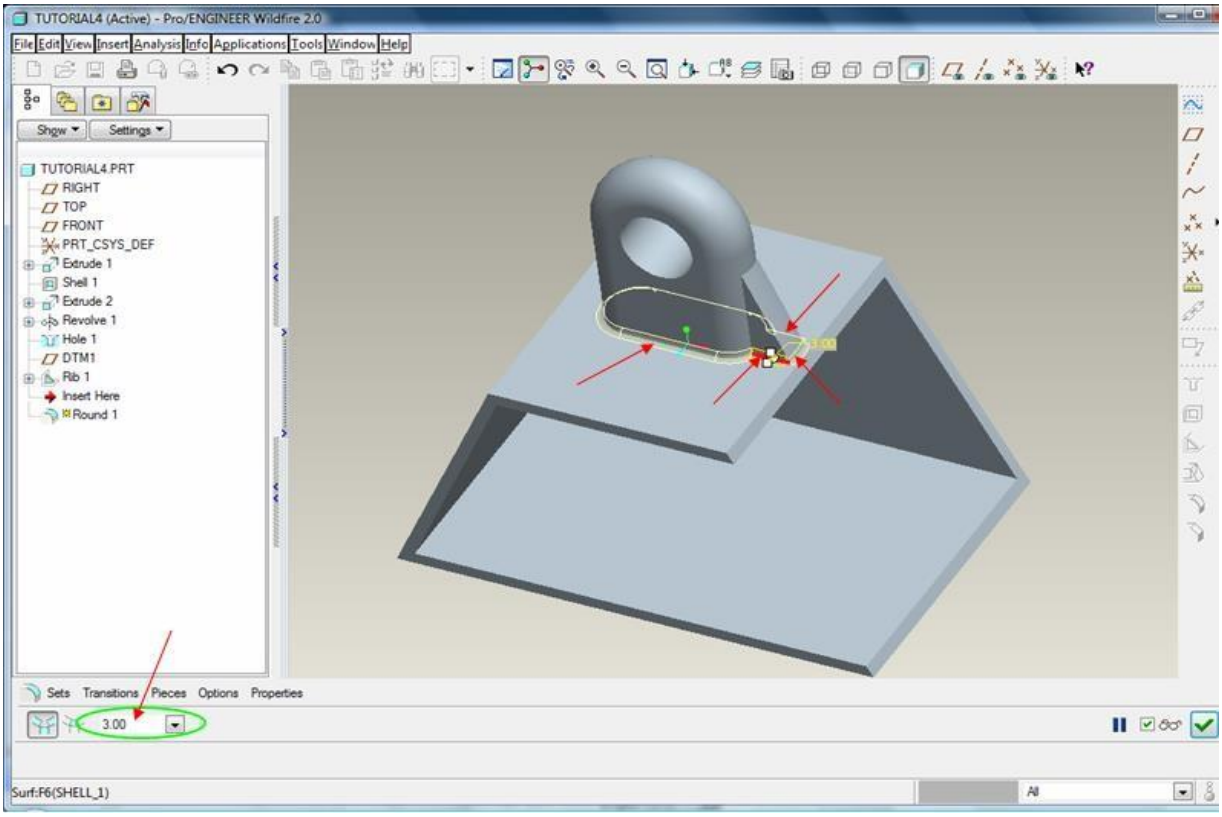
Select [**File**] -> [**Save**] from Menu Bar then click [**Ok**] to save the part.

Chamfering:

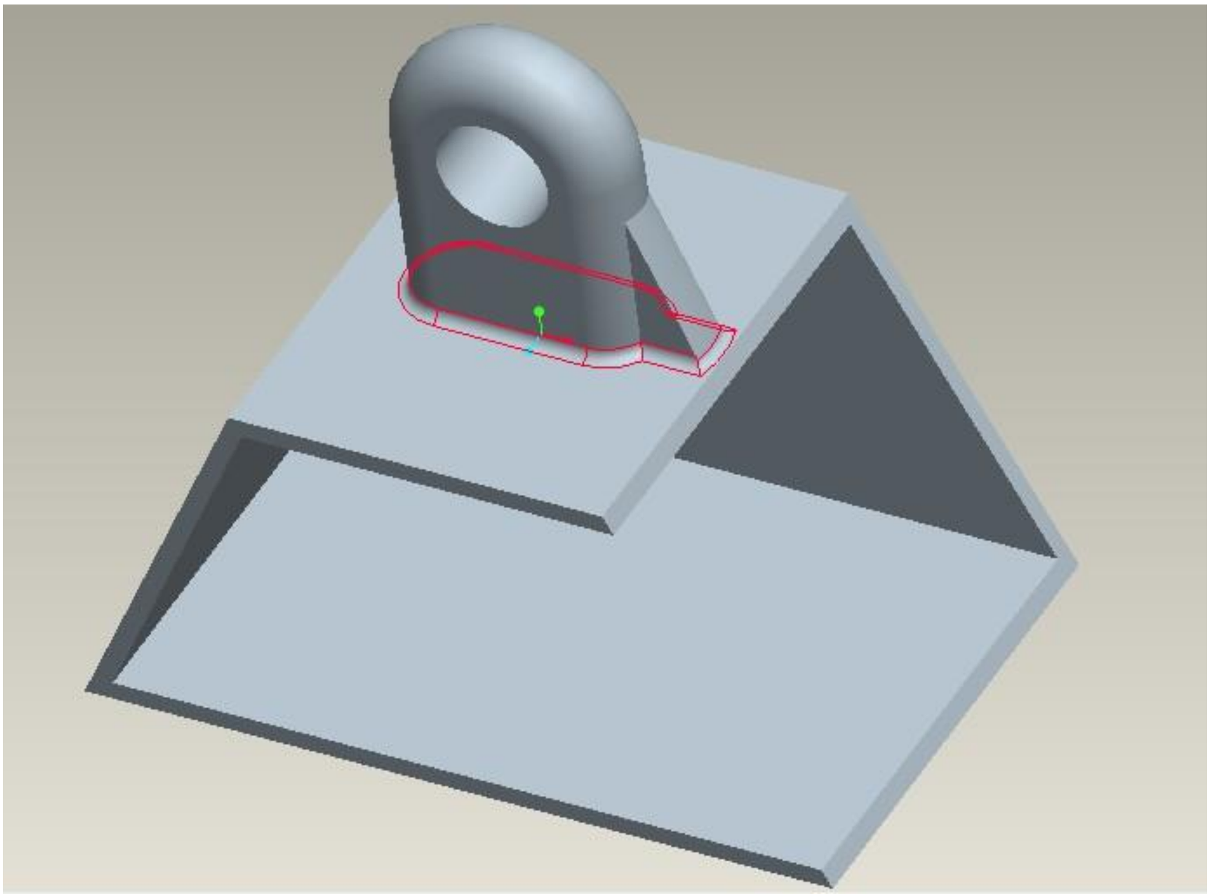
1. Select the **Round**  icon from the Toolbar at the right of the screen (*or choose [**Insert**] -> [**Round**] from the Menu Bar)*. The options of the Round tool will be displayed in Dashboard at the bottom of the Pro/E main window.

2. From the dashboard set the ***Radius*** of the round to be **3** as seen Figure.

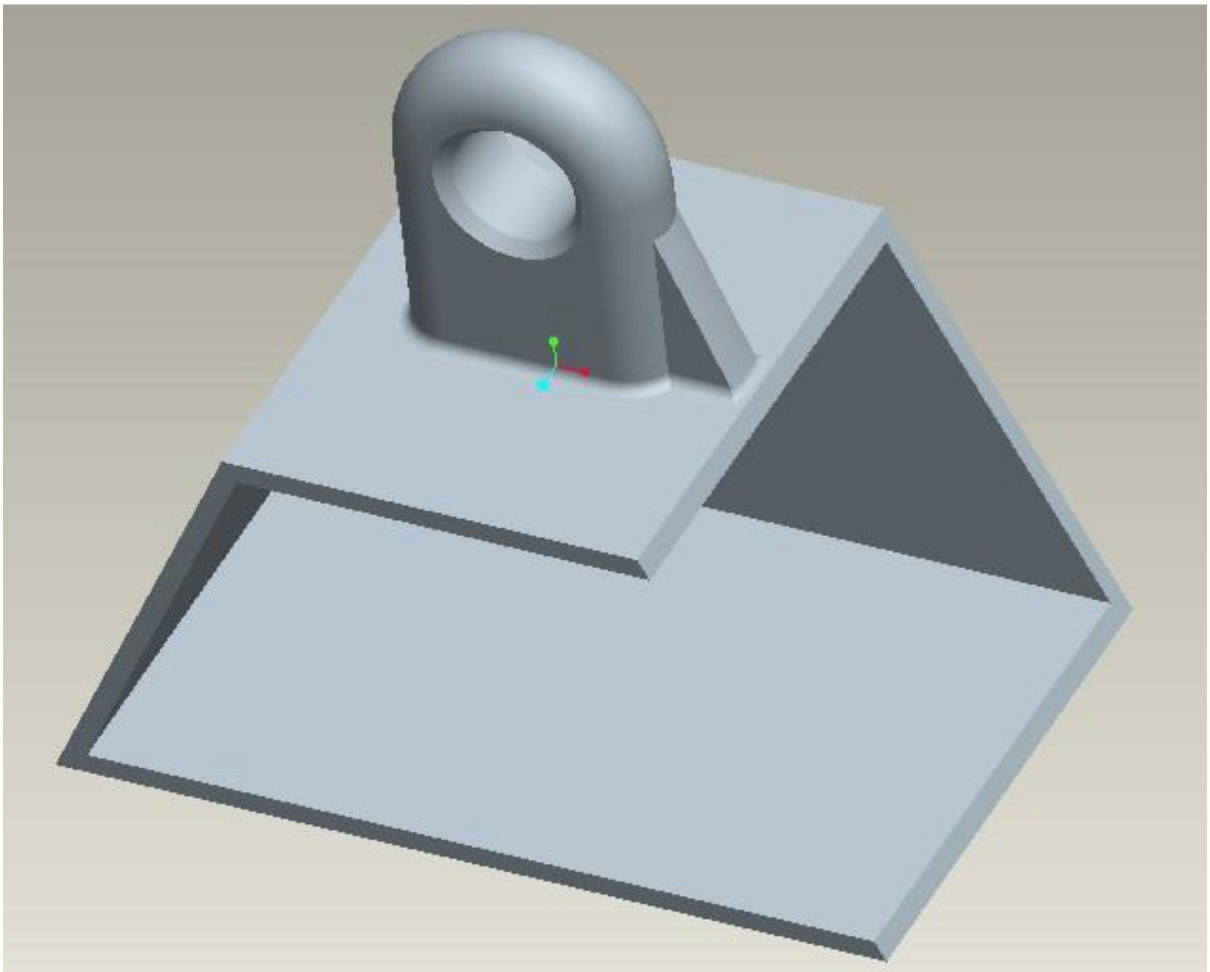
3. With the **left mouse button** click the lower edge of Extrude 2 and <u>all</u> the lower edges of the Rip as shown in the figure.



Click the **check**  button to complete the round. Your part should look similar to that seen in Figure.



4. Select **Chamfer**  icon from the Toolbar at the right of the screen (*or choose [**Insert**] -> [**Chamfer**] from the Menu Bar)*. The options of the Chamfer tool will be displayed in Dashboard at the bottom of the Pro/E main window.
5. From the Dashboard set the chamfer ***depth*** to be **3**, as seen in Figure 4.26.
6. With the **left mouse button** click on the edge of the hole as shown in the figure.
7. Click the **check**  button to complete the chamfer. If you followed the instructions correctly, your part would look similar to that shown in Figure.



Select [**File**] -> [**Save**] from Menu Bar then click [**Ok**] to save the part.

Experiment-3

Advance Solid Part Modeling Advanced Selection, Creating Sweeps and Blends Sweeps with Variable Sections Helical Sweeps & Swept Blends Relations, Parameters & Family Tables Measuring and Inspecting Models.

