CATIA LAB **R20** **II B.Tech-I Sem**

CATIA LAB LABORATORY MANUAL

B.TECH

(II YEAR-I SEM)

(2021-22)

DEPARTMENT OF MECHANICALENGINEERING

SRI VENKATESWARA COLLEGE OFENGINEERING AND TECHNOLOGY (AUTONOMOUS)

SRI VENKATESWARA COLLEGE OF ENGINEERING AND TECHNOLOGY
(AUTONOMOUS)
R.V.S. NAGAR, CHITTOOR-517 127, ANDHRA PRADESH
DEPARTMENT OF MECHANICAL ENGEERING

Vision of Mechanical Engineering

Providing excellent technical education in Mechanical Engineering with the help of state of art infrastructure and carvethe youth to suit the global needs.

Mission of Mechanical Engineering

Provide excellent Teaching-Learning process using state of art facilities to help a holistic growth in the disciplines of Thermal, Design, Manufacturing, Management and Quality areas with an emphasis on practical applications. Stimulate innovative thinkingleading to higher learning.

Programme Educational Objectives (PEO's) of UG:

PEO1	Pursue higher education in the varied fields of mechanical engineering and management.
PEO2	Secure a career placement in core and allied areas
PEO3	Develop skills to undertake entrepreneurship and lifelong learning

PROGRAMME SPECIFIC OUTCOMES (PSOs) of UG

PSO1	Apply the knowledge of manufacturing, thermal and industrial engineering to formulate, analyze and provide solutions to the problems related to mechanical systems
PSO2	Apply the design concepts and modern engineering software tools to model mechanical systems in various fields such as machine elements, thermal, manufacturing, industrial and inter-disciplinary fields.

DO'S

- Wear uniform, shoes & safety glasses
- Please follow instructions precisely as instructed by your supervisor.
- If any part of the equipment fails while being used, report it immediately to your supervisor.
- Students should come with thorough preparation for the experiment to be conducted.
- Students will not be permitted to attend the laboratory unless they bring the practical recordfully completed in all respects pertaining to the experiment conducted in the previous class.
- All the calculations should be made in the observation book. Specimen calculations for oneset of readings have to be shown in the practical record.
- Wherever graphs are to be drawn, A-4 size graphs only should be used and the same shouldbe firmly attached to the practical record.
- Practical record should be neatly maintained.
- Students should obtain the signature of the staff-in-charge in the observation book aftercompleting each experiment.
- Theory regarding each experiment should be written in the practical record before procedurein your own words.

DONT'S

- Do not touch hot work piece
- Do not start the experiment unless your setup is verified & approved by your supervisor.
- Do not leave the experiments unattended while in progress.
- Do not crowd around the equipment's & run inside the laboratory.
- Don't wear rings, watches, bracelets or other jewellery
- Don't wear neck ties or loose turn clothing of any kind.
- Do not eat or drink inside labs.
- Do not wander around the lab and distract other students
- Do not use any machine that smokes, sparks, or appears defective

LIST OF EXPERIMENTS

S.No.	Experiment Name
1	**Solid Model - Experiment 1**
2	**Solid Model - Experiment 2**
3	**Solid Model - Experiment 3**
4	**Solid Model - Experiment 4**
5	**Solid Model - Experiment 5**
6	**Solid Model - Experiment 6**
7	**Self Learning - Experiment 7**
8	**Self Learning - Experiment 8**
9	**Self Learning - Experiment 9**
10	**Self Learning - Experiment 10**

EXPERIMENT 1

In this tutorial, you will draw the sketch of the model shown in Figure 1-31. The sketch is shown in Figure 1-32. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

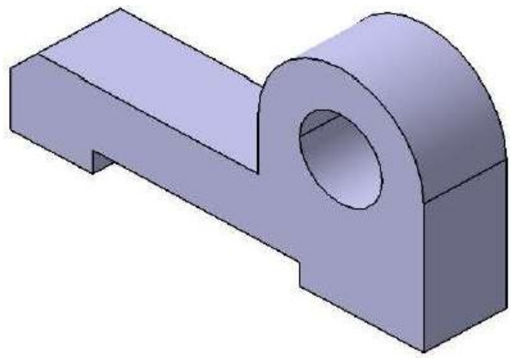


Figure 1-31 *Solid Model for Tutorial 1*

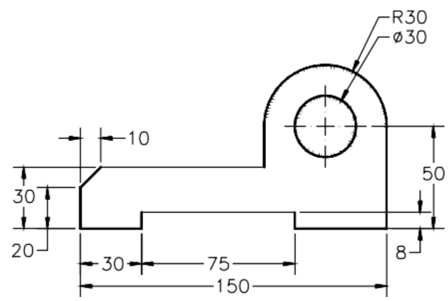


Figure 1-32 *Sketch of the model*

The following steps are required to complete this tutorial:

a. Start CATIA V5 and then start a new CATpart file.
b. Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, refer to Figures 1-35 and 1-36.
c. Save and close the file.

Starting CATIA V5 and Opening a New Part File

1. Start CATIA V5 by choosing **Start > Programs (All Programs** if you are working with Windows XP **> CATIA > CATIA V5R13** or by double-clicking on the shortcut icon of CATIA V5R13 available on the desktop of your computer.

   A new **Product1** file is started.

2. On choosing **Close** from **File** menu, the start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Part Design** to make sure that you are in **Part Design** workbench. To open a new file in **Part Design** workbench, choose **File > New** from menu bar. The **New** dialog box is displayed, as shown in Figure 1-33.

3. Select **Part** from the **List of Types** list box from this dialog box and choose the **OK** button.

   A new file in the **Part Design** workbench is opened.

4. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane, to invoke the **Sketcher** workbench. The screen that is displayed by invoking the **Sketcher** workbench, is shown in Figure 1-34. 

You will draw the sketch in two sections, first as the outer loop and second as the inside circle.

Drawing the Outer Loop of the Sketch

It is recommended to create the sketch symmetrically around the origin. This will reduce the time required for constraining and dimensioning the sketches. The outer loop of sketch can be drawn using the **Line** and the **Arc** tools. You will start drawing the outer loop from the lower left corner of the sketch.

1. Invoke the **Line** tool by choosing the **Line** button from the **Profile** toolbar. 
2. Choose the **Snap to Point** button from the Sketch tools toolbar, if not already chosen.
3. Move the cursor in the third quadrant. The coordinates of the point will be displayed along with the cursor.
4. Click at the point whose coordinates are -50mm, -30mm, and then move the cursor horizontally toward the right.

   You will notice that the color of line turns blue, when you move the cursor horizontally.

   Refer to Figure 1-32. The length of the first horizontal line at the lower left corner of the sketch is 30mm. Therefore, move the cursor until the length of the line is shown as 30mm in the **L** edit box of the **Sketch tools** toolbar.

5. Press the left mouse button, when the length of the line in the **L** edit box of the **Sketch tools** toolbar displays a value of 30mm.

   The first horizontal line is drawn. You will notice a **Horizontal** constraint is applied on it. After the line is drawn, it is still active and is displayed in orange color. Left click in the geometry area to make sure it is no more selected.
   first horizontal line.

   As soon as you specify the endpoint of line, the **Line** tool is terminated. Therefore, you need to choose this button again and again to draw multiple lines. You can avoid this by double-clicking on the **Line** button in the **Profile** toolbar. Now, the line tool will not be terminated until you terminate it by pressing ESC key twice on the keyboard.

6. Double-click on the **Line** button to invoke the **Line** tool and select the endpoint of the

7. Press the TAB key thrice on the keyboard to highlight the value displayed in the **L** edit

box of the **Sketch tools** toolbar. Type **8** in this edit box and press the ENTER key.

8. Now move the cursor vertically upward and click when a vertical line is displayed.

   A vertical line of length 8mm will be drawn. You will notice that this line is no longer in the select mode and you are prompted to select the start point of the next line. This is because of double-clicking on the **Line** button. It makes the **Line** tool active till you invoke any other tool.

9. Select the endpoint of the vertical line as the start point of the second horizontal line. Enter **75** in the **L** edit box of the **Sketch tools** toolbar. Move the cursor horizontally toward the right and click when a horizontal line is displayed.

   This draws the second horizontal line of length 75mm.

10. Select the endpoint of the second horizontal line, as the start point of the second vertical line and move the cursor vertically downward. Click when the length of the line in the **L** edit box shows a value of 8mm.

    This draws the second vertical line of length 8mm.

11. Select the endpoint of the second vertical line as the start point of the third horizontal line and move the cursor horizontally toward the right. Click to draw the third horizontal line, when the length of the line in the **L** edit box shows a value of 45mm.

