

Tutorial 3

Editing and Modifying Sketches

Learning Objectives

After completing this tutorial, Student will be able to:

- *Edit sketches using various editing tools*
- *Create circular patterns of sketched entities*
- *Create rectangular patterns of sketched entities*
- *Modify sketched entities*

EDITING SKETCHED ENTITIES

SOLIDWORKS provides you with a number of tools that can be used to edit the sketched entities. You can trim, extend, offset, or mirror the sketched entities using these tools. You can also perform various other editing operations by using these tools. Various editing operations and the tools used to perform them are discussed in this tutorial.

Tutorial Exercise 1

In this tutorial, you will create the base sketch of the model shown in Figure 3-1. The sketch of the model is shown in Figure 3-2. You will create the sketch of the base feature by using the sketch tools. Also, you will modify and edit the sketch using various modifying options. Do not create the center marks and centerlines, as they are for your reference only.

(Expected time: 30 min)

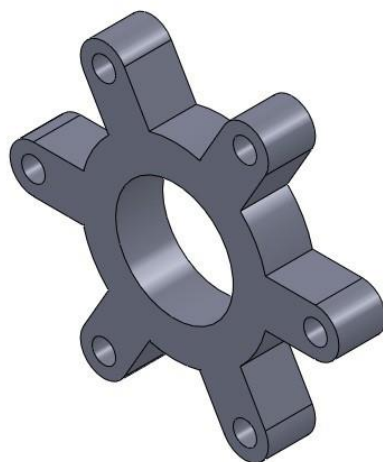


Figure 3-1 Solid model for Tutorial 1

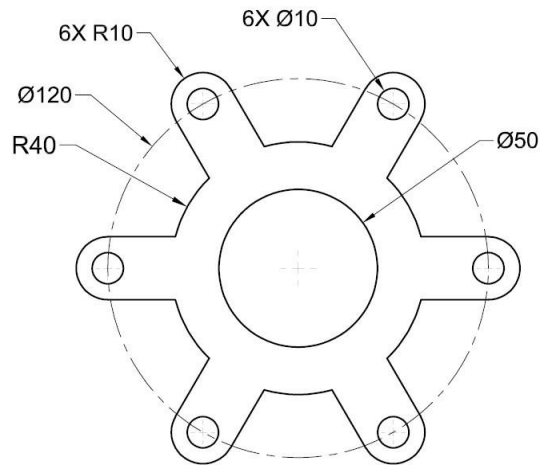


Figure 3-2 Sketch for Tutorial 1

The following steps are required to complete this tutorial:

- a. Start SOLIDWORKS and then a new part document.
- b. Invoke the sketching environment.
- c. Modify the snap, grid, and units settings.
- d. Draw outer loop of the sketch, refer to Figures 3-5 through 3-11.
- e. Draw the inner cavity of the sketch by using the **Circle** and **Circular Sketch Pattern** tools, refer to Figure 3-12.
- f. Save the sketch.

Starting SOLIDWORKS and a New SOLIDWORKS Document

1. Start SOLIDWORKS by choosing **Start > All Programs > SOLIDWORKS 2016 > SOLIDWORKS 2016** or by double-clicking on the shortcut icon of SOLIDWORKS 2016 on the desktop of your computer; the SOLIDWORKS window is displayed.
2. Choose the **New** tool from the Menu Bar; the **New SOLIDWORKS Document** dialog box is displayed.
3. In this dialog box, the **Part** button is chosen by default. Choose the **OK** button from this dialog box; a new SOLIDWORKS part document is invoked.

Invoking the Sketching Environment

Next, you need to invoke the sketching environment.

1. Choose the **Sketch** tab from the CommandManager. Next, choose the **Sketch** tool from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select a plane to create the sketch. Also, the three default planes (**Front Plane**, **Right Plane**, and **Top Plane**) available in SOLIDWORKS are temporarily displayed on the screen.
2. Select the **Front Plane** as the sketching plane; the sketching environment is invoked and the plane is oriented normal to the view.