Introduction:

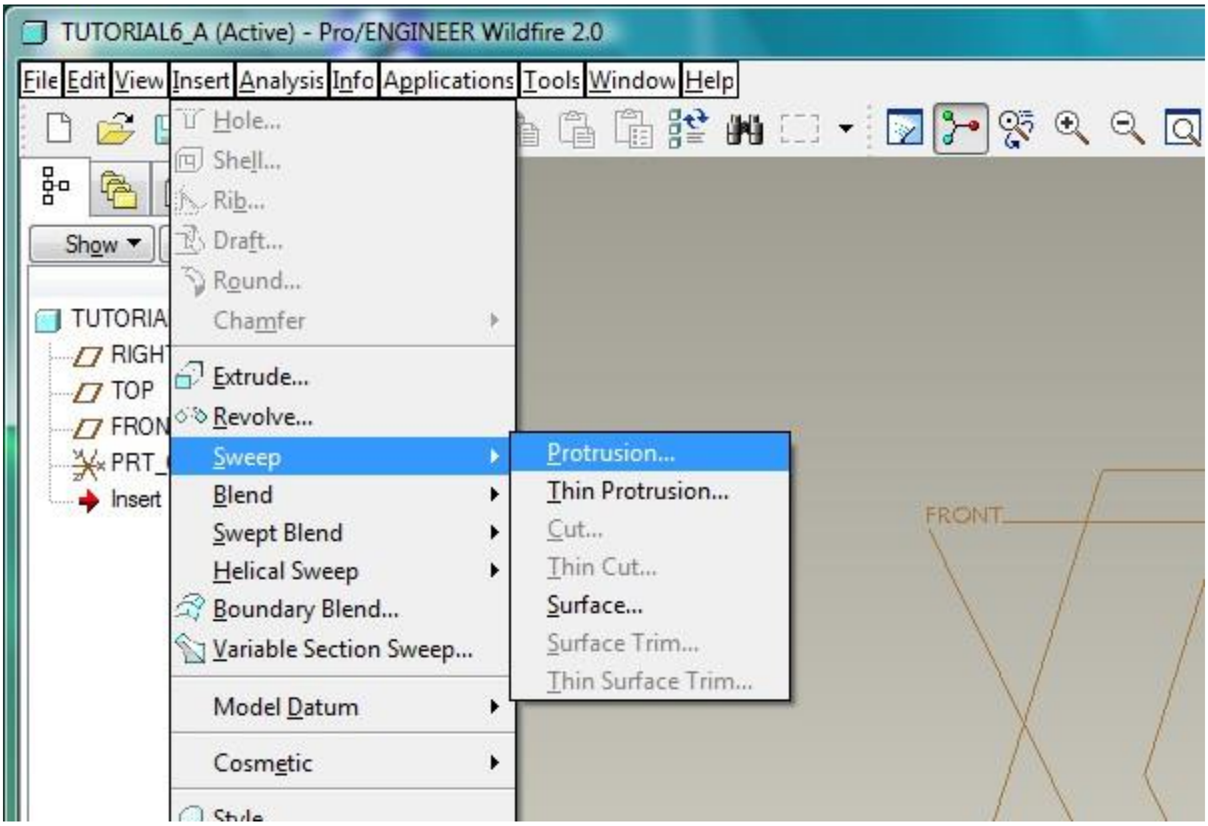
Sweeps, blends and helical sweeps are some of the useful Pro/E commands that can be used for creating parts which can not be made using extrusions or revolves. In an extrusion, the cross-section moves normal to the sketching plane (i.e., moves along a straight line) and in a revolve, the cross-section moves around an axis (i.e., moves along a circle). In a sweep, however, in addition to the cross-section, you also sketch the path along which this cross section will be moved. Blends allow you to draw several “spaced” cross-sections and join them together to create a solid. Finally, helical sweeps allow you to draw a cross- section that moves along a helix such as in the case of springs or screws.

Creating a swept part:

1. Start Pro/E Wildfire.
2. Choose [**File**] -> [**Set Working Directory…**], and select a folder to save your work in.
3. Select [**File**] -> [**New**], and type the part name [**Tutorial6_A**] in the Text Box, then click the [**Ok**] button.

Select [**Insert**] -> [**Sweep**] -> [**Protrusion**] from the Menu Bar, as seen in Figure.

> Important note: Similar to the Extrude and Revolve commands, you can use the sweep option to create a solid, cut, thin solid, etc., however, you need to choose the option you want before stating the sweep.



Two pop-up windows will open as shown in Figure 6.2. The upper pop-up window is the Sweep definition menu which contains all the elements that need to be defined in order to complete the sweep and this window will remain open until you complete the sweep. The small arrow (indicated in the figure) identifies the step you are currently working on. The lower pop- up window is the Menu Manager and the contents of this menu will change according to your progress.

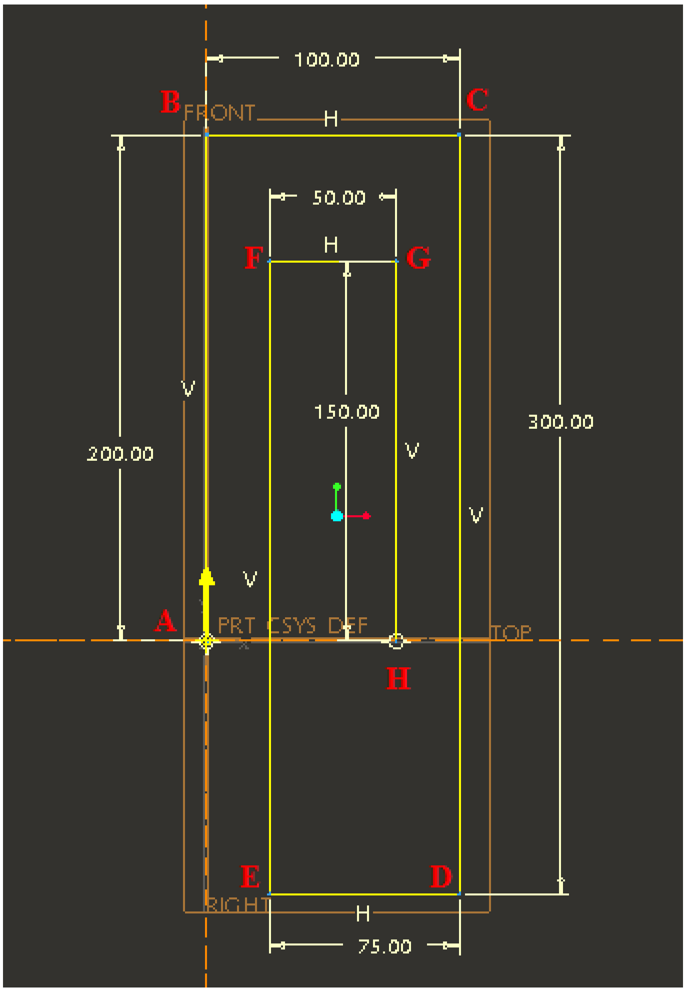
1. Select [**Sketch Traj**] from the **SWEEP TRAJ** menu in

2. the *Menu Manager. This will allow you to sketch th trajectory of the sweep.*
3. Select the plane labeled as **FRONT**, and then select [**Okay**] from the

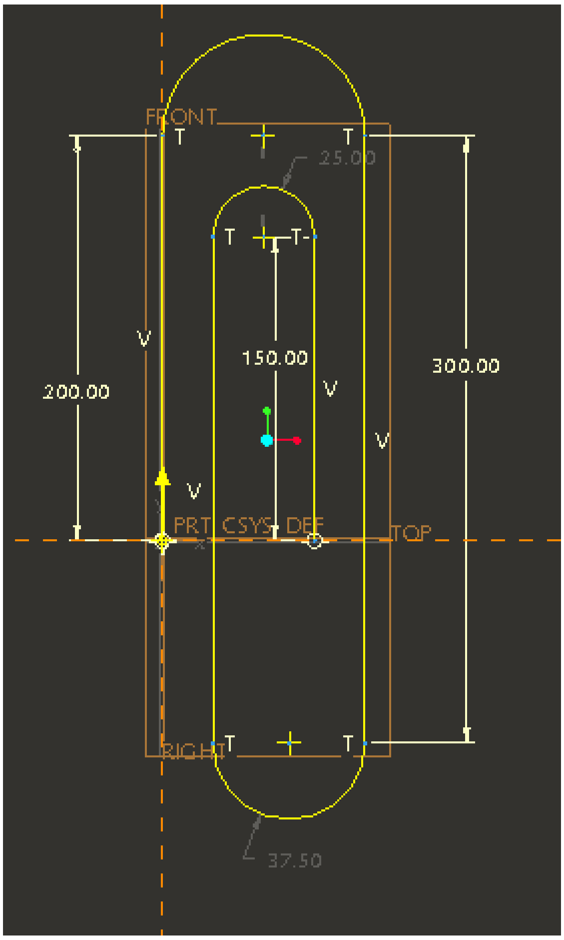
DIRECTION menu in the *Menu Manager*.

4. Select [**Default**] from the **SKET VIEW** menu in the *Menu Manager*. Pro/E will switch to the Sketcher Mode.
5. [**Close**] the References pop-up window.
6. Using the **line** tool  draw the path shown in Figure starting from point **A**, **B**, ... to pint **H**. Note *that the yellow arrow seen in the sketch indicates the start point*.
7. Set the dimensions of the sketch as shown in the figure.

Delete the three horizontal lines, then using the **Arc** tool  draw

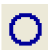


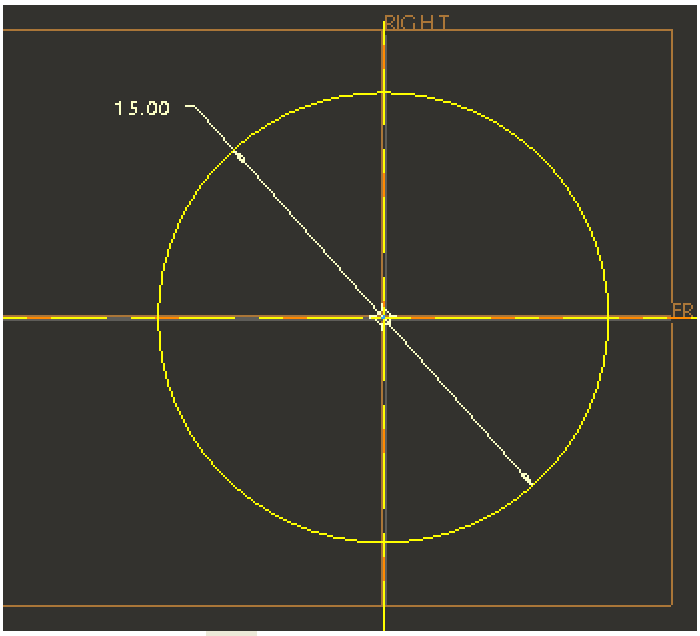
three tangent arcs connecting the vertical lines as seen in Figure.



Click the **Check** button  from the Sketcher Toolbar to complete the sketch of the trajectory.

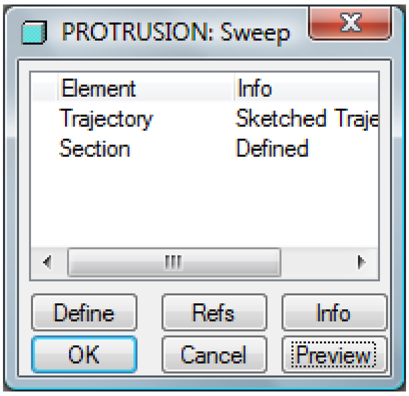
Note how the view gets rotated such that you can start drawing the cross- section of the swept part. The two yellow crossed lines that appear in the sketcher screen indicate the location of the trajectory start point. The location of your cross-section relative to that start point defines how the section will be swept relative to the trajectory.

Using the **circle** tool  draw a circle centered at the crosses lines and set the diameter of the circle as seen in Figure.