12. Select the endpoint of the previous line as the start point of the third vertical line and move the cursor vertically upwards. Click when the length of the line is 50mm.

    This draws the third vertical line of length 50mm. Next, you can draw the arc.

13. To draw the arc, first invoke the **Circle** toolbar, by choosing the down arrow available on the right of the **Circle** button, from the **Profile** toolbar. Choose the **Three Point Arc** button from to invoke the **Three Point Arc** tool.

14. Select the start point of the arc as the endpoint of the previous vertical line and click on it.

15. Move the cursor to a point whose coordinates are 70mm, 50mm. These are displayed in the **Sketch tools** toolbar and also on top of the cursor. Click on this point to define the second point.

16. Move the cursor to specify the third point of the arc. Click on the point when the cursor snaps a location 40mm, 20mm in the geometry area. The coordinate values are displayed on top of the cursor.

This draws the arc for the outer loop. As the arc is in selection mode, click anywhere in the geometry area to end the selection mode. Now, to continue drawing the outer loop, you need to invoke the **Line** tool again.

17. Double-click on the **Line** button from the **Profile** toolbar to invoke the **Line** tool.

18. Select the endpoint of the arc as the start point of the fourth vertical line. Move the cursor vertically downward to draw it. Click when the length value of the line is 20mm in the **L** edit box of the **Sketch tools** toolbar.

    This draws the fourth vertical line of length 20mm. The line is no longer in selection mode and you are prompted to enter the start point of the next line.

19. Select the endpoint of the previous line as the start point of the fourth horizontal line. Move the cursor horizontally toward left. Click when the length of the line in the **L** edit box of the **Sketch tools** toolbar shows a value of 80mm.

    This draws the fourth horizontal line of length 80mm. Note that the line is green in color, as it passes through the origin.

20. Select the endpoint of the previous line as the start point of the inclined line. Move the cursor such that the line is drawn at an angle of 225-degree. The current angle will be displayed in the **A** edit box of the **Sketch tools** toolbar. Click when a vertical inferencing line is displayed between the endpoint of the inclined line and the start point of the first horizontal line. This draws the inclined line of horizontal length values 10mm.

21. Select the endpoint of the inclined line as the start point of the next line. Move the cursor vertically downwards. Click when the length of the line in the **L** edit box shows a value of 20mm.

    This completes the sketch of the outer loop. It is recommended to modify the geometry area, such that the sketch fits in the screen. This is done using the **Fit All In** tool.

22. Choose the **Fit All In** button from the **View** toolbar to fit the current sketch on the screen.

    The outer loop of the sketch is completed and is shown in Figure 1-35. The display of the constraints is turned off using the **Hide/Show** tool.

Drawing Inner Circle

The circle will be drawn using the **Circle** tool.

1. Choose the **Circle** button from the **Circle** toolbar to invoke the **Circle** tool. You are prompted to define the center point of the circle.

2. Move the cursor to a point whose coordinates are 70mm, 20mm. Click when the cursor snaps to this point.

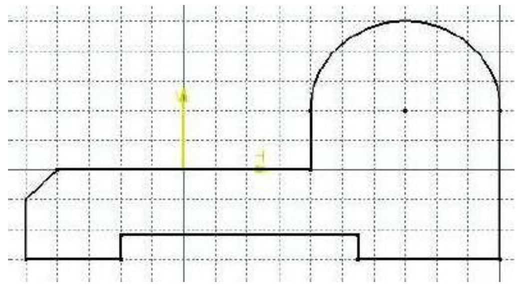


***Figure 1-35** Outer loop of the sketch*

3. Move the cursor horizontally toward the right and click when the radius of the circle in the **R** edit box of the **Sketch tools** toolbar shows a value of 15mm. Click anywhere to remove the circle from selection.

   This completes the sketch for Tutorial 1. The final completed sketch for Tutorial 1 with the display of constraints turned on is shown in Figure 1-36.

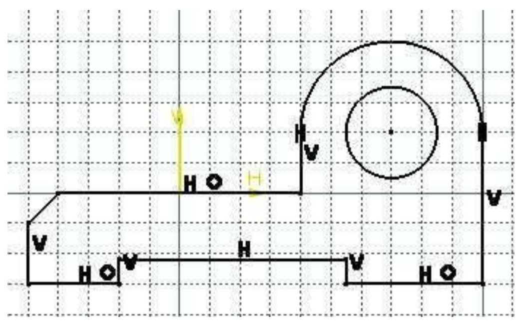


***Figure 1-36** Final sketch for Tutorial 1*

EXPERIMENT 2

In this tutorial, you will draw the sketch of the model shown in Figure 1-37. The sketch is shown in Figure 1-38. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

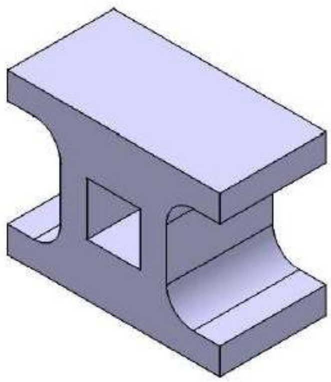


Figure 1-37 Solid Model for Tutorial 2

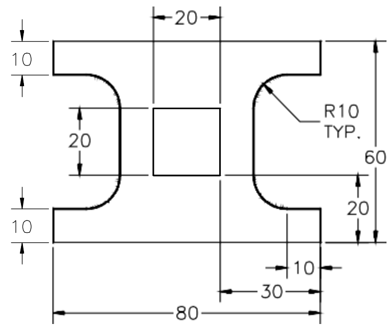


Figure 1-38 Sketch of the model

The following steps are required to complete this tutorial:

a. Start a new CATpart file.
b. Draw the sketch of the model using the **Profile** and **Rectangle** tool, refer to Figures 1-39 through 1-41.
c. Save and close the file.

Starting New Part File

1. Choose **File > New** from the menu bar. The **New** dialog box is displayed.

2. Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button. A new file in **Part Design** workbench will open.

3. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane, to invoke the **Sketcher** workbench.

You will draw the sketch in two sections, first the outer loop and next the inner cavity.

Drawing the Outer Loop of the Sketch

You will draw the outer loop of the sketch using the **Line** and the **Arc** tool. Start drawing the outer loop from the left corner of the sketch. It is recommended to keep the origin in middle of the drawn sketch, as this will reduce the time required for constraining and dimensioning the sketches. This will also helps you to capture the design intent very easily.

1. Invoke the **Profile** tool from the **Profile** toolbar.

2. Move the cursor in the third quadrant. The coordinates of the point will be displayed above the cursor.

3. Specify the start point of the line at the point whose coordinates are -40, -30 and then move the cursor horizontally toward the right.

   You will notice that the color of line turns blue, when you move the cursor horizontally.

4. Move the cursor to a location whose coordinates are 40, -30. The coordinates of the point can be seen on top of the cursor.

5. Specify the endpoint of the line at this location. A rubber band line is attached to the cursor. Move the cursor vertically upward.

6. Specify the endpoint of the second line on the point whose coordinates are 40mm, -20mm.

   A rubber band line is attached to the cursor.

7. Move the cursor horizontally toward the left and specify the endpoint of the third line where the value of the coordinates is 30, -20.

   After drawing these three lines, draw a tangent arc using the **Tangent Arc** option available in the **Profile** tool.

8. Choose the **Tangent Arc** button available in the **Sketch tools** toolbar.

9. Move the cursor to a location whose coordinates are 20, -10 and specify the endpoint of the tangent arc. Figure 1-39 shows the sketch, after drawing three lines and the tangent arc. The system switches back to the **Line** mode.

10. Move the cursor vertically upward to a location whose coordinates are 20, 10.

11. Specify the endpoint of the line at this location.

Next, you need to draw a tangent arc by switching to the arc mode using the **Tangent Arc** option available in the **Profile** tool.

12. Choose the **Tangent Arc** button from the **Sketch tools** toolbar.

13. Move the cursor to a location whose coordinates are 30, 20 and specify the endpoint of the tangent arc.

    The system switches back to the **Line** mode.

14. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is 40, 20.

15. Move the cursor vertically upward and specify the endpoint of the line, when the value of the coordinates is 40, 30.

16. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is -40, 30.

17. Move the cursor vertically downward and specify the endpoint of the line, when the value of the coordinates is -40, 20.

18. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is -30, 20.

    Next, you need to draw a tangent arc by switching to the tangent arc mode.

19. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch the tangent arc mode.

20. Move the cursor to a location whose coordinates are -20, 10 and specify the endpoint of arc at this location.

    The system switches back to the Line mode.

21. Move the cursor vertically downward and specify the endpoint of the line, where the value of the coordinates is -20, -10.