Modifying the Snap, Grid, and Unit Settings

It is assumed that while installing SOLIDWORKS, you have selected the **MMGS** (**millimeters, gram, second**) option for measuring the length. Therefore, the length of an entity will be measured in millimeters in the current file. But, if you select some other unit at the time of installation, you need to change the linear and angular units before drawing the sketch. For this tutorial, you need to modify the grid and snap settings so that the cursor jumps through a distance of 10 mm.

1. Choose the **Unit system** button located in the Status Bar; a flyout is displayed with a tick mark next to the unit system of the current document, as shown in Figure 3-3.

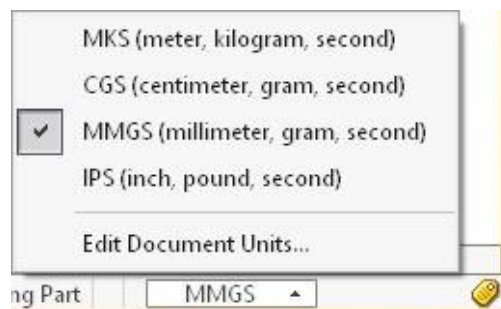


Figure 3-3 The flyout showing the currently selected unit system

2. Choose the **Edit Document Units** option from this flyout; the **Document Properties - Units** dialog box is displayed.

Note

If you have selected millimeters as the unit of measurement while installing SOLIDWORKS, you can skip step 3 in this section.

3. Select the **MMGS (millimeter, gram, second)** in the **Unit system** area. Also, select the **degrees** option from the **Angle** area in the **Units** column.

Note

*You can also select the required unit system for the current document by selecting the required unit system from the **Unit system** flyout directly. However, in this case, if you are in the sketcher environment, you will automatically exit from the sketcher environment.*

As evident from Figure 3-2, the dimensions in the sketch are multiples of 10. Therefore, you need to modify the grid and snap settings such that the cursor jumps through a distance of 10 mm.

4. Select the **Grid/Snap** option on the left of the **Document Properties - Units** dialog box to display all the options related to the grid and snap. Also, note that the name of the dialog box is modified to the **Document Properties - Grid/Snap** dialog box.
5. In this dialog box, set the value in the **Major grid spacing** and **Minor-lines per major** spinner to **100** and **10**, respectively.
6. Invoke the **System Options - Relations/Snaps** dialog box by choosing the **Go To System Snaps** button from the **Document Properties - Grid/Snap** dialog box. Next, make sure that the **Grid** check box is selected in it.
7. Choose the **OK** button to exit from the dialog box.

Now, when you move the cursor, the coordinates that are displayed closer to the lower right corner of the drawing area show an increment of 10 mm.

As evident from Figure 3-2, the sketch consists of the outer loop and inner cavities. It is recommended that you create the outer loop of the sketch first and then the inner cavities.

Drawing the Outer Loop of the Sketch

The outer loop of the sketch will be drawn by using the **Circle**, **Line**, and **Trim Entities**

tools.

1. Choose the **Circle** tool from the **Circle** flyout in the **Sketch CommandManager**; the **Circle PropertyManager** is displayed with the **Circle** button chosen in the **Circle Type** rollout.

Tip

*You can also choose an appropriate method to create a circle from the **Circle Type** rollout of the **Circle PropertyManager**. The **Circle** tool of this rollout is used to create a circle by specifying the center point of the circle and then defining its radius. The **Perimeter Circle** tool in this rollout is used to draw a circle by defining three points that lie on its periphery.*

2. Move the cursor toward the origin and specify the center point of the circle when it snaps to the origin and the symbol of coincident relation is displayed below the cursor.
3. Move the cursor horizontally toward the right and click when the radius of the circle above the cursor shows the value 40.
4. Right-click in the drawing area to invoke the shortcut menu. Now, choose the **Select** option from the shortcut menu to exit the **Circle** tool.
5. Choose the **Zoom to Area** tool from the **View (Heads-Up)** toolbar. Press and hold the left mouse button and drag the cursor to define a window such that the sketched circle is placed in the window. Next, release the left mouse button to increase the display area of the sketch.