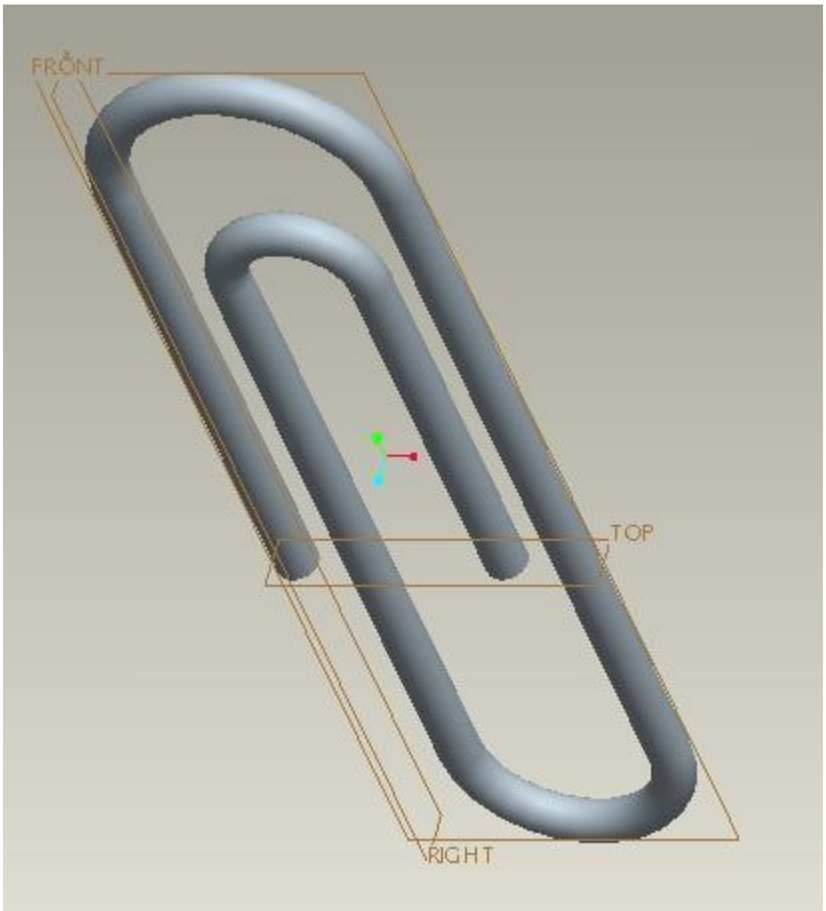
1. Click the **Check** button  from the sketcher toolbar to complete the sketch of the cross-section.

Note: Once you completed the definition of the cross section you can see how the swept part looks by clicking the [**Preview**] button in the Sweep definition pop-up menu, see Figure 6.6. Also, note that the information given in that window indicates that both elements of the sweep have been completed. If you need to modify any of the elements of the sweep (Trajectory or Section), you just need to select it from the window then press the [**Define**] button.



Click the [**OK**] button from the Sweep definition pop-up menu to complete the swept part.

Select [**View**] -> [**Orientation**] -> [**Standard Orientation**] from the menu bar. Your part should look similar to that shown in Figure.



1. Select [**File**] -> [**Save**] from Menu Bar then click [**Ok**] to save the part.
2. Select [**File**] -> [**Close window**] from Menu Bar to close the part such that you start a new part.

Creating blending part:

1. Select [**File**] -> [**New**], and type the part name [**Tutorial6_B**] in the Text Box, then click the [**Ok**] button.
2. Select [**Insert**] -> [**Blend**] -> [**Protrusion**] from the Menu Bar. *The Menu Manager pop-up window will open.*

3. Select [**Done**] from the Menu Manager pop-up window. The Blend definition pop-up menu will now appear and it will remain open until you complete the blend. Similar to the Sweep, this menu contains all the elements that need to be defined in order to complete the Blend and the small arrow indicates the step you are currently working on.
4. Select [**Smooth**] and then [**Done**] from the **ATTRIBUTES** menu in the Menu Manager.

Select the plane labeled as **FRONT**, then select [**Okay**] from the **DIRECTION** menu in the Menu Manager, as seen in Figure.

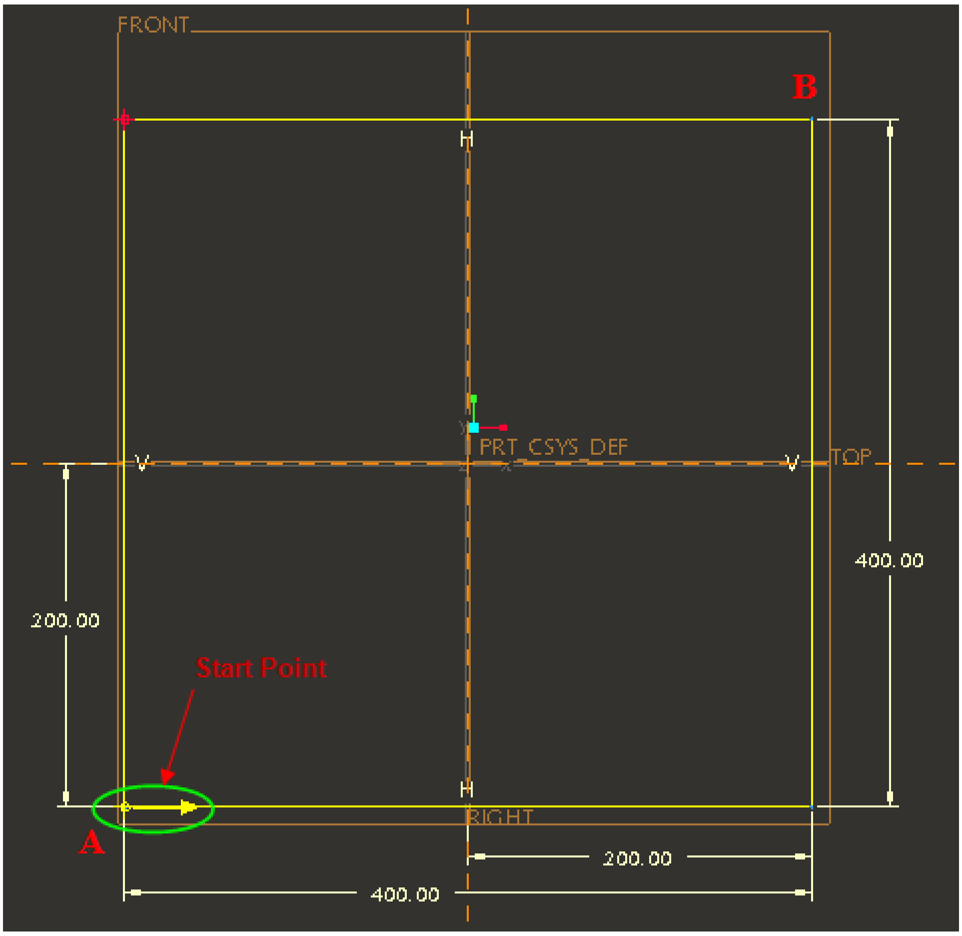
5. Select [**Default**] from the **SKET VIEW** menu in the Menu Manager. Pro/E will switch to the Sketcher Mode.

6. [**Close**] the References pop-up window.

7. Draw the square shown in Figure 6.9 using the **rectangle** tool  and starting from point **A** to point **B** as shown in the figure, and then set the dimensions as seen in the figure.

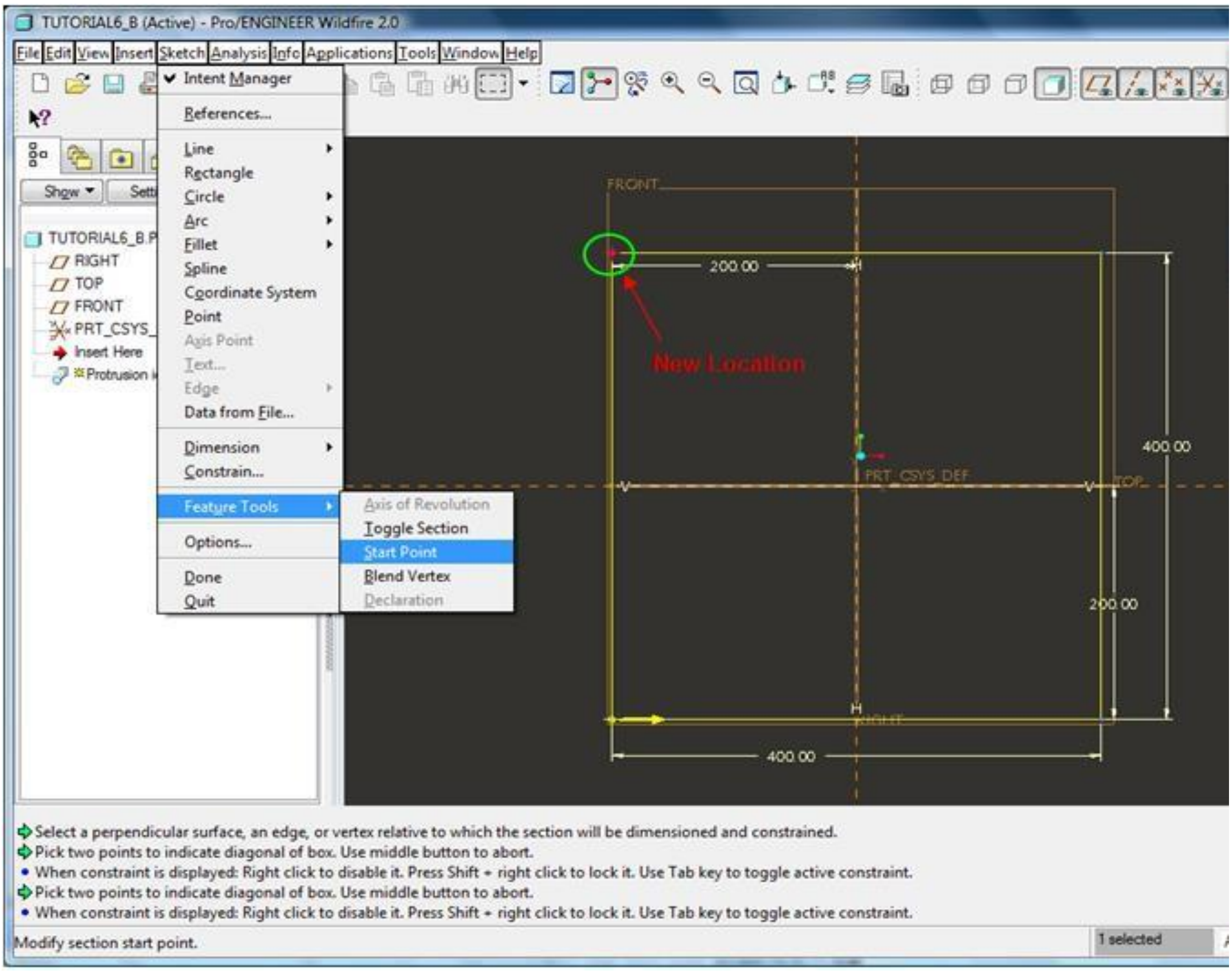
   Note: *A yellow arrow, as indicated in the figure, will be displayed at point*

   A *(if you start from point **A**) where this arrow indicates the start point. The meaning of start points for blended sections will become clear as you go on with this tutorial.*



[Figure 6.9]

8. Now we will move the start point to a new location, to do so, first you need to select the new location of the start point by clicking on the new point such that it is highlighted in red (as seen in Figure 6.10) then from the menu bar select [**Sketch**] -> [**Feature Tools**] -> [**Start Point**].