22. Switch to the Tangent mode and move the cursor to a location whose coordinates are -30mm, -20mm. Specify the endpoints of the tangent arc at this location.

23. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of coordinates is -40, -20.

24. Move the cursor vertically downward and specify the endpoint of the line when it snaps the start point of the outer loop. The sketch after completing the outer loop of the sketch, after hiding the constraints, is shown in Figure 1-40.

Drawing the Inner Cavity of the Sketch

After drawing the outer loop of the sketch, you need to draw its inner rectangular cavity. You will use the **Rectangle** tool to draw the inner cavity.

1. Choose the **Rectangle** tool from the **Profile** toolbar. 

2. Move the cursor to a location whose coordinates are -10, 10. Specify the upper left corner of the rectangle at this location.

3. Move the cursor to a location whose coordinates are 10, -10. Specify the lower-right corner of the rectangle at this location.

4. Choose the Fit All in button from the View toolbar to fit the sketch in the geometry area.

The final sketch, after drawing the inner loop, is shown in Figure 1-41. Note that the display of constraints has been turned on in this figure.

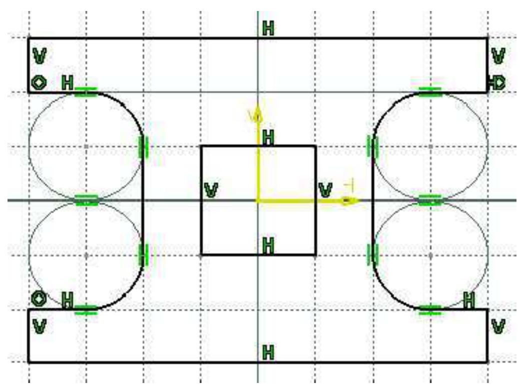


Figure 1-41 *Final sketch after drawing inner loop of the sketch*

Saving the Sketch

After completing the sketch you need to save it. As mentioned earlier, you need to save each tutorial of this chapter in the *c01* folder in the *CATIA* folder.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse for the folder named *c01* that you created in the last tutorial.

2. Enter the name of the file as *c01tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File** > **Close** from the menu bar.

EXPERIMENT 3

In this tutorial, you will draw the sketch of the model shown in Figure 1-42. The sketch is shown in Figure 1-43. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

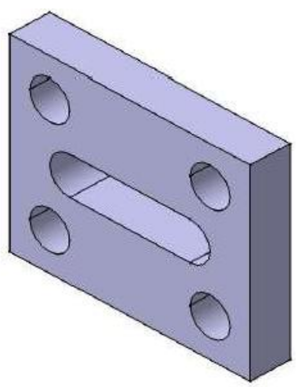


Figure 1-42 Solid model for Tutorial 3

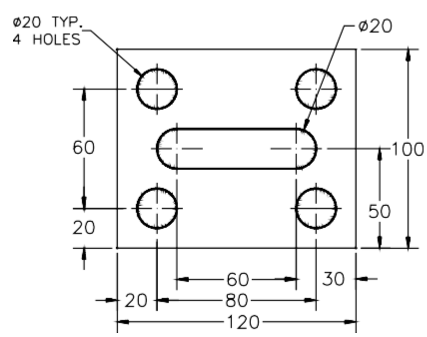


Figure 1-43 Sketch for the solid model

The following steps are required to complete this tutorial:

a. Start a new CATpart file.
b. Draw the sketch of the model using the **Rectangle**, **Profile**, and the **Circle** tools, refer to Figures 1-44 through 1-46.
c. Save the sketch and close the file.

Starting New Part File

1. Choose **File > New** from the menu bar; the **New** dialog box is displayed.

2. Select **Part** from the **List of Types** list box in this dialog box. Choose the **OK** button. A new file in the **Part Design** workbench will be opened.

3. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane.

This sketch will be drawn in two parts. Initially, you will draw the outer loop of the sketch, that is, a rectangle. Next, you need to draw the inner loops of the sketch, which consists of multiple inner loops that are four holes and an elongated hole. First you will draw an elongated hole using the **Profile** tool and then the four holes using the **Circle** tool.

Drawing the Outer Loop of the Sketch

The outer loop of the sketch will be drawn using the **Rectangle** tool.

1. Choose the **Rectangle** button from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are -60, -50 and specify the lower left corner of the rectangle.

3. Move the cursor to the location whose coordinates are 60, 50 and specify the upper right corner of the rectangle. Figure 1-44 shows the outer loop of the sketch drawn using the **Rectangle** tool.

Drawing the Inner Loop of the Sketch

After drawing the outer loop of the sketch, draw its inner loop.

1. Choose the **Profile** button from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are -30, 10 and specify the start point of the line.

3. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is 30, 10.

Next, you need to draw a tangent arc by switching over to the **Tangent Arc** option using the **Sketch tools** toolbar.

4. Choose the **Tangent Arc** button from **Sketch tools** toolbar to switch over to the arc mode.

5. Move the cursor to a location whose coordinates are 30, -10 and specify the endpoint of the tangent arc.

   The system switches over to the **Line** mode.

6. Move the cursor to the location whose coordinates are -30, -10 and specify the endpoint of the line.

7. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch over to the arc mode.

8. Move the cursor to the start point of the first horizontal line of the elongated hole. Specify the endpoint of the arc when it snaps the start point.

   The sketch, after drawing the elongated hole, is shown in Figure 1-45.

9. Choose the **Circle** button from the **Circle** toolbar.

10. Move the cursor to a location whose coordinates are 40, 30 and specify the center point of the circle.

11. Specify the value of **10** as the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

    You will observe that a radius dimension is displayed attached to circle because you have specified the value of the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

Figure 1-45 *Sketch after drawing the elongated hole*

12. Choose the **Circle** button from the **Circle** toolbar.

13. Move the cursor to a location whose coordinates are 40, -30 and specify the center point of the circle.

14. Specify the value of 10 as the radius in the **Radius** edit box available in the **Sketch tools** toolbar.

15. Similarly, draw the other two circles. The coordinates of the center point of other two circles are -40, 30 and -40, -30 respectively. The final sketch, with the display of constraints turned on, is shown in Figure 1-46.

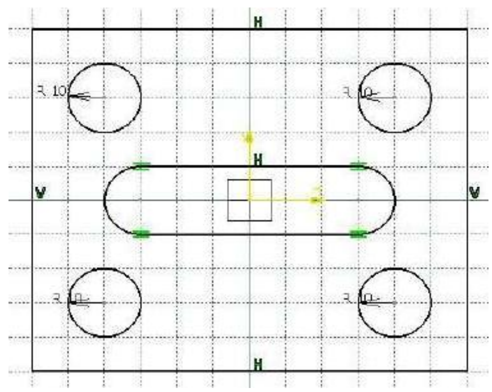


Figure 1-46 *Final sketch*

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse for the folder named *c01* that you created in the first tutorial.

2. Enter the name of the file as *c01tut3* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File** > **Close** from the menu bar.

EXPERIMENT 4

In this tutorial, you will draw the sketch of the model shown in Figure 1-47. The sketch is shown in Figure 1-48. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Start a new CATpart file.
b. Draw the sketch of the model using the **Profile** and the **Circle** tool, refer to Figures 1-49 and 1-50.
c. Save the sketch and close the file.

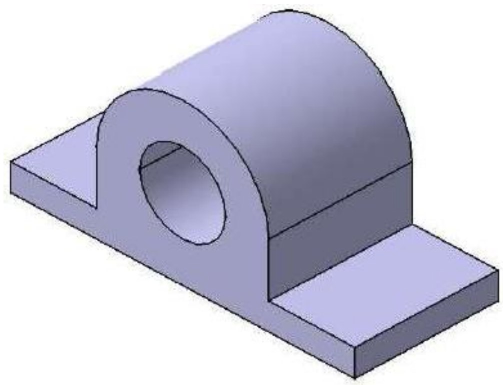


Figure 1-47 *Solid model for Tutorial 4*

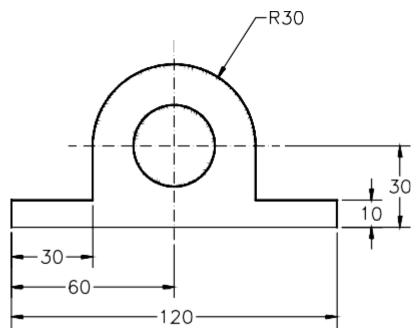


Figure 1-48 *Sketch for the solid model*

Starting a New Part File

1. Choose **New** from **File** menu; the **New** dialog box is displayed.

2. Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button. A new file **Part Design** workbench is opened.

3. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane to invoke the **Sketcher** workbench.