The **Zoom to Area** tool is used to magnify a specified area of the drawing window. The portion of the drawing that is inside the magnified area can be viewed in the current window.

6. Exit the tool by choosing the **Select** option from the shortcut menu that is displayed after right-clicking in the drawing area.
7. Press the right mouse button and drag the cursor; the Mouse Gesture is displayed in the drawing area, refer to Figure 3-4. Move the cursor over the **Circle** tool; the **Circle** tool is invoked and the cursor is changed to a circle cursor.

Note

You can also invoke the **Circle** tool from the **Sketch CommandManager**, as discussed earlier.



Figure 3-4 Tools displayed in the Mouse Gesture in the sketching environment

In SOLIDWORKS, when you press the right mouse button and drag the cursor in a direction, a set of radially arranged tools are displayed. This action is called Mouse Gesture. After displaying the tools by using the Mouse Gesture, you can move the cursor over a particular tool to invoke it.

By default, four tools are displayed in the Mouse Gesture. However, you can customize the Mouse Gesture and display eight tools. To customize the Mouse Gesture, invoke the **Customize** dialog box by choosing the **Tools > Customize** from the SOLIDWORKS menus and then choose the **Mouse Gestures** tab. Next, select the **8 gestures** radio button from the top right corner of the dialog box to display the Mouse Gesture with 8 tools. The **Enable mouse gestures** check box in this dialog box is used to enable or disable the Mouse Gesture.

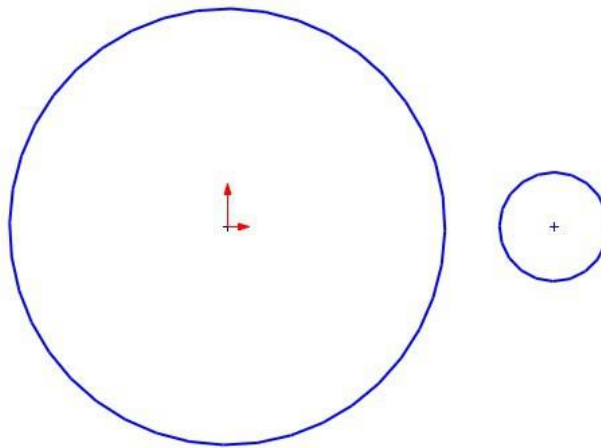
You can also customize the commands of the Mouse Gesture. To do so, click on the respective field in the display area of the dialog box; a down arrow will be displayed. Next, click on the down arrow; a flyout will be displayed. Choose the required option from this flyout to assign the respective tool to the Mouse Gesture. To view only the commands that are assigned to the Mouse Gesture, select the **Show only commands with mouse gestures assigned** check box.

Note

The tools displayed by Mouse Gesture are different for different environments. For

*example, when you invoke the Mouse Gesture in the **Part** environment, it displays the tools of the **Part** environment. Similarly, when you invoke the Mouse Gesture in the **Sketching** environment, it displays the tools of the **Sketch** environment.*

8. Move the cursor to the location whose coordinates are 60, 0, 0 and specify the center point of the circle at this location by clicking the left mouse button.
9. Move the cursor horizontally toward the right and click the left mouse button when the radius above the cursor shows 10 mm. Next, exit the tool by right-clicking in the drawing area and choosing the **Select** option from the shortcut menu. Figure 3-5 shows the sketch after drawing the circles of 80 mm and 20 mm diameters.



***Figure 3-5** Sketch after drawing circles*

10. Invoke the **Line** tool from the Mouse Gesture. To do so, press and hold the right mouse button and drag the cursor to a small distance; a set of sketching tools is displayed. Move the cursor over the **Line** tool; the **Line** tool is invoked and the cursor is converted into a line cursor.