<u>Now we completed the sketch of the first cross-section of the Blend, so we will move to the next section.</u>

9. From the Menu Bar select [**Sketch**] -> [**Feature Tools**] -> [**Toggle Section**].

   Note: *Once you move to the next cross-section, the <u>previous cross- section will not disappear</u> but its color will change to light gray. You still can modify the dimensions of the previous section if you want to do that.*

10. Select the **line** tool  and draw two diagonal lines connecting the corners of the square as shown in Figure 6.11.

Select the two diagonal lines (such that they are highlighted in red as seen in the figure, you need to hold down the **Ctrl** button on the keyboard while selecting the second line) then from the menu bar

1. select [**Edi** Select the **Divide an entity**  icon from the Sketcher Toolbar (as shown in Figure 6.12) then divide the circle into *four arcs* by clicking on the points of intersection of the circle with the diagonal construction lines as indicated in the figure. *Note that the construction diagonal lines were created just to be used as guidance when dividing the circle.*

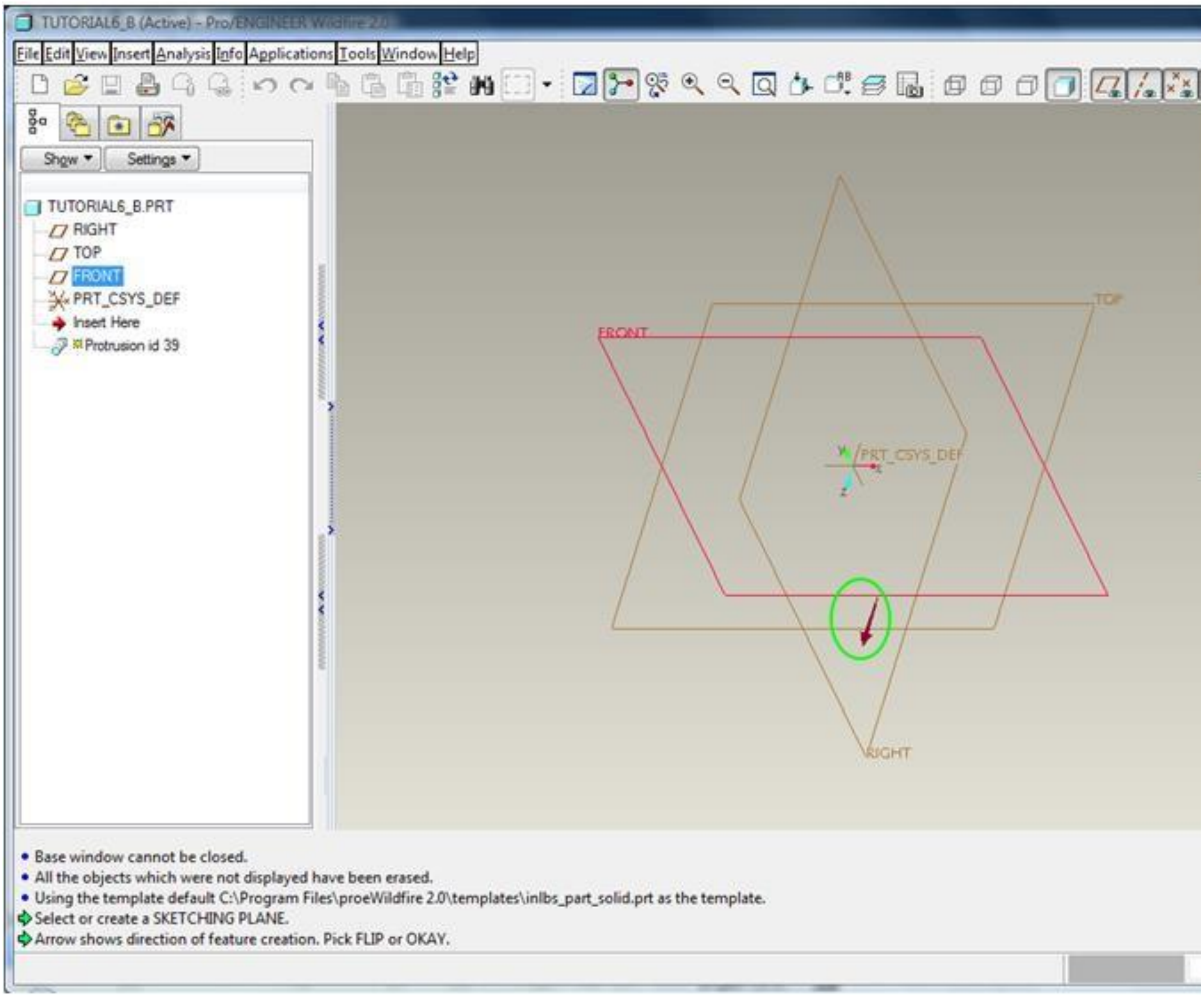
   Important note: *Make sure that you divide the circle into four arcs (no less and no more). You can make sure that the circle is now divided to arcs by moving the mouse over the circumference of the circle and observing that each time one of the four arcs is highlighted. The reason we have to divide the circle into four arcs it to match the number of corners "vertices" of this cross-section with the previous cross-section (i.e., the square which contain four corners). It should be understood that blended cross sections must have the same number of vertices (or segments).*

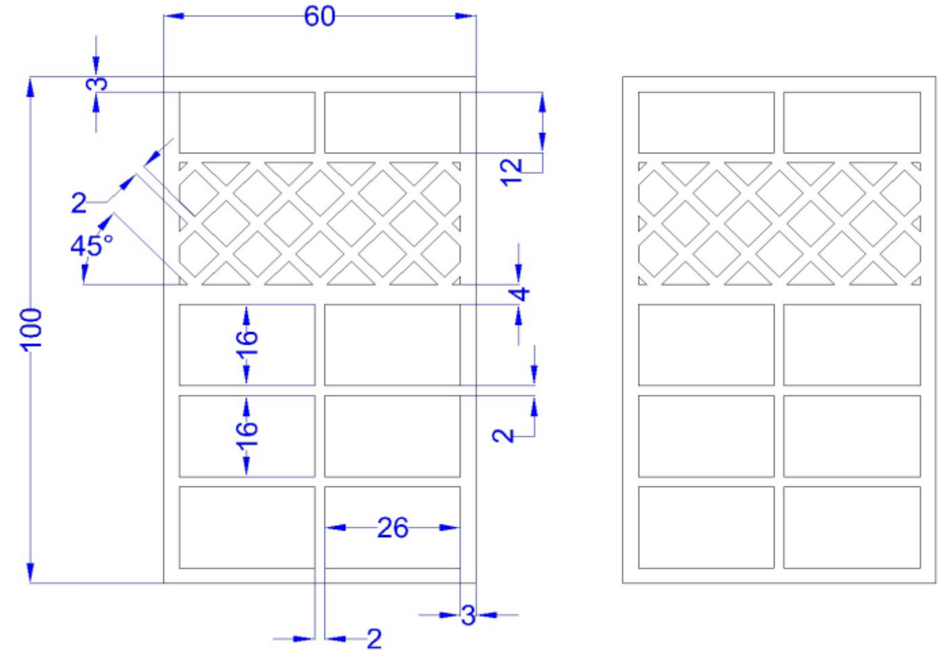
2. Once you divide the circle a start point arrow will be displayed at one of the points (the location of the start point depends on the order in which you divided the circle). If the location of your start point does not match the location of the start point shown in Figure 6.13, then move the start point to the location shown in the figure using the procedure described previously in step 9.

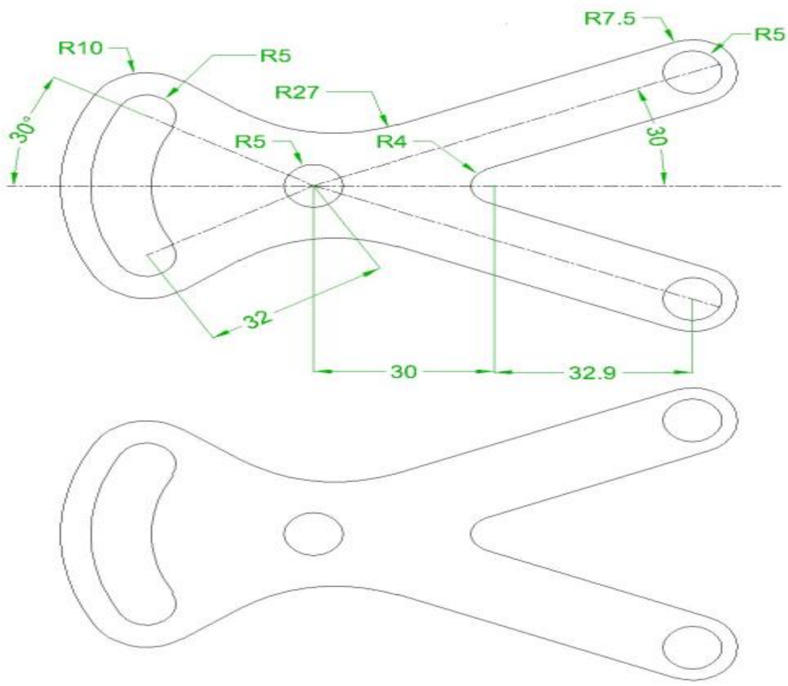
Note: When the different cross-sections of a blend are joined, the start point of each cross-section will be directly connected to the start point of the next cross-section and the remaining points will be connected Note that at the beginning we chose a Smooth blend and that option causes the cross sections to be connected using spline curves. Now we will change the option to Straight such that the sections will be connected using straight lines where that will enable us to recognize the different cross sections that we defined. To do that select [**Attributes**] from the Blend definition menu then click the [**Define**] button, as shown in Figure 6.16.

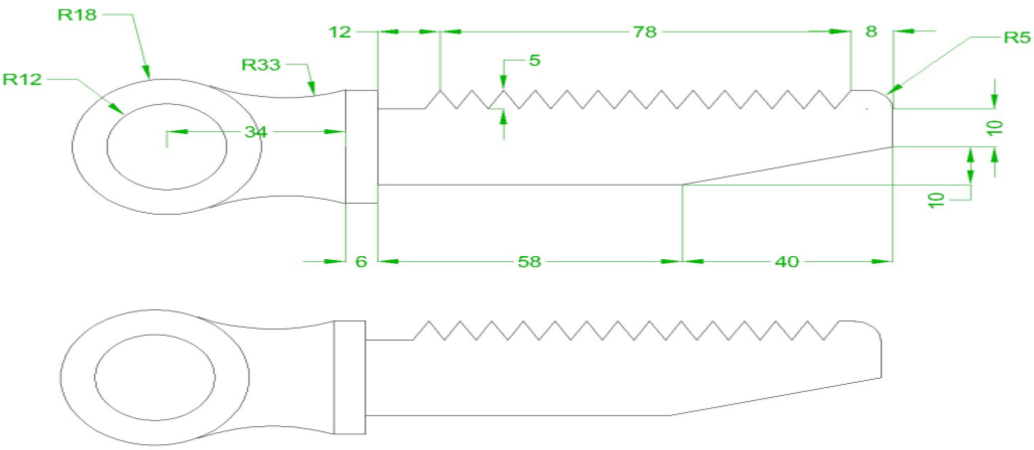
11. Select [**Straight**] from the **ATTRIBUTES** menu in the Menu Manager, then select [**Done**].
12. Click the [**Preview**] button from the Blend definition menu and see what the part looks like.