This sketch will be drawn in two parts. Initially, you will draw the outer loop of the sketch using the **Profile** tool and then the inner loop of the sketch, which is a hole.

Drawing the Outer Loop of the Sketch

The outer loop of the sketch will be drawn using the **Profile** tool. In this sketch the lower left corner of the sketch is coincident to the origin of the **Sketcher** workbench. The resulting sketch will be drawn in the first quadrant.

1. Choose the **Profile** button from the **Profile** toolbar.

2. Move the cursor to a location whose coordinates are 0, 0 and specify the start point of the line.

3. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of coordinates is 120, 0.

4. Move the cursor vertically upward and specify the endpoint of the line, when the value of coordinates is 120, 10.

5. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is 90, 10.

6. Move the cursor vertically upward and specify the endpoint of the line, when the value of the coordinates is 90, 30.

After drawing these four lines, the next element that you need to draw is a tangent arc. You need to use the **Tangent Arc** option from the **Sketch tool** toolbar to draw it.

7. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch to the **Tangent Arc** mode.

8. Move the cursor to a location whose coordinates are 30, 30 and specify the endpoint of the tangent arc at this location.

   The system switches back to the **Line** mode.

9. Move the cursor vertically downward and specify the endpoint of the line, when the value of the coordinates is 30, 10.

10. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is 0, 10.

11. Move the cursor vertically downward and specify the endpoint of the line such that the endpoint is coincident with the start point of the first line.

    The sketch, after drawing the outer loop, is shown in Figure 1-49.

Drawing the Inner Loop of the Sketch

The inner loop of the sketch consists of a circle that will be drawn, using the **Circle** tool, concentric to the arc of the outer loop.

1. Choose the **Circle** button from the **Circle** or **Profile** toolbar.

2. Move the cursor to the center point of the circular arc and specify the center point of the circle.

3. Specify the value of **15,** as the radius of circle in the **Radius** edit box provided in the **Sketch tools** toolbar.

The final sketch, after drawing the inner loop is shown in Figure 1-50. Note that the display of constraints is turned on in this figure.

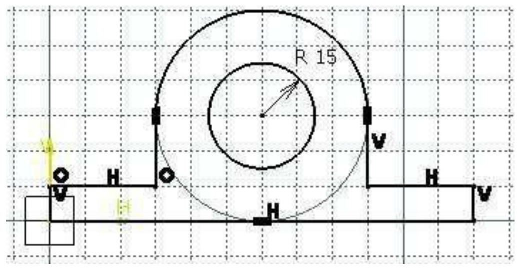


Figure 1-50 *Final Sketch*

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box and browse for the *c01* folder. 

2. Enter the name of the file as *c01tut4* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.

3. Close the part file by choosing **File** > **Close** from the menu bar.

EXPERIMENT 5

In this tutorial, you will create the model of the nozzle of a vacuum cleaner shown in Figure 5-70. The views and dimensions of this model are shown in Figure 5-71. (**Expected time: 45 min**)

The following steps are required to complete this tutorial:

a. Start a new file in the **Part** workbench and create the base feature of the model by extruding the sketch along the selected direction, refer to Figures 5-72 through 5-76.
b. Create the second feature of the model by extruding a sketch using the **Drafted Fillet Pad** tool, refer to Figures 5-77 and 5-78.
c. Create the third feature of the model, which is a cut feature. It will be used to remove the unwanted portion of the second feature, refer to Figures 5-79 and 5-80.
d. Apply fillets to all edges of the model, refer to Figures 5-81 through 5-84.
e. Shell the model using the **Shell** tool, refer to Figures 5-85 and 5-86.

Figure 5-70 *Model of Vacuum Cleaner for Tutorial 1*

The base feature of this model is created by first creating a plane at an angle of 26-degree and then extruding a sketch drawn on that plane. The sketch will be extruded along a selected direction. In this model, you will learn a technique to create the reference sketch first and then follow it to create the model. Therefore, you will first draw the reference sketch.

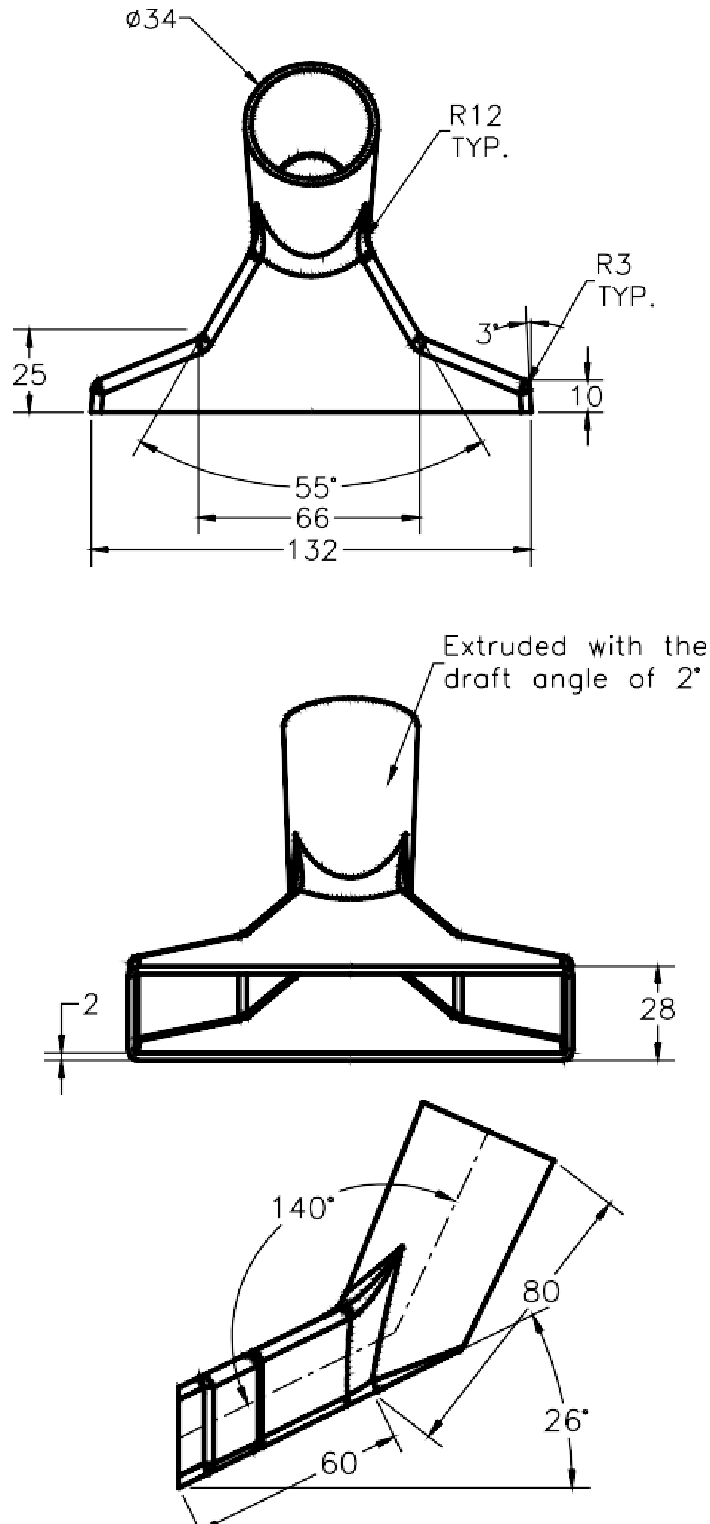


Figure 5-71 *Views and dimensions of Vacuum Cleaner for Tutorial 1*

Creating the Base Feature of the Model

1. Start a new file in the **Part** workbench. Select the ZX plane and invoke the **Sketcher** workbench.

2. Draw the sketch, as shown in Figure 5-72, and then exit the **Sketcher** workbench.

3. Select the YZ plane and invoke the **Sketcher** workbench. Place a point colinear to the X-axis at any distance, as shown in Figure 5-73. Exit the **Sketcher** workbench.

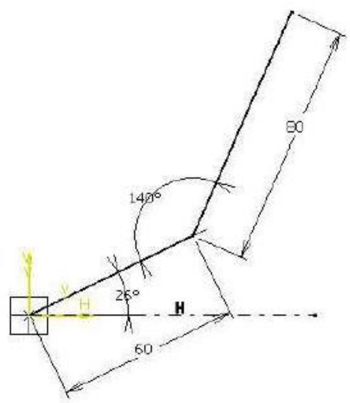


Figure 5-72 *Reference sketch*

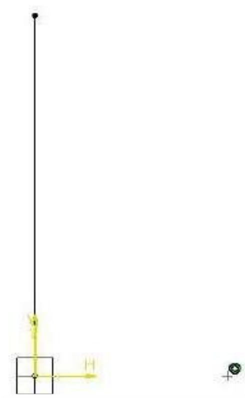


Figure 5-73 *Point to be placed*

After drawing the reference sketch and placing the point, you need to create a plane that will be used as the reference plane to create the base feature.