Note

*You can also invoke the **Line** tool from the **Sketch CommandManager**, as discussed earlier.*

11. Move the line cursor to the location whose coordinates are 60, 10, 0. Next, specify the start point of the line when the line cursor snaps to the circle of diameter 20 mm and displays the symbol of coincident relation below the line cursor.

12. After specifying the first point of the line, move the line cursor horizontally toward the circle of diameter **80** mm; a rubber band line is attached to the cursor.
13. Specify the end point of the line when the line cursor snaps to the circle of diameter 80 mm and the symbol of coincident and horizontal relations are displayed below the line cursor.
14. Right-click in the drawing area and choose the **End Chain** option from the shortcut menu; the current chain ends but the **Line** tool still remains active.

Note

*When you terminate the process of drawing a line by double-clicking on the screen or by choosing the **End Chain** from the shortcut menu, the current chain would end. However, the **Line** tool would remain active.*

15. Move the line cursor to the location whose coordinates are 60, -10, 0. Now, specify the start point of the line when the line cursor snaps to the circle of diameter 20 mm and the symbol of coincident relation is displayed below the line cursor.
16. Move the line cursor horizontally towards the circle of diameter 80 mm; a rubber band line is attached to the cursor.
17. Specify the end point of the line when the line cursor snaps to the circle of diameter 80 mm and the symbols of coincident and horizontal relations are displayed below the line cursor.
18. Right-click in the drawing area and choose the **Select** option from the shortcut menu to exit the **Line** tool. Figure 3-6 shows the sketch after drawing the horizontal lines.

Now, you need to trim the unwanted entities of the sketch by using the **Trim Entities** tool.

19. Choose the **Trim Entities** tool from the **Sketch CommandManager**; the **Trim PropertyManager** is displayed on the left of the drawing area, refer to Figure 3-7.

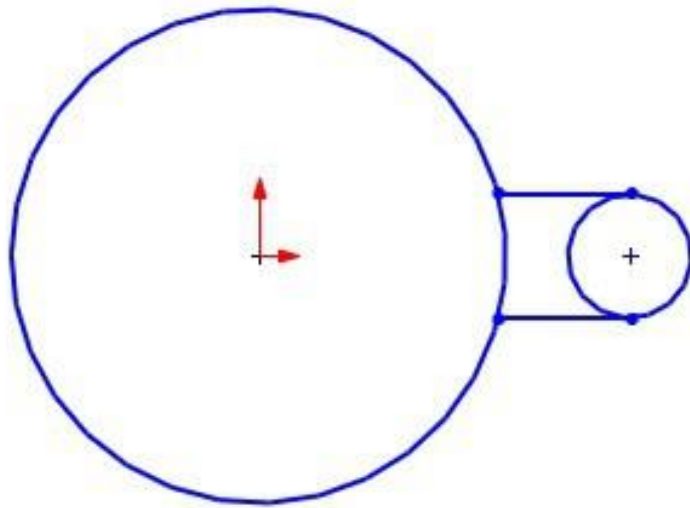


Figure 3-6 Sketch after drawing the lines



Figure 3-7 The Trim PropertyManager

The **Trim Entities** tool is used to trim the unwanted entities in a sketch. You can use this tool to trim a line, arc, ellipse, parabola, circle, spline, centerline intersecting another line,

arc, ellipse, parabola, circle, spline, or centerline. You can also extend the sketched entities using the **Trim Entities** tool.

The options of the **Trim PropertyManager** are discussed next.

Message Rollout

The **Message** rollout in the **Trim PropertyManager** informs you about the procedure of trimming and extending the sketched elements, depending upon the option selected in the **Options** rollout.

Options Rollout

The **Options** rollout displays all options that are used to trim the sketched entities. These options are discussed next.