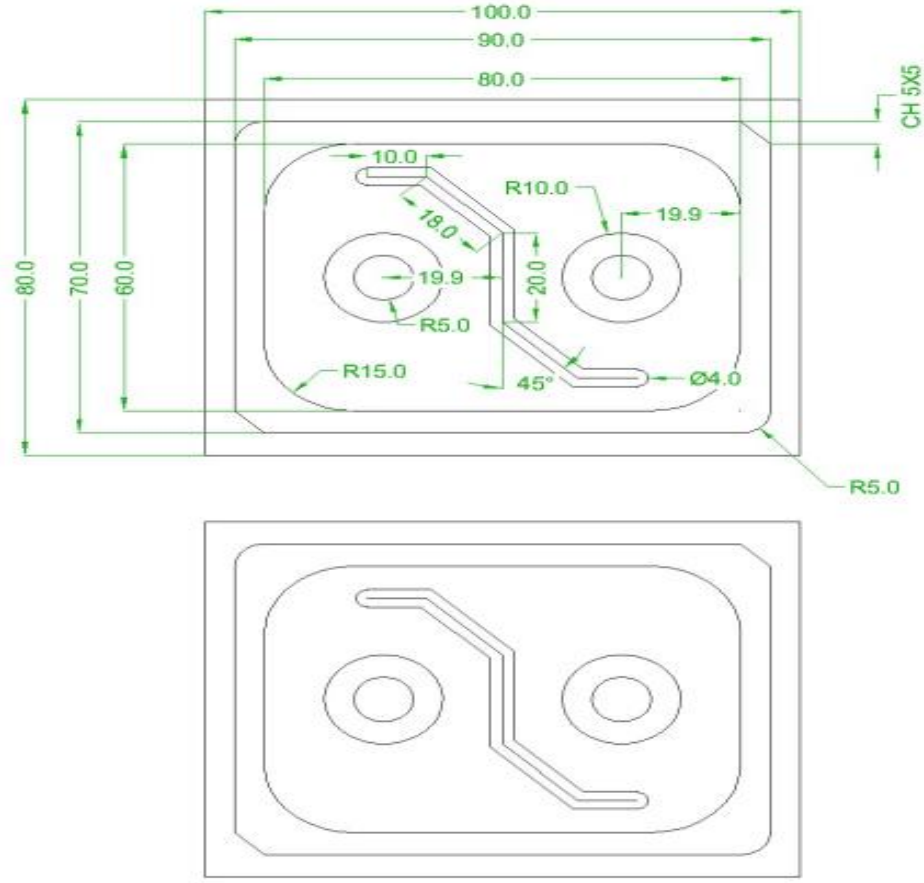
in the same order.**t**] -> [**Toggle Construction**] as shown in the figure




60
3
12
2
45°
100
4
16
16
2
26
3
2

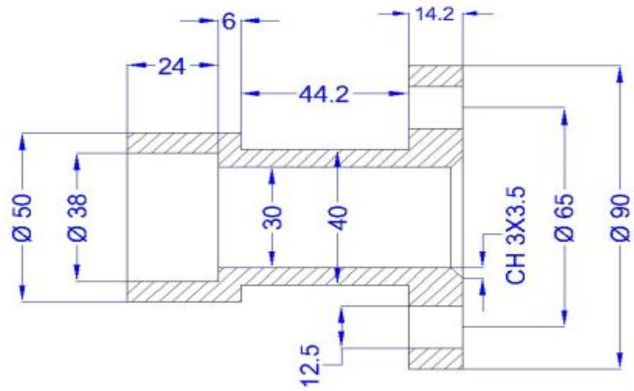

R7.5
R5
R10
R5
R27
30°
R5
R4
30
32
30
32.9


R18
R12
R33
34
12
78
8
R5
5
10
10
6
58
40

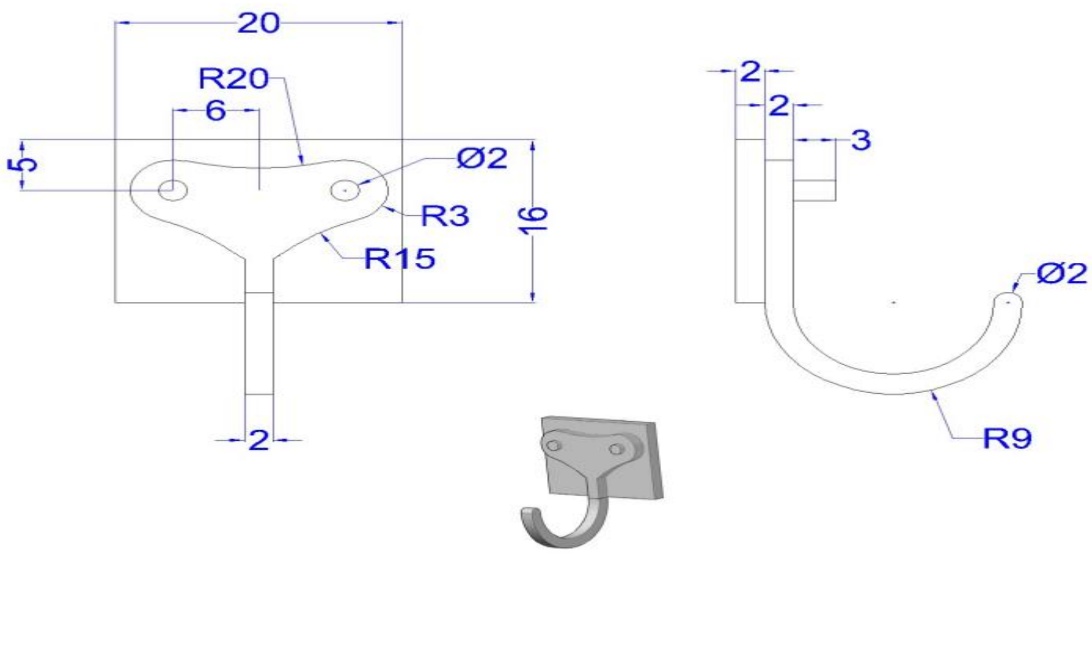

100.0
90.0
80.0
CH 5X5
10.0
R10.0
19.9
18.0
19.9
20.0
R5.0
80.0
70.0
60.0
R15.0
45°
Ø4.0
R5.0

3D EXERCISES

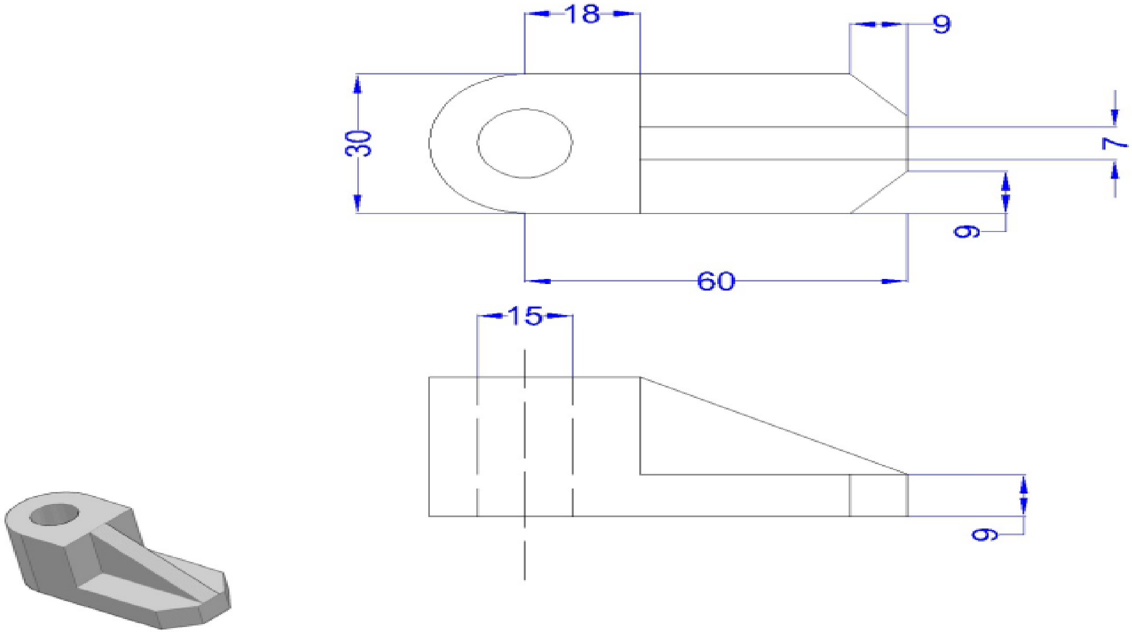
1.



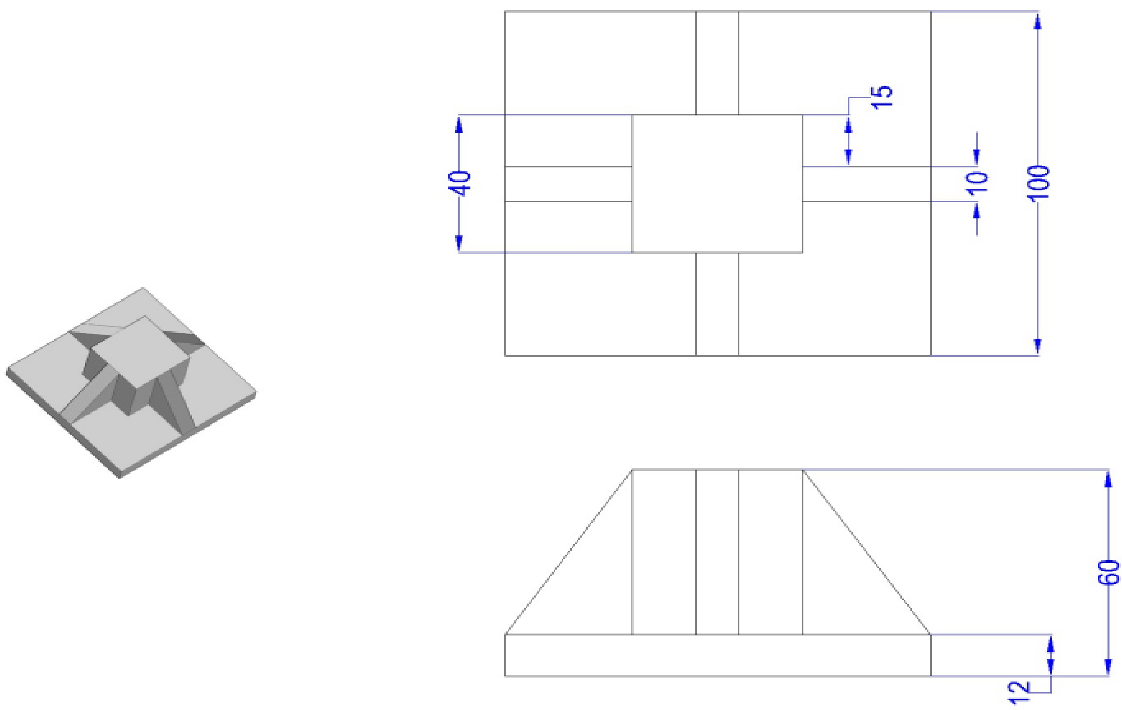
2.



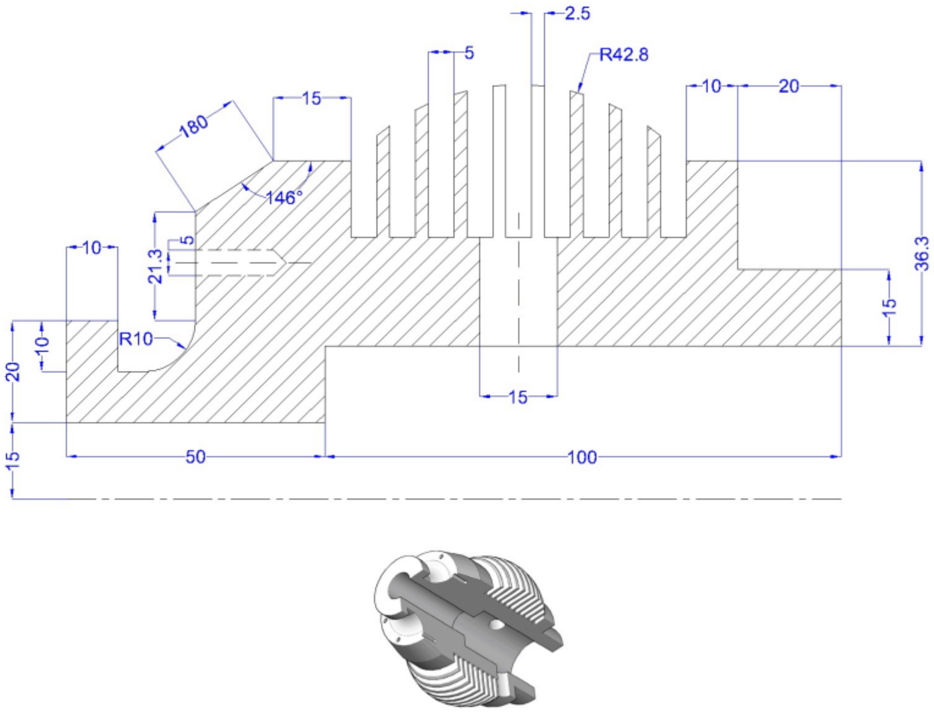
3.



4.