4. Create a plane by selecting three points, as shown in Figure 5-74.

5. Invoke the **Sketcher** workbench after selecting the newly created plane as the sketching plane and draw the sketch, as shown in Figure 5-75.

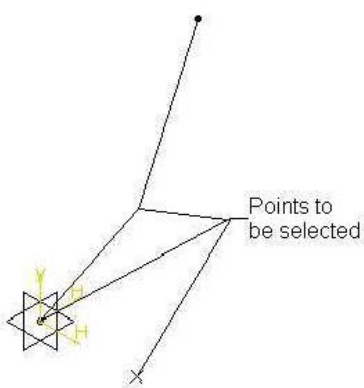


Figure 5-74 *Points to be selected to create plane*

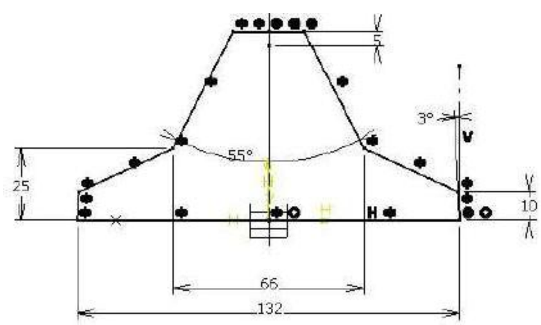


Figure 5-75 *Sketch of the base feature*

6. Exit the **Sketcher** workbench. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed.

7. Set the value of the **Length** spinner to **28**. The preview of the extruded feature is displayed in the geometry area. If the sketch is extruded in the downward direction, then choose the **Reverse Direction** button to flip the direction of feature creation.

8. Now, choose the **More** button to expand the **Pad Definition** dialog box.

9. Clear the **Normal to profile** check box provided in the **Direction** area and select the XY plane as the direction of extrusion.

10. Choose the **OK** button from the **Pad Definition** dialog box to complete the feature creation. The model, after creating the base feature, is shown in Figure 5-76.

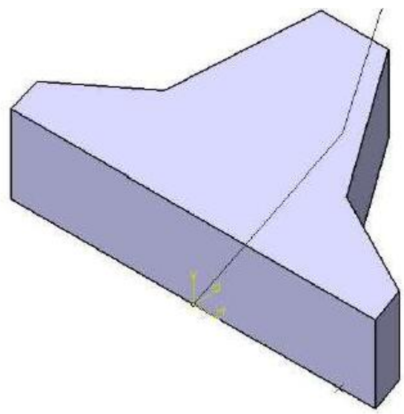


Figure 5-76 *Model after creating the base feature*

Creating the Second Feature

The second feature of this model is a drafted extrude feature created using the **Drafted Filleted Pad** tool. In this feature, you will extrude the sketch drawn on a plane created normal to the right line of the reference sketch.

1. Invoke the **Plane** tool and select the **Normal to curve** option from the **Plane type** drop-down list.

2. Now, select the right line of the reference sketch as the curve and then select the upper endpoint of the same line as the point on which the plane will be created. The preview of the plane is displayed in the geometry area.

3. Choose the **OK** button from the **Plane Definition** dialog box.

4. Use the newly created plane to invoke the **Sketcher** workbench and draw the sketch, as shown in Figure 5-77.

5. Exit the **Sketcher** workbench and invoke the **Drafted Filleted Pad** tool.

6. Set the value of the **Length** spinner to **85** and select the newly created plane from the geometry area as the second limit.

7. Set the value of the draft angle in the **Angle** spinner to **2deg**. Choose the **Reverse Direction** button to flip the direction of feature creation.

8. Clear all the radio buttons available in the **Fillets** area and choose the **OK** button from the **Drafted Fillet Pad Definition** dialog box. The model, after creating the second feature, is shown in Figure 5-78.

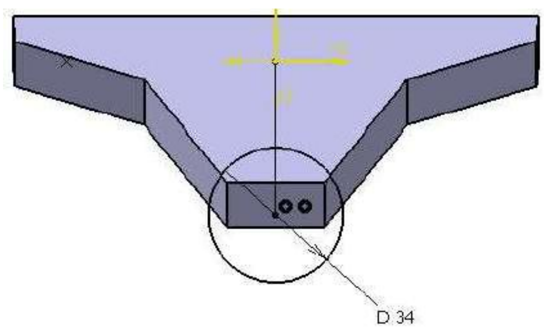


Figure 5-77 *Sketch for the second feature*

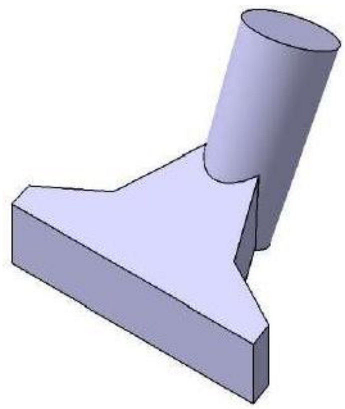


Figure 5-78 *Resulting second feature*

Next, you need to create the third feature of the model to remove the unwanted portion of the second feature.

9. Select the ZX plane and invoke the **Sketcher** workbench. Draw the open sketch, as shown in Figure 5-79, and exit the **Sketcher** workbench.

10. Extrude the sketch using the **Pocket** tool up to last on both the sides of the sketch.

11. Using the Hide/Show tool, hide Sketch1, Sketch2, Plane1, and Plane2. The model, after creating the third feature, is shown in Figure 5-80.

Filleting the Edges of the Model

Next, you need to fillet two sets edges of the model. You need to apply the fillet feature twice because two sets of edges need different fillet radii. First you will fillet the set of edges that needs the fillet radius of 12.

1. Double-click on the **Edge Fillet** button in the **Dress-Up Features** toolbar; the **Edge Fillet Definition** dialog box is displayed.

2. Select the edges, as shown Figure 5-81, and set the value of the **Radius** spinner to **12**.

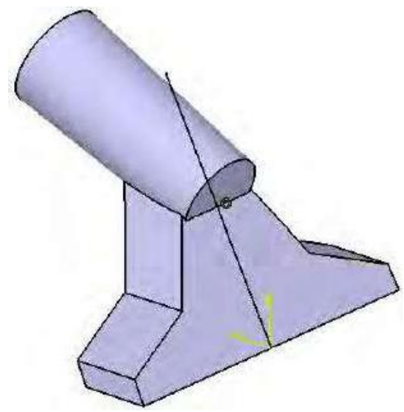


Figure 5-79 *Sketch for the* ***Pocket*** *feature*

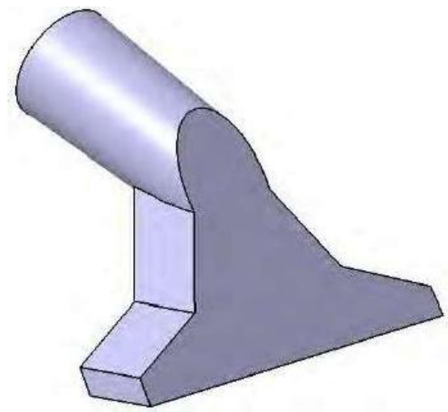


Figure 5-80 *Model after creating the third feature*

3. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after creating the first set of fillet, is shown in Figure 5-82.

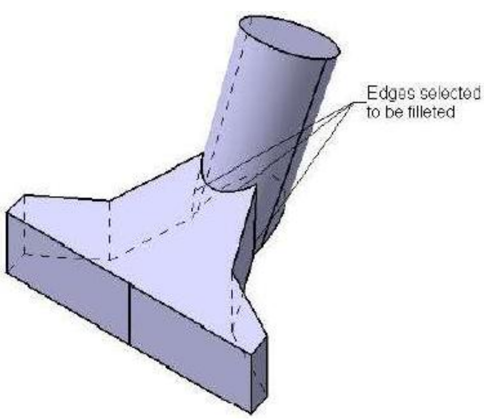


Figure 5-81 *Edges to be selected*

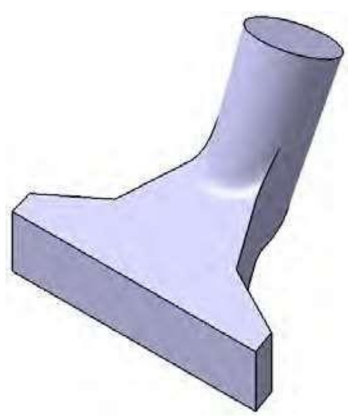


Figure 5-82 *Model after creating the fillet*

Next, you need to apply fillet to the second set of edges. Because you double clicked on the **Edge Fillet** button, the **Edge Fillet Definition** dialog box is again displayed.