Power trim: When the **Power trim** button is chosen in the **Options** rollout, the **Message** rollout of the **Trim PropertyManager** will inform you about the procedure of trimming and extending the sketched elements. To trim the unwanted portion of a sketch using this option, press and hold the left mouse button and drag the cursor. You will notice that a gray-colored drag trace line is displayed along the path of the cursor. When you drag the cursor across the unwanted sketched entity, it will be trimmed and a small red-colored box will be displayed in its place. You can continue trimming the entities by dragging the cursor across them. After trimming all unwanted entities, release the left mouse button.

To extend or shorten an entity dynamically when the **Power trim** button is chosen in the **Trim PropertyManager**, click once on the entity and then move the cursor; the entity will extend or shorten dynamically depending upon the direction of movement. Move the cursor up to a level to which the entity has to be extended or shortened. Press the left mouse button to complete the operation.

To extend a line or a curve such that it intersects the other entity, select the first entity and then the second entity; the first entity will extend and intersects the second entity. While extending, if the first entity cannot intersect the second entity, then the first entity will extend up to the apparent intersection point.

Note

If the first entity cannot be extended to intersect the second entity, then a tooltip will be

displayed stating that the selected entity cannot be extended to intersect the second entity.

Corner: The **Corner** button in the **Options** rollout is used to trim or extend the sketched entities in such a way that the resulting entities form a corner. To trim the unwanted elements using this option, choose the **Corner** button from the **Options** rollout; you will be prompted to select an entity. Select the entity from the geometry area; you will be prompted to select another entity. When you move the cursor over the second entity, the preview of the resulting entity will be displayed in a different color. In the second entity, select the portion to be retained.

You can also extend the entities using this tool. To do so, choose the **Corner** button from the **Options** rollout and select the entities to be extended; the selected entities will be extended to their apparent intersection.

Trim away inside: The **Trim away inside** button in the **Options** rollout is used to trim the portion of the selected entity that lies inside two bounding entities. To do so, invoke the **Trim PropertyManager** and choose the **Trim away inside** button from the **Options** rollout; the **Message** rollout will be displayed informing you to select the two bounding entities, and then to select the entities to be trimmed. Select the bounding entities from the drawing area. Now, select the entities to be trimmed from the drawing area; the portion of the entity inside the bounding entities will be removed and the portion outside the bounding entities will be retained.

Trim away outside: The **Trim away outside** button in the **Options** rollout is used to trim the portion of an entity outside the bounding entities. To do so, invoke the **Trim PropertyManager** and choose the **Trim away outside** button from the **Options** rollout; the **Message** rollout will inform you to select the two bounding entities, and then to select the entities to be trimmed.

Trim to closest: The **Trim to closest** button is used to trim the selected entity to its closest intersection. To do so, invoke the **Trim PropertyManager** and then choose the **Trim to closest** button from the **Options** rollout; the cursor will be replaced by the trim cursor. Move the trim cursor near the portion of the sketched entity to be removed; the entity or the portion of the entity to be removed will be highlighted in orange. Press the left mouse button to remove the highlighted entity.

You can also extend the sketched entities when the **Trim to closest** button is chosen in the **Trim PropertyManager**. To do so, move the trim cursor to the entity to be extended; the sketched entity will turn orange. Now, press the left mouse button and

drag the cursor to the entity up to which it has to be extended. You will notice the preview of the extended entity. Release the left mouse button when the preview of the extended entity appears; the entity will be extended.

20. Choose the **Trim to closest** button from the **Options** rollout of the **Trim PropertyManager**; the cursor is replaced by the trim cursor.
21. Move the trim cursor near the sketched entities to be removed one by one. Refer to Figure 3-8 for the entities to be trimmed. Next, click the left mouse button to trim them. Figure 3-9 shows the sketch after trimming the entities.