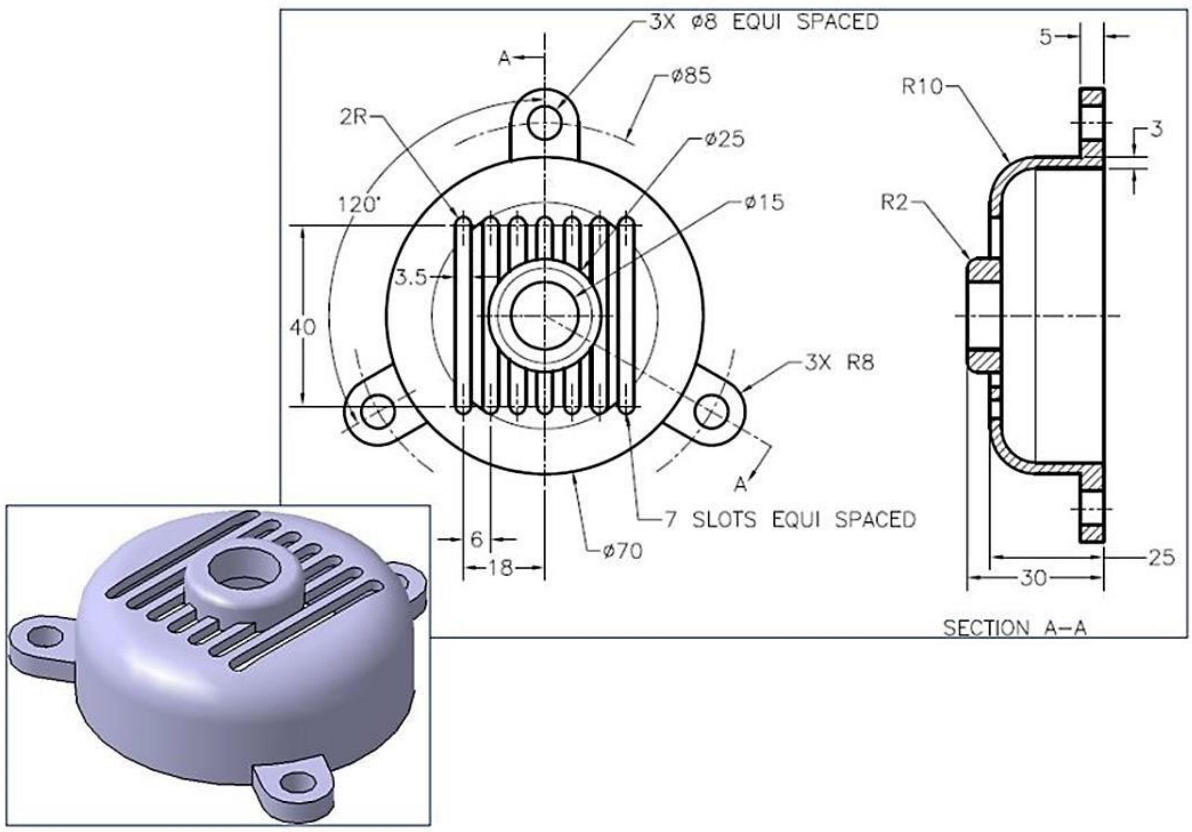


5.



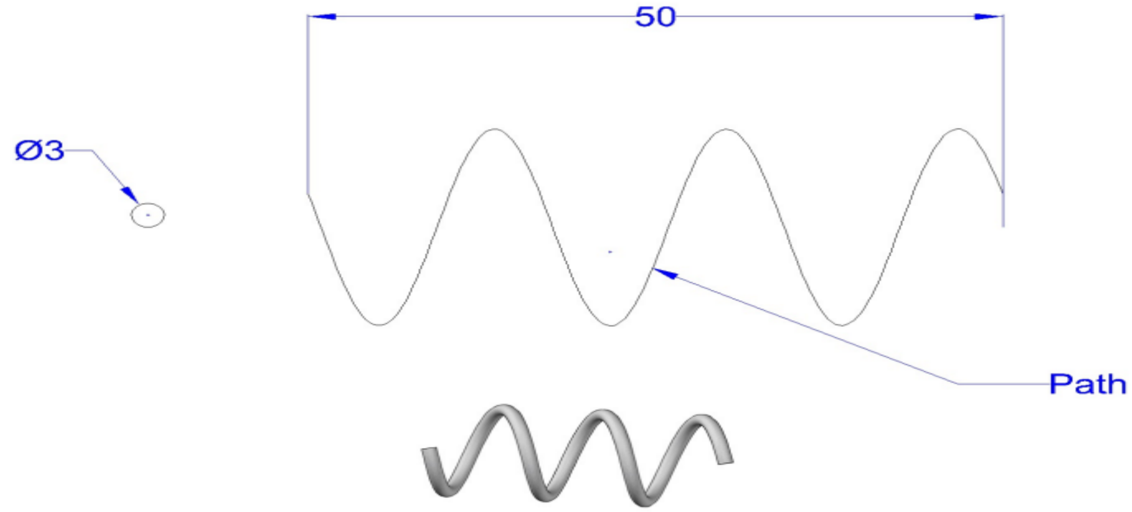
2.5
5
R42.8
15
180
146°
10
20
21.3
5
10
R10
20
15
15
50
100
36.3
15

6.

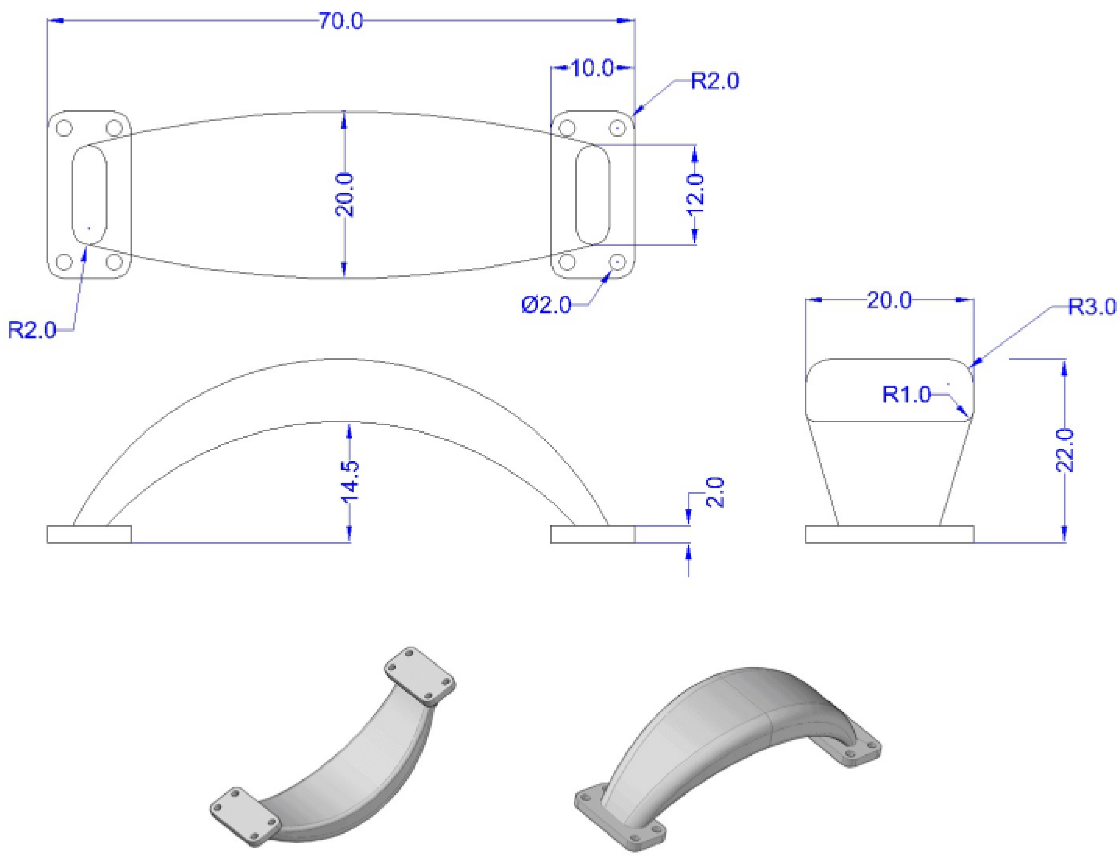


3X Ø8 EQUI SPACED
A
Ø85
2R
Ø25
120°
Ø15
3.5
40
3X R8
A
7 SLOTS EQUI SPACED
6
18
Ø70
5
R10
3
R2
25
30
SECTION A-A

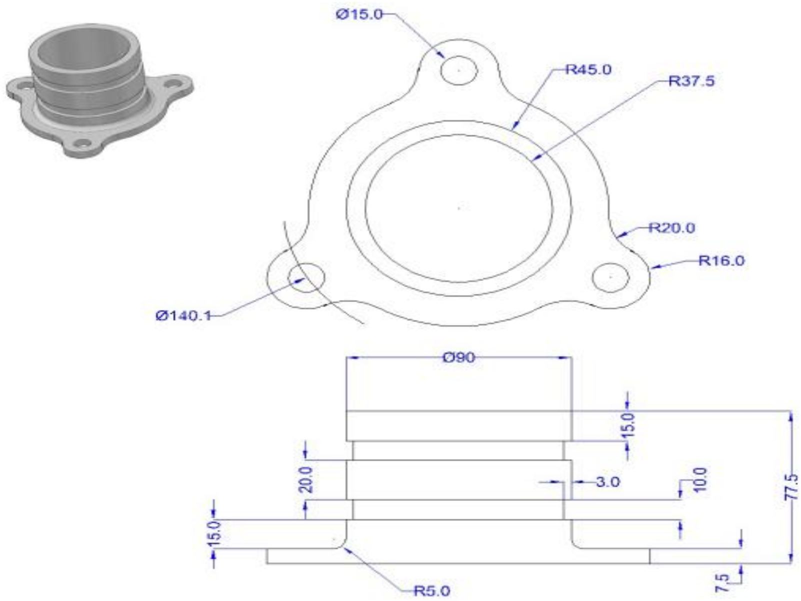
7.



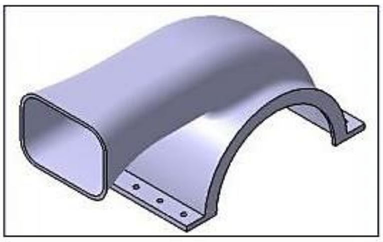
8.

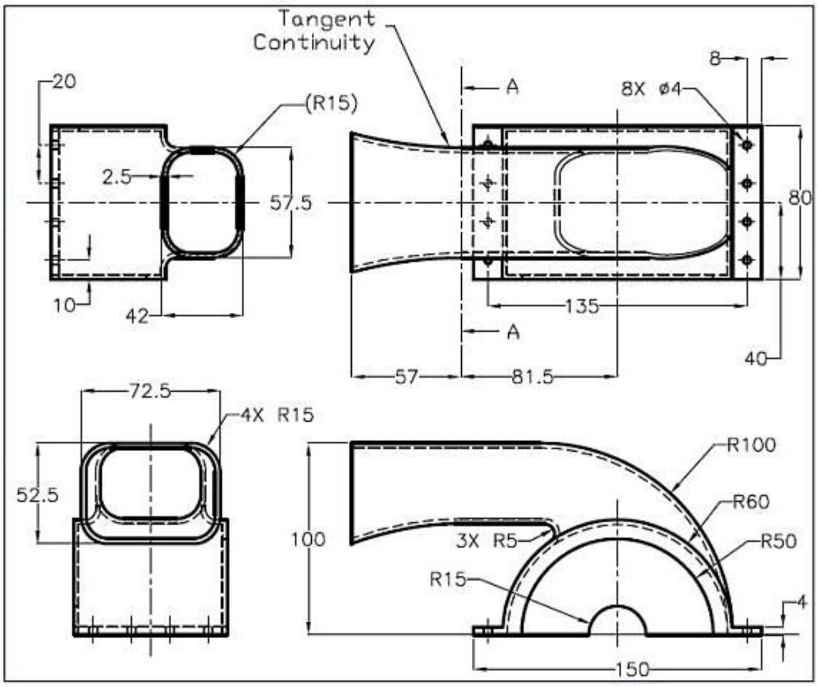


9.