4. Select all edges of the model, except the edges that are shown in Figure 5-83.

5. Set the value of the **Radius** spinner to **3** and choose the **OK** button from the **Edge Fillet Definition** dialog box. Cancel this dialog box when it is again displayed. The model, after applying fillet to the second set of edges, is shown in Figure 5-84.

Creating the Shell Feature

The last feature that you need to create is the shell feature. The shell feature will also be used to remove the end faces of the model, leaving behind a thin walled structure.

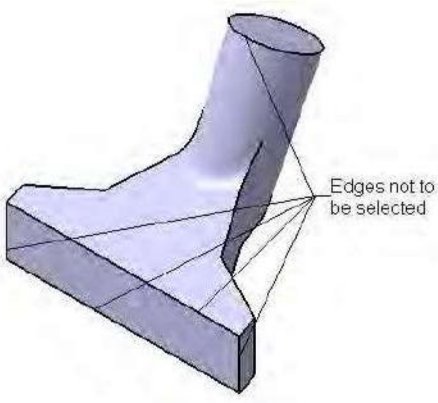


Figure 5-83 Edges not to be selected

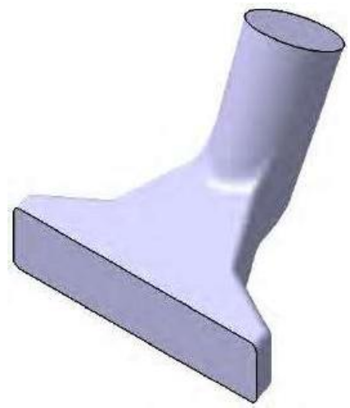


Figure 5-84 Model after creating second fillet

1. Choose the **Shell** button from the **Dress-Up Features** toolbar; the **Shell Definition** dialog box is displayed. 

2. Select the faces to be removed, as shown in Figure 5-85.

3. Set the value of the **Default inside thickness** spinner to **2** and choose the **OK** button from the **Shell Definition** dialog box. The final model, after creating the shell feature, is shown in Figure 5-86.

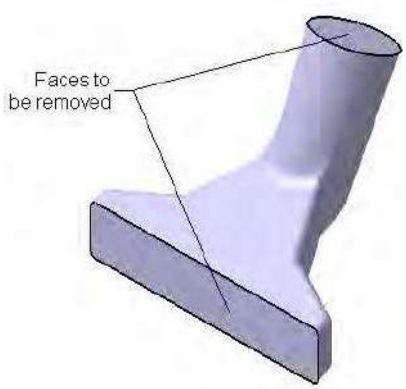


Figure 5-85 Faces to be removed

Figure 5-86 Final model after shelling

Saving and Closing the Files

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create *c05* folder inside the *CATIA* folder.

2. Enter the name of the file as *c05tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c05* folder.

3. Close the part file by choosing **File** > **Close** from the menu bar.

EXPERIMENT 6

In this tutorial, you will create the model of the plastic cover shown in Figure 5-87. The views and dimensions of this model are shown in Figure 5-88. (**Expected time: 30 min**)

Figure 5-87 *Model of the Plastic Cover for Tutorial 2*

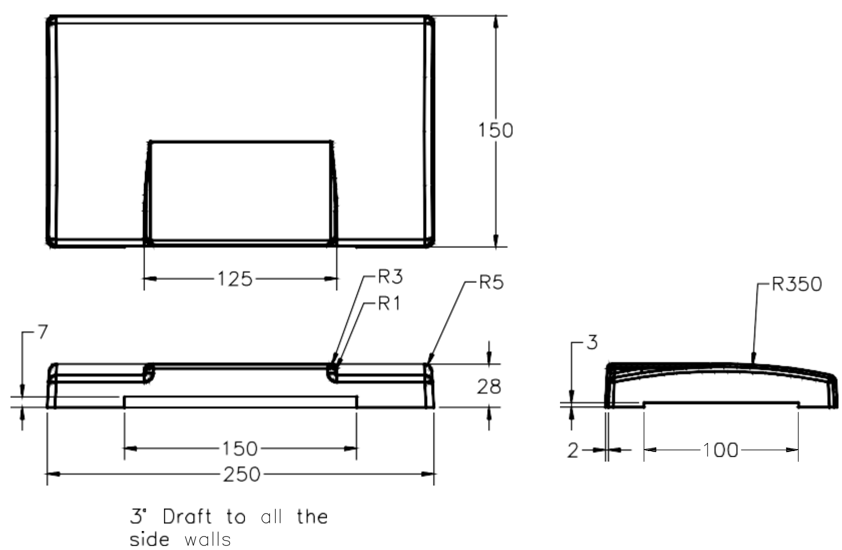


Figure 5-88 *Views and dimensions of the Plastic Cover for Tutorial 2*

The following steps are required to complete this tutorial:

a. Create the base feature of the model by extruding the sketch drawn on ZX plane equally to both the sides of the sketch plane, refer to Figures 5-89 and 5-90.
b. Create the second feature by extruding the sketch drawn on a plane created at an offset distance from the XY plane, refer to Figures 5-91 and 5-92.

c. Add the draft feature to all faces of the model except the upper and the lower faces, refer to Figure 5-93.
d. Fillet the edges of the model, refer to Figures 5-94 through 5-99.
e. Shell the model using the **Shell** tool by removing the bottom face of the model, refer to Figures 5-100 and 5-101.
f. Create two pocket features to complete the model, refer to Figure 5-102.

Creating the Base Feature of the Model

First, you need to create the base feature of the model. The base feature of the model will be created by extruding the sketch drawn on the ZX plane and the sketch will be extruded equally to both the sides of the sketching plane using the **Mirrored extent** option.

1. Start a new part file. Select the ZX plane as the sketching plane and invoke the **Sketcher** workbench.

2. Draw the sketch, as shown in Figure 5-89, and exit the **Sketcher** workbench.

3. Invoke the **Pad Definition** dialog box and set the value of the **Length** spinner to **125**.

4. Select the **Mirrored extent** check box and choose the **OK** button from the **Pad Definition** dialog box. The model, after creating the base feature, is shown in Figure 9-90.

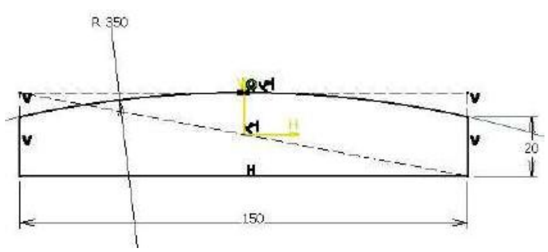


Figure 5-89 *Sketch of the base feature*

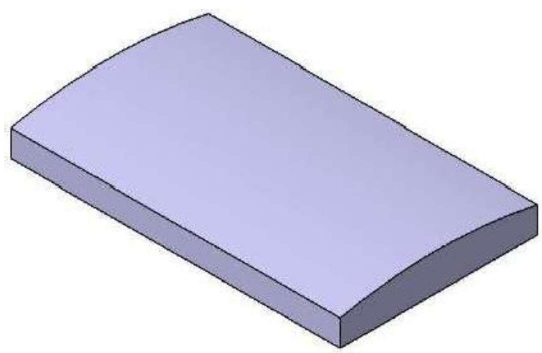


Figure 5-90 *Model after creating the base feature*

Creating the Second Feature

The second feature of the model will be created by extruding the sketch drawn on a plane created at an offset distance of 14 from the XY plane.

1. Create a plane at an offset distance of 14 mm from the XY plane.

2. Invoke the **Sketcher** workbench using the newly created plane as the sketching plane.

3. Draw the sketch, as shown in Figure 5-91, and exit the **Sketcher** workbench.

4. Invoke the **Pad Definition** dialog box and choose the **Reverse Direction** button.

5. Select the **Up to next** option from the **Type** drop-down list and exit the **Pad Definition** dialog box. The model, after creating the second feature, is shown in Figure 5-92.