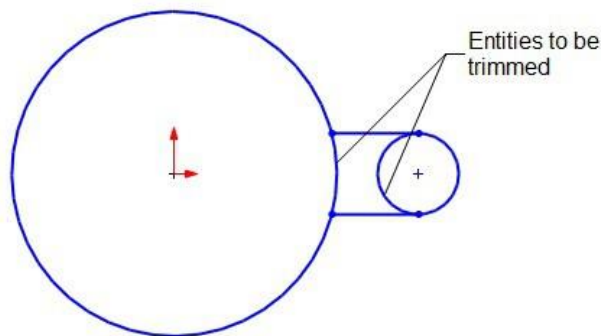


Figure 3-8 Entities to be trimmed

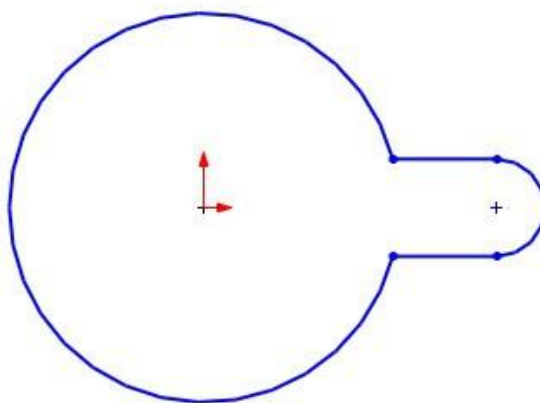


Figure 3-9 Sketch after trimming the entities

22. Next, choose the **Close Dialog** button from the **Trim PropertyManager** to exit it.
23. Press the CTRL key and select both the horizontal lines and the arc of radius 10

mm. Next, choose the **Circular Sketch Pattern** tool from the **Linear Sketch Pattern** flyout in the **Sketch CommandManager**; the **Circular Pattern PropertyManager** is displayed, refer to Figure 3-10 and the preview of the circular pattern with default settings is displayed in the drawing area.

Also, you will notice that the center of the circular pattern is placed at the origin and an arrow is displayed, indicating that the origin is the center of the circular pattern. If required, you can modify the center of the circular pattern by setting the coordinates of the point in the **Center X** and **Center Y** spinners in the **Parameters** rollout of the PropertyManager. Alternatively, drag the arrow displayed at the center of the pattern to the required location.

The **Circular Sketch Pattern** tool is used to create a circular pattern of the selected entities. You can also create linear pattern of the selected entities. To do so, you need to choose the **Linear Sketch Pattern** tool from the **SketchCommandManager**. Next, you need to define the first and second linear directions for creating the linear pattern. Also, you can define the number of instances in respective direction and spacing between each instance.