Ø15.0
R45.0
R37.5
R20.0
R16.0
Ø140.1
Ø90
15.0
20.0
3.0
10.0
77.5
15.0
R5.0
7.5

10.

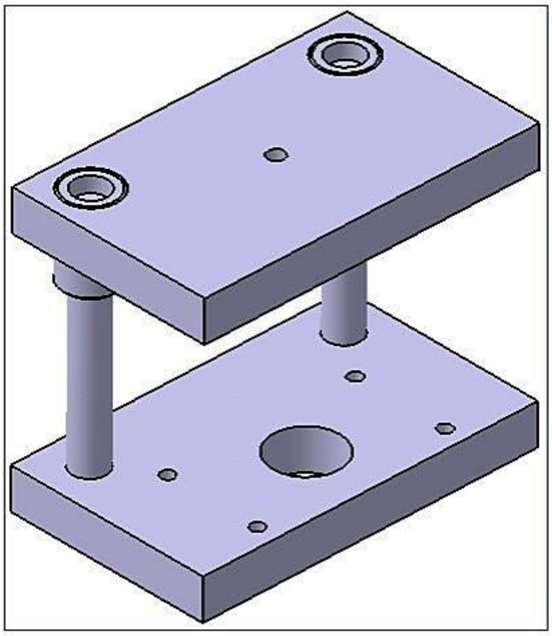


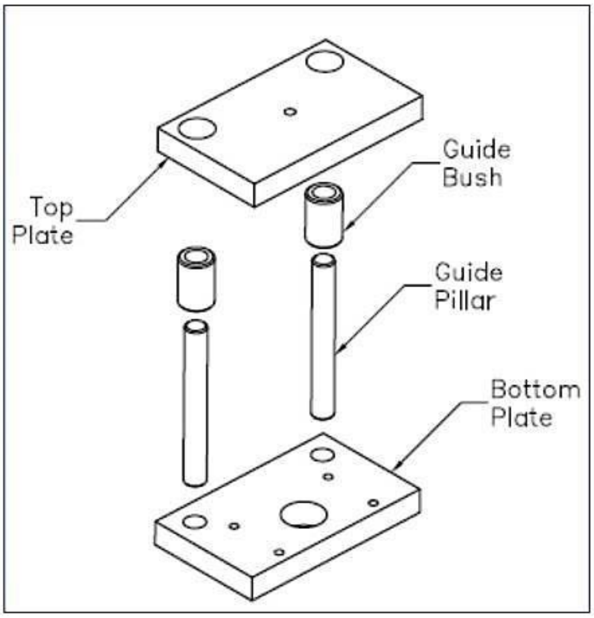

Tangent Continuity
20
(R15)
2.5
57.5
10
42
A
8X ø4
8
80
135
40
57
81.5
72.5
4X R15
52.5
R100
R60
R50
100
3X R5
R15
4
150

ASSEMBLY EXERCISES

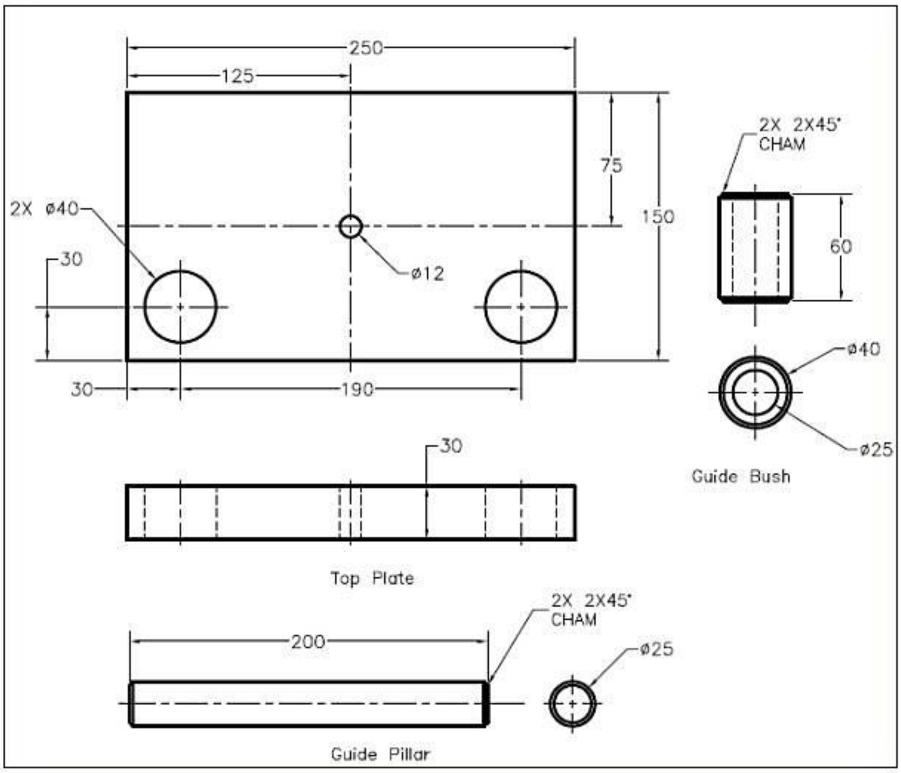
1.

- **Press Tool Base Assembly**

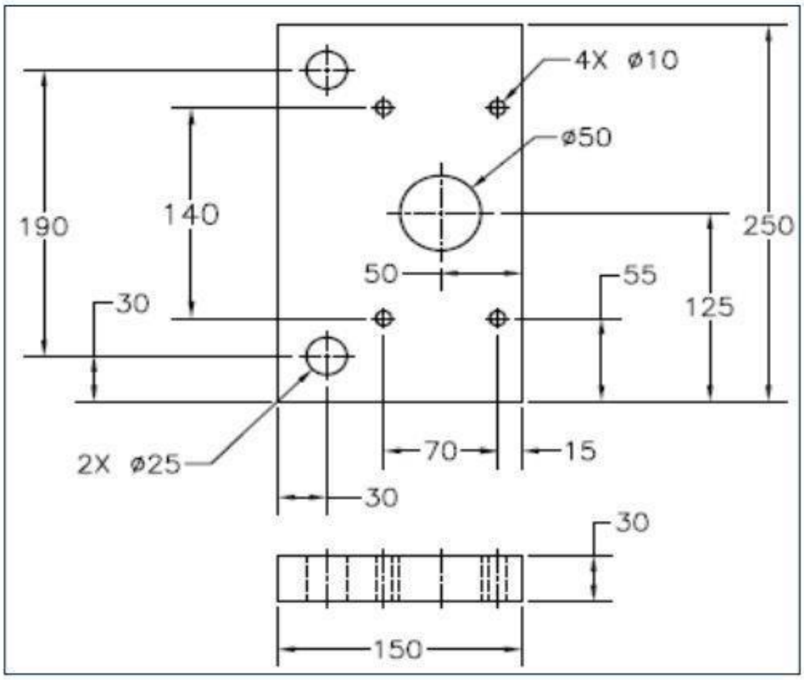




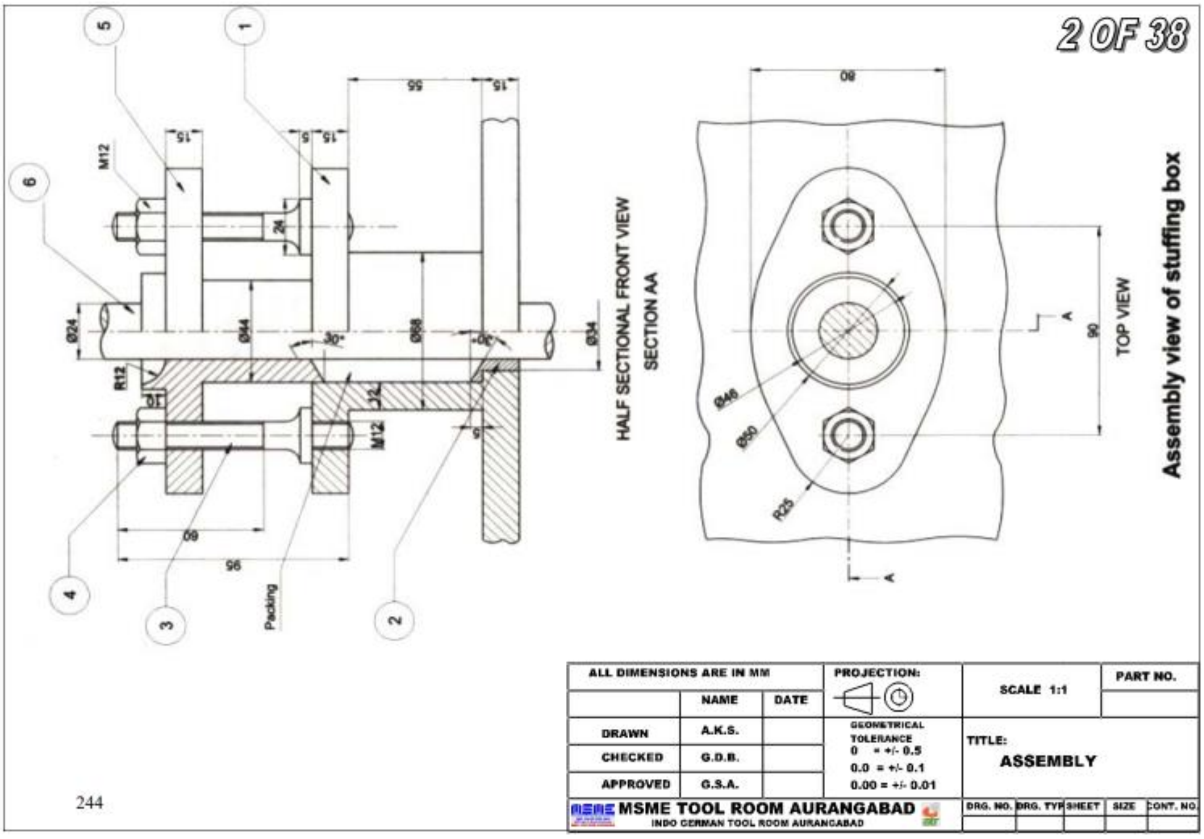
- **Top Plate, Guide Pillar and Guide Bush**