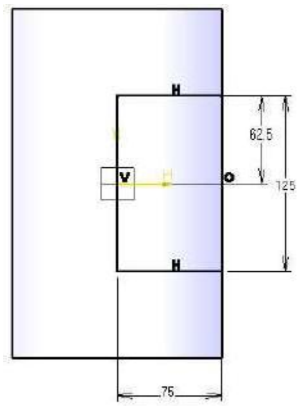


Figure 5-91 *Sketch of the second feature*

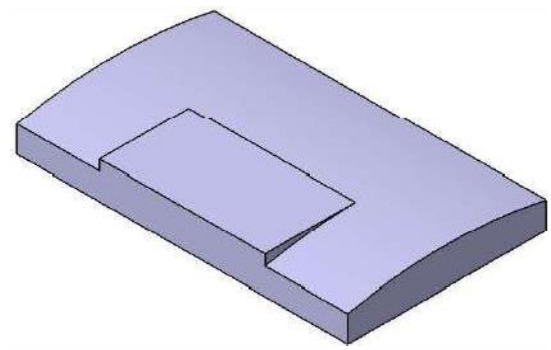


Figure 5-92 *Model after creating the second feature*

Adding Draft to the Faces of the Model

After creating the second feature of the model, you need to add draft to faces of the model. The draft angle is added to the model to make sure that the component is smoothly ejected from the die. Draft angle is one of the most important aspects of designing the components to be formed, molded, or casted.

1. Choose the **Draft Angle** button from the **Dress-Up Features** toolbar. The **Draft Definition** dialog box is displayed and you are prompted to select the faces to draft.

2. Select all the vertical faces of the base feature and the second feature from the geometry area.

3. Click once on the **Selection** selection area available in the **Neutral Element** area and select the bottom face of the base feature as the neutral element. Make sure that the pulling direction is in the upwards direction.

4. Set the value of the **Angle** spinner to **3** and choose the **OK** button from the **Draft Definition** dialog box. The model, after creating the draft feature, is shown in Figure 5-93.

Filleting the Edges of the Model

Next, you need to fillet the edges of the model. In this model, you need to fillet three separate set of edges using the **Edge Fillet** tool.

1. Choose the **Edge Fillet** button from the **Dress-Up Features** toolbar; the **Edge Fillet Definition** dialog box is displayed.

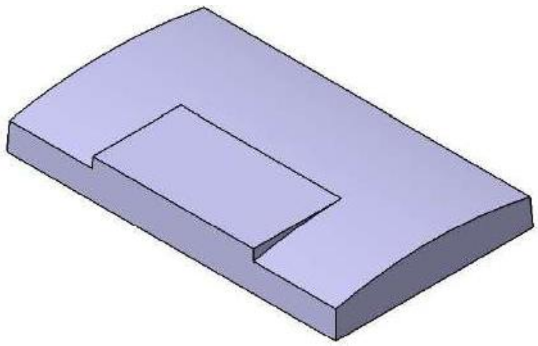


Figure 5-93 Model after drafting all the vertical faces

2. Select the edges shown in Figure 5-94 and set the value of the **Radius** spinner to **3**.

3. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after filleting the first set of edges, is shown in Figure 5-95.

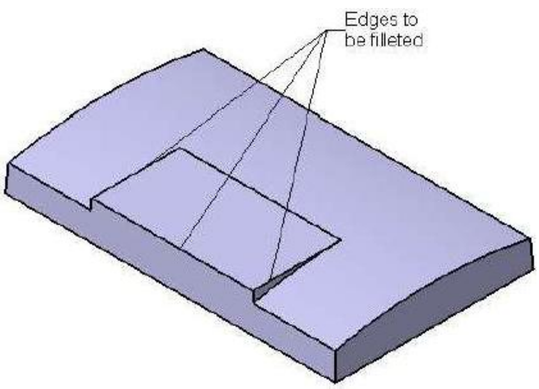


Figure 5-94 Edges to be selected

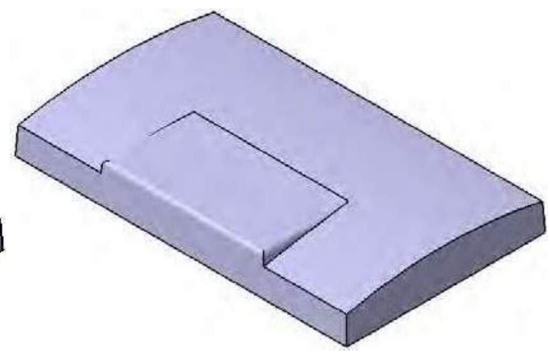


Figure 5-95 Model after filleting the first set of edges

4. Invoke the **Edge Fillet Definition** dialog box again to fillet the second set of edges.

5. Select the edge shown in Figure 5-96 and set the value of the **Radius** spinner to **1**.

6. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after the second set of edges, is shown in Figure 9-97.

7. Invoke the **Edge Fillet Definition** dialog box again to fillet the third set of edges.

8. Select all the edges of the model, except the edges shown in Figure 5-98, and set the value of the **Radius** spinner to **5**.

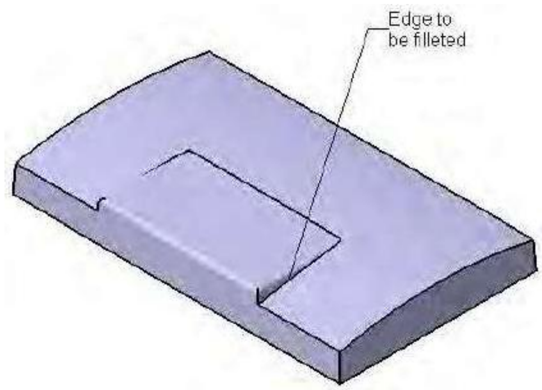


Figure 5-96 Edge to be filleted

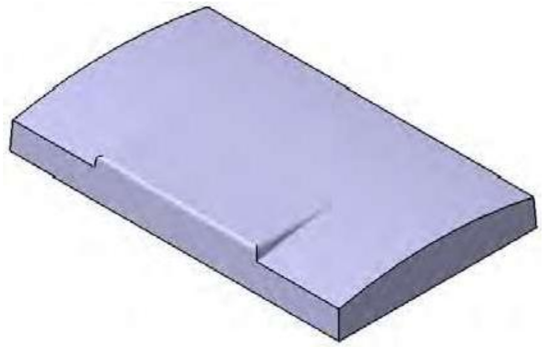


Figure 5-97 Model after filleting the second set

9. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The resulting filleted model is shown in Figure 5-99.

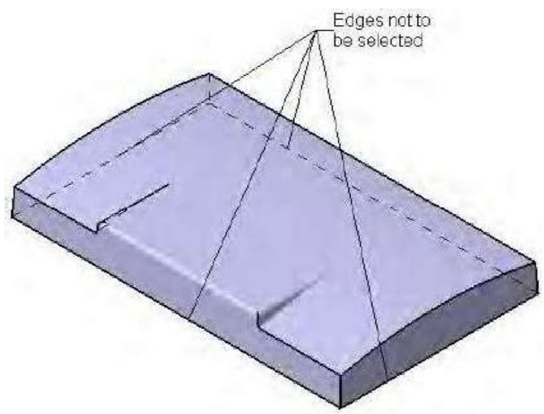


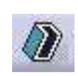
Figure 5-98 Edges not to be selected

Figure 5-99 Resulting filleted model.

Creating the Shell Feature

Finally, you need to shell the model and remove its bottom face.

It is always recommended to shell the model after adding the draft angle and the fillet feature to maintain the draft angle and the fillet curvature on the inside walls of the shelled model.

1. Choose the **Shell** button from the **Dress-Up Features** toolbar. The **Shell Definition** dialog box is displayed. 

2. Select the face to be removed, as shown in Figure 5-100, and set the value of the **Default inside thickness** to **2**.

3. Choose the **OK** button from the **Shell Definition** dialog box. The rotated view of the model, after adding the shell feature, is shown in Figure 5-101.

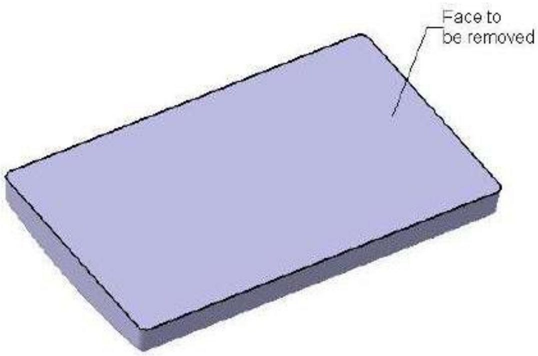


Figure 5-100 *Face to be removed* ***Figure 5-101*** *Resulting shelled model*

4. Use the **Pocket** tool to create the two pocket features. The final model, after creating the other two features, is shown in Figure 5-102.