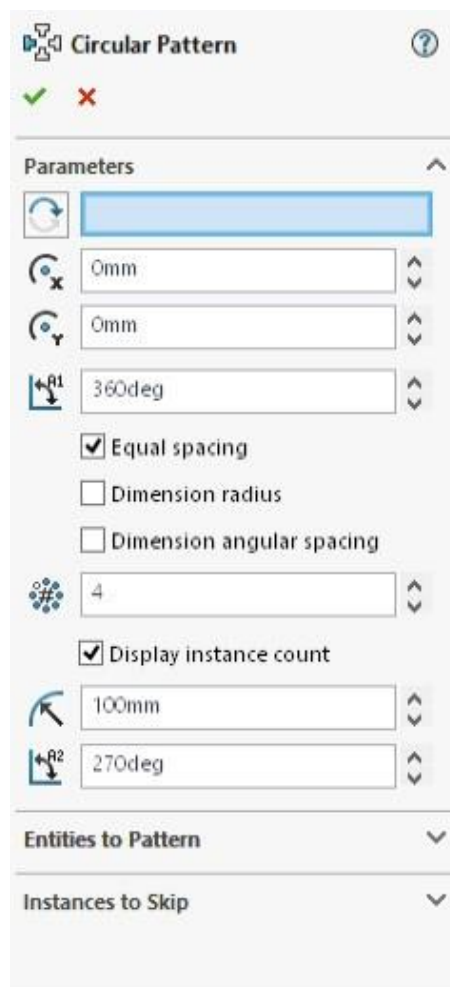


Figure 3-10 The Circular Pattern PropertyManager

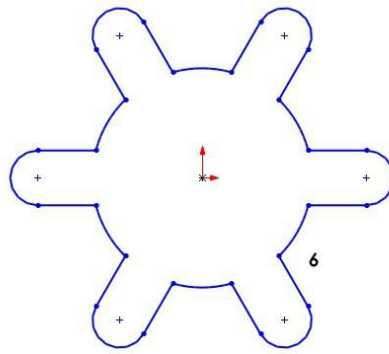


Figure 3-11 Sketch after drawing the outer loop

24. Set the value of the **Number of Instances** spinner to **6**.
25. Make sure that the **Equal spacing** check box is selected. Next, clear the **Dimension radius** and **Dimension angular spacing** check boxes.
26. Select the **Display instance count** check box, if it is not selected by default. Accept the other default values and choose the **OK** button to create a pattern.

The **Display instance count** check box is used to display the number of instances in the resulting sketch pattern.

27. Trim the unwanted portion of the 80 mm diameter circle using the **Trim Entities** tool. You need to use the **Trim to closest** button for this trimming.
28. Choose **Close Dialog** after trimming. The outer loop of the sketch is created, as shown in Figure 3-11.

Drawing Inner Cavities of the Sketch

As is evident from Figure 3-2, you need to create seven circles. Out of seven circles, six are to be of 10 mm diameter. Therefore, after drawing the first circle of 10 mm diameter, you need to create the other five circles of same diameter by creating a circular pattern of the parent circle.

1. Invoke the **Circle** tool by using the Mouse Gesture. As soon as you invoke the **Circle** tool, the **Circle PropertyManager** is displayed.

2. Select the center point of the 10 mm radius arc on the left quadrant of the larger circle as the center point of the new circle.
3. Press and hold the CTRL key and draw a circle of radius close to 5. Make sure you press the CTRL key so that the cursor does not snap to the points or grid.
4. Set **5** in the **Radius** spinner in the **Circle PropertyManager**. Next, choose the **Close Dialog** button to exit the **Circle PropertyManager**.

Now, you need to create the other five instances of the circle by using the **Circular Sketch Pattern**.

5. Choose the **Circular Sketch Pattern** tool from the **Linear Sketch Pattern** flyout in the **Sketch CommandManager**; the **Circular Pattern PropertyManager** is displayed and a preview of the circular pattern is displayed with an arrow at the center.
6. Set **6** in the **Number of Instances** spinner.
7. Make sure that the **Equal spacing** check box is selected. Next, clear the **Dimension radius** and **Dimension angular spacing** check boxes.
8. Select the **Display instance count** check box. Accept the remaining default values and choose the **OK** button to create the pattern.
9. Choose the **Circle** tool from the **Sketch CommandManager**; the **Circle PropertyManager** is invoked. Specify the center point of the circle at the origin. Next, press and hold the CTRL key and draw a circle of radius close to 25 mm.
10. Enter **25** in the **Radius** spinner in the **Circle PropertyManager**. Next, choose the **Close Dialog** button to exit the **Circle PropertyManager**. The final sketch of the model is shown in Figure 3-12. In this figure, all relations are hidden for clarity.

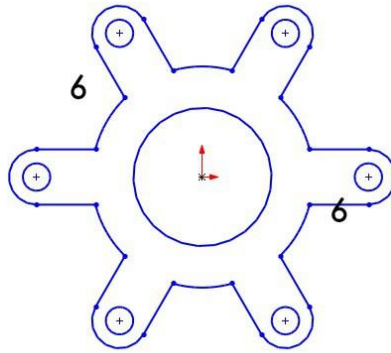


Figure 3-12 *Final sketch*

Saving the Sketch

1. Choose the **Save** button from the Menu Bar; the **Save As** dialog box is displayed.
2. Browse to the *SOLIDWORKS Tutorials* folder, choose the **New Folder** button from the **Save As** dialog box, specify the name of the new folder as *c03*, and then press ENTER twice. Next, enter the name of the document as **c03_tut01** in the **File name** edit box and choose the **Save** button; the document is saved at the location */Documents/SOLIDWORKS Tutorials/c03*.
3. Close the file by choosing **File > Close** from the SOLIDWORKS menus.