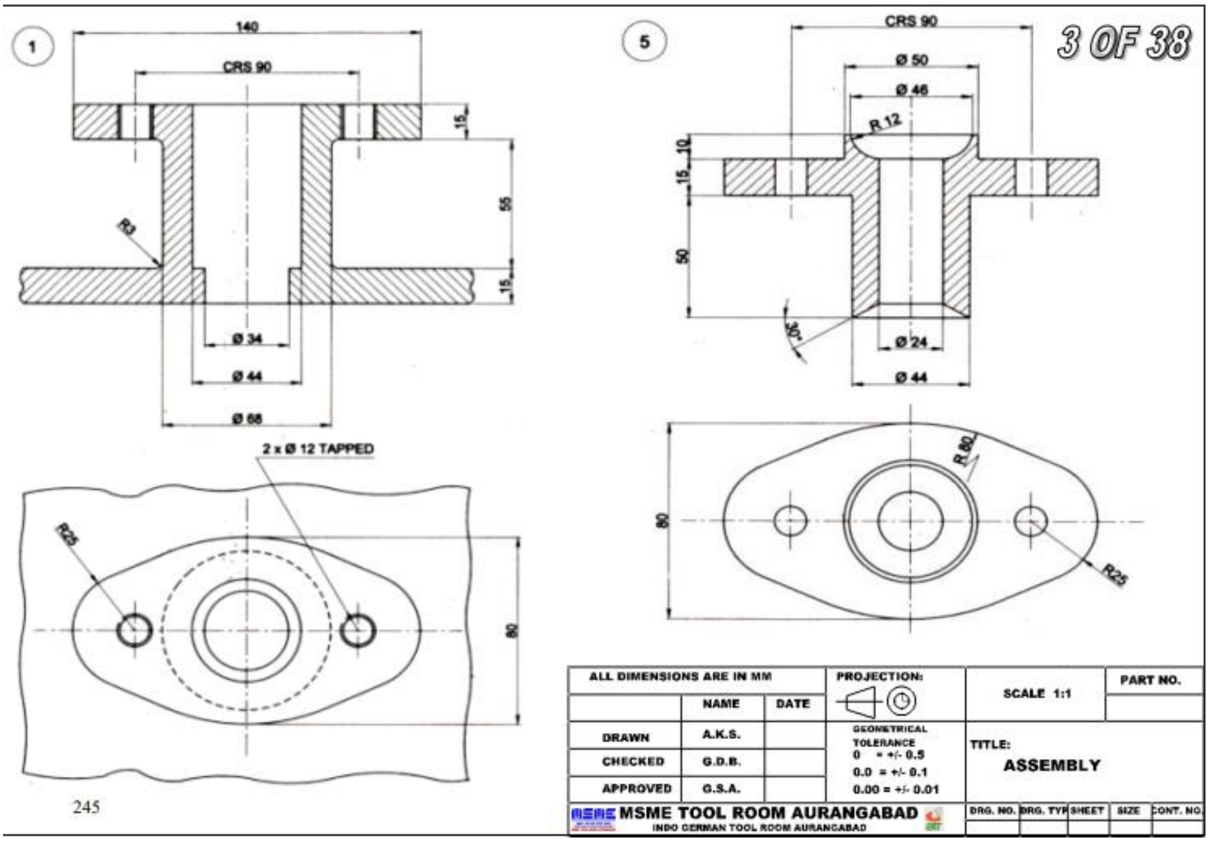


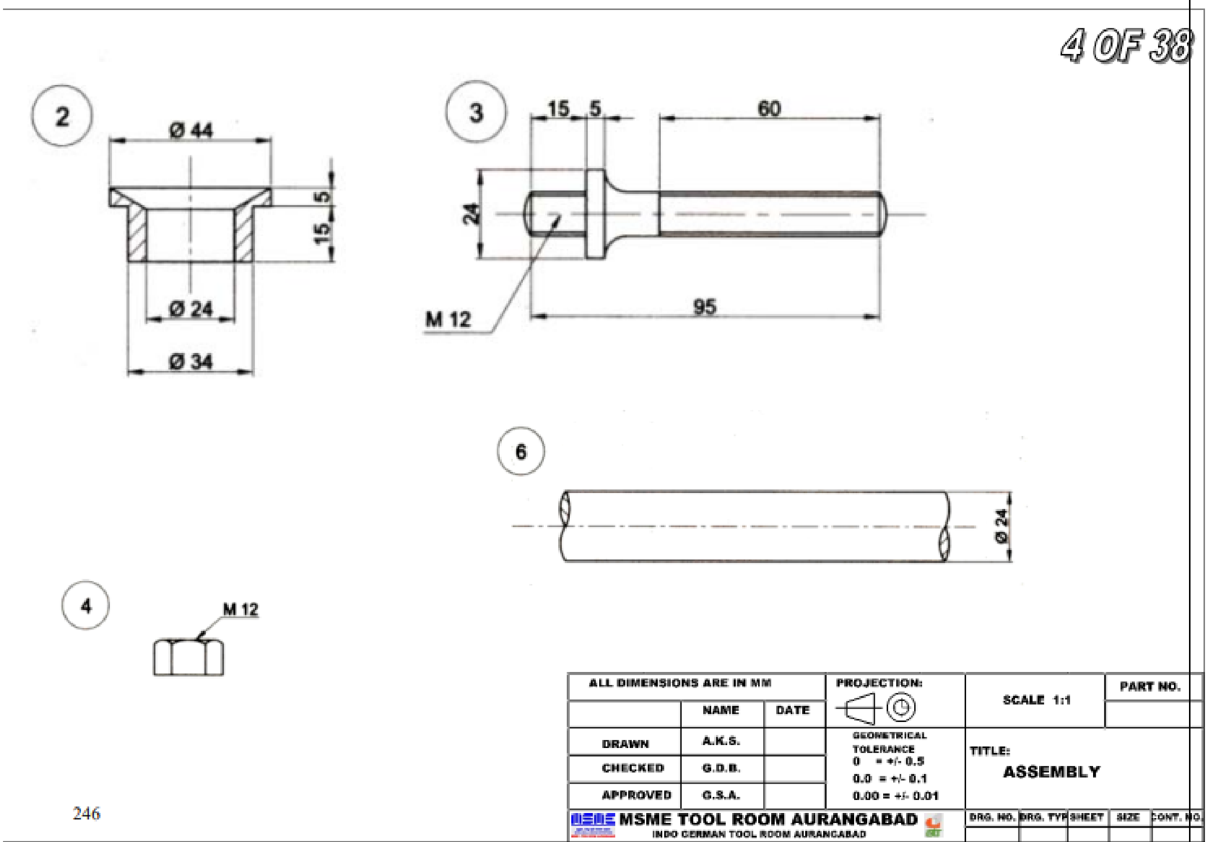
- **Bottom Plate**



2.




3 OF 38
1
140
CRS 90
15
55
15
R3
Ø 34
Ø 44
Ø 68
2 x Ø 12 TAPPED
R25
80
5
CRS 90
Ø 50
Ø 46
R 12
10
15
50
30°
Ø 24
Ø 44
R 60
80
R25
ALL DIMENSIONS ARE IN MM
PROJECTION:
SCALE 1:1
PART NO.
NAME
DATE
DRAWN
A.K.S.
CHECKED
G.D.B.
APPROVED
G.S.A.
GEOMETRICAL TOLERANCE
0 = +/- 0.5
0.0 = +/- 0.1
0.00 = +/- 0.01
TITLE:
ASSEMBLY
MSME TOOL ROOM AURANGABAD
INDO GERMAN TOOL ROOM AURANGABAD
DRG. NO.
DRG. TYP
SHEET
SIZE
CONT. NO.
245


4 OF 38
2
Ø 44
5
15
Ø 24
Ø 34
3
15
5
60
24
M 12
95
6
Ø 24
4
M 12
ALL DIMENSIONS ARE IN MM
PROJECTION:
SCALE 1:1
PART NO.
NAME
DATE
DRAWN
A.K.S.
CHECKED
G.D.B.
APPROVED
G.S.A.
GEOMETRICAL TOLERANCE
0 = +/- 0.5
0.0 = +/- 0.1
0.00 = +/- 0.01
TITLE:
ASSEMBLY
MSME TOOL ROOM AURANGABAD
INDO GERMAN TOOL ROOM AURANGABAD
DRG. NO.
DRG. TYP
SHEET
SIZE
CONT. NO.
246

3.

ASM: 41

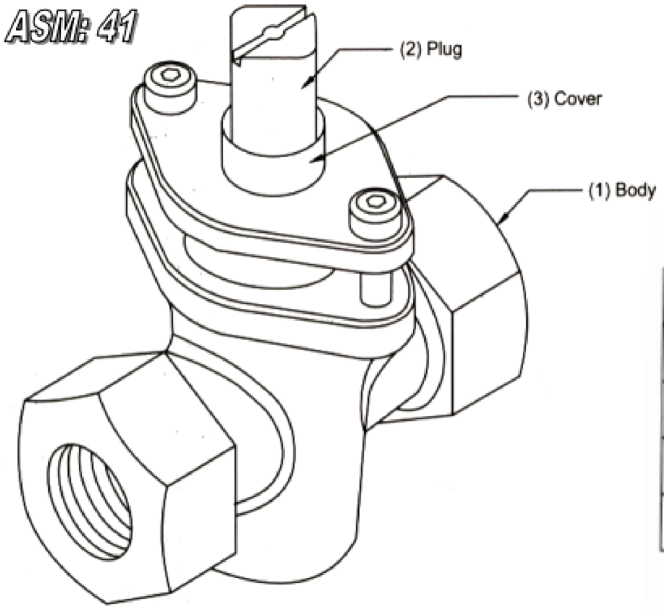


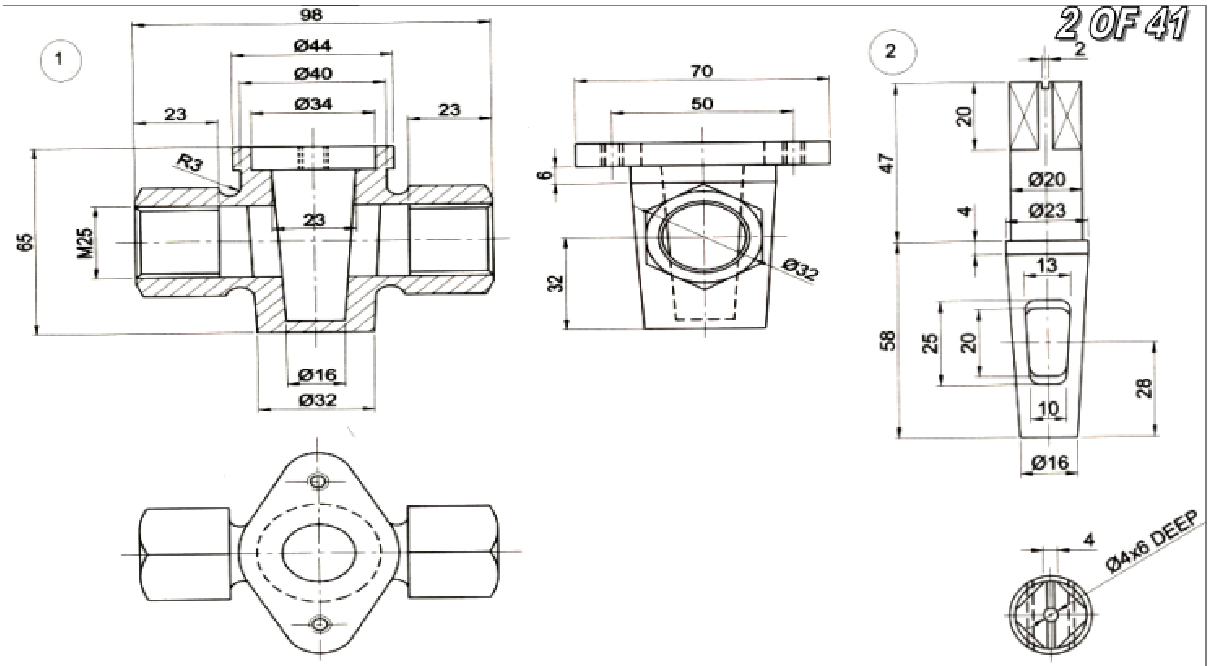
Figure Stop cock

ALL DIMENSIONS IN mm

4	SEAL	RUBBER	1
3	COVER	GUN METAL	1
2	PLUG	GUN METAL	1
1	BODY	GUN METAL	1
ITEM	DESCRIPTION	MATERIAL	NO OFF

Parts details of stop cock

ALL DIMENSIONS ARE IN MM			PROJECTION:	SCALE 1:1	PART NO.
	NAME	DATE			
DRAWN	A.K.S.		GEOMETRICAL TOLERANCE 0 = +/- 0.5 0.0 = +/- 0.1 0.00 = +/- 0.01	TITLE: ASSEMBLY	
CHECKED	G.D.B.				
APPROVED	G.S.A.				
MSME TOOL ROOM AURANGABAD INDO GERMAN TOOL ROOM AURANGABAD				DRG. NO. DRG. TYP SHEET SIZE CONT. NO.	



ALL DIMENSIONS ARE IN MM			PROJECTION:	SCALE 1:1	PART NO.
	NAME	DATE			
DRAWN	A.K.S.		GEOMETRICAL TOLERANCE 0 = +/- 0.5 0.0 = +/- 0.1 0.00 = +/- 0.01	TITLE: ASSEMBLY	
CHECKED	G.D.B.				
APPROVED	G.S.A.				
MSME TOOL ROOM AURANGABAD INDO GERMAN TOOL ROOM AURANGABAD				DRG. NO. DRG. TYP SHEET SIZE CONT. NO.	