Figure 5-102 *Final model after creating the remaining features*

EXPERIMENT 7

Draw the sketch of the model shown in Figure 1-51. The sketch to be drawn is shown in Figure 1-52. Do not dimension the sketch, as the solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

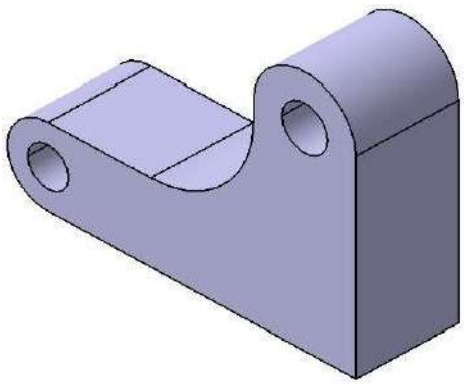


Figure 1-51 *Solid Model for Exercise 1*

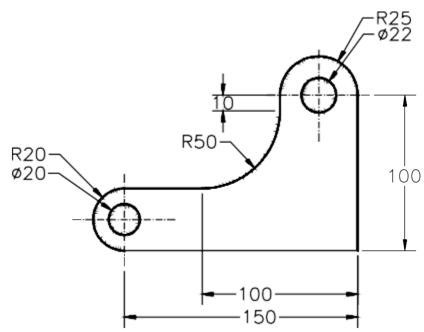


Figure 1-52 *Sketch of the model*

EXPERIMENT 8

Draw the sketch of the model shown in Figure 1-53. The sketch to be drawn is shown in Figure 1-54. Do not dimension the sketch as the solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

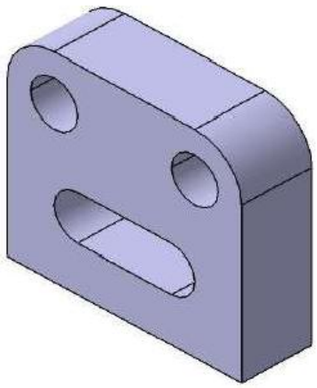


Figure 1-53 *Solid Model for Exercise 2*

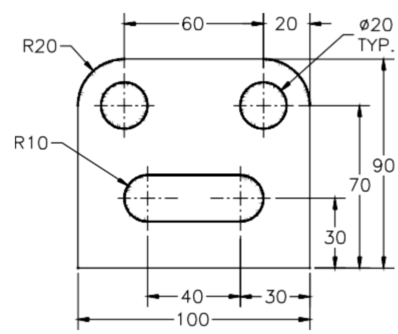


Figure 1-54 *Sketch of the model*

EXPERIMENT 9

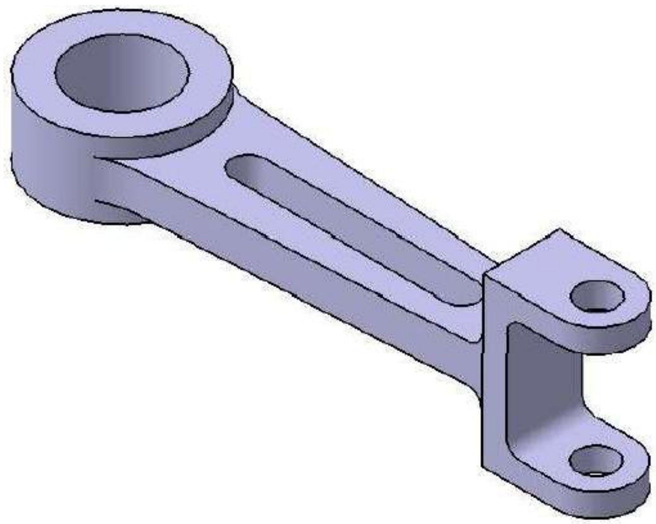


Figure 5-103 *Model of the Clutch Lever for Exercise 1*

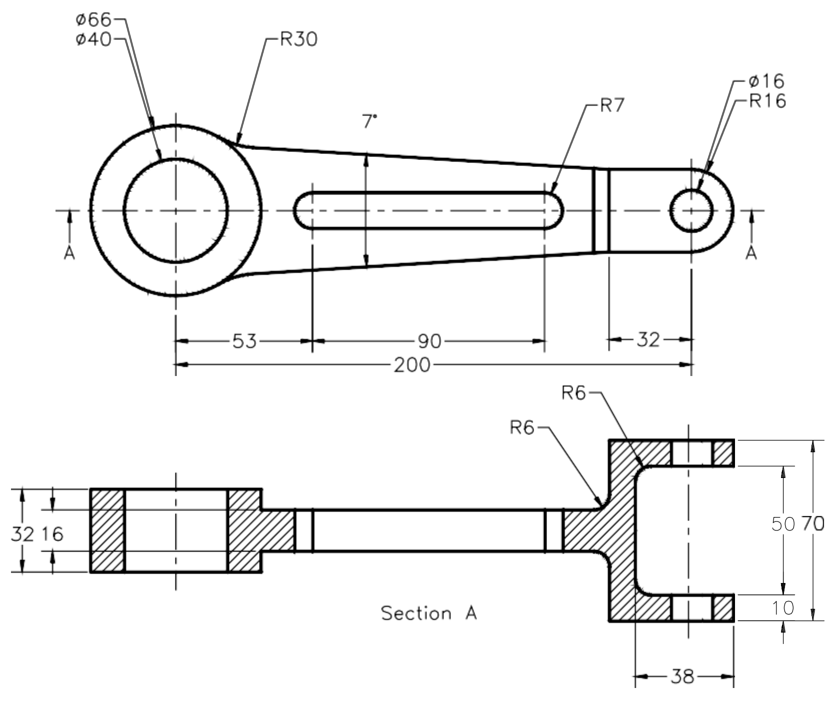


Figure 5-104 *Views and dimensions of the Clutch Lever for Exercise 1*

EXPERIMENT 10

Create the model of the Clamp Stop shown in Figure 5-105. The views and dimension of the model are shown in Figure 5-106. **(Expected time: 1 hr)**

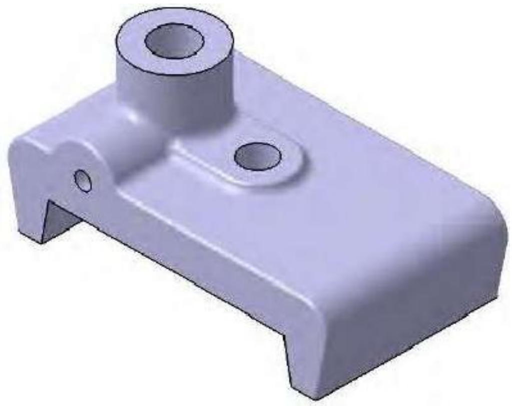


Figure 5-105 *Model of the Clamp Stop for Exercise 2*

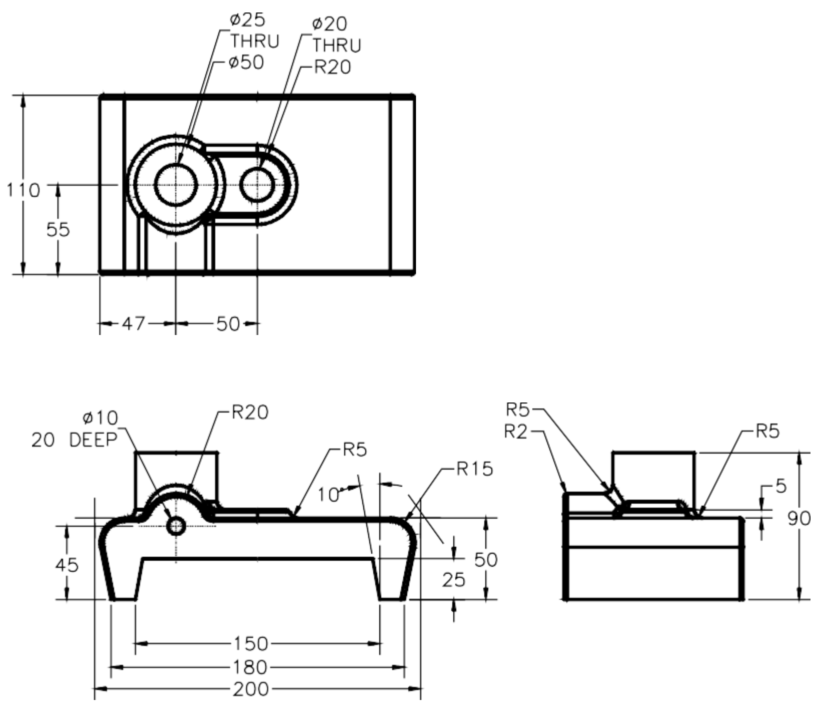


Figure 5-106 *Views and dimensions of the Clamp Stop for Exercise 2*