Tutorial Exercise 2

In this tutorial, you will create the base sketch of the model shown in Figure 3-13. The sketch of the model is shown in Figure 3-14. You will draw the sketch with a mirror line. The dimensions given in Figure 3-14 are for your reference only. (**Expected time: 30 min**)

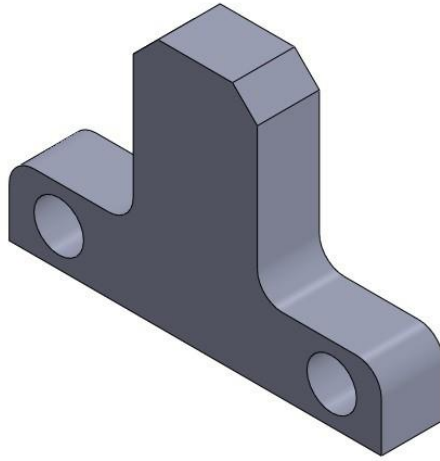


Figure 3-13 Solid model for Tutorial 2

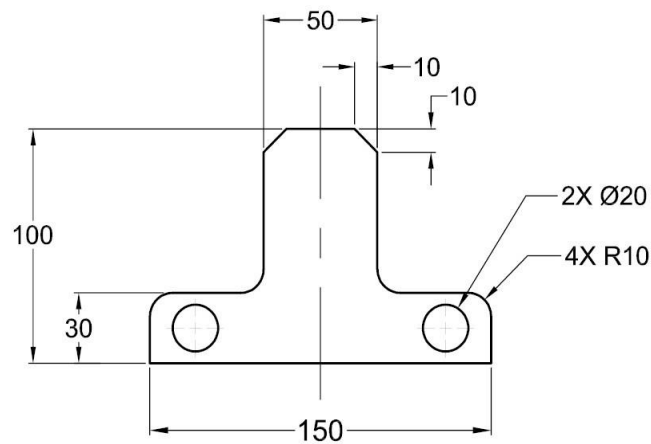


Figure 3-14 Sketch for Tutorial 2

The following steps are required to complete this tutorial:

- Start SOLIDWORKS and then a new part document.
- Invoke the sketching environment.
- Modify the snap, grid, and units settings.
- Create vertical centerline and convert it into a mirror line using the **Dynamic Mirror** tool.
- Create the left side sketch; the sketch will automatically be mirrored on the right side of the mirror line, refer to Figure 3-17.
- Apply fillets, refer to Figure 3-20.
- Apply chamfers, refer to Figure 3-23.
- Save the sketch.

Starting SOLIDWORKS and a New SOLIDWORKS Document

1. Start SOLIDWORKS and then invoke the **New SOLIDWORKS Document** dialog box by choosing the **New** tool from the Menu Bar. Next, invoke the **Part** environment.

Invoking the Sketching Environment

Now, you will invoke the sketching environment.

1. Choose the **Sketch** tab from the **CommandManager**. Next, choose the **Sketch** tool from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.
2. Select the **Front Plane**; the sketching environment is invoked and the plane is oriented normal to the view.

Modifying the Snap, Grid, and Unit Settings

Before drawing the sketch, you need to modify the grid and snap settings to make the cursor jump through a distance of 5 mm.

1. Choose the **Options** button from the Menu Bar; the **System Option - General** dialog box is displayed.
2. Choose the **Document Properties** tab and select the **Grid/Snap** option from the area on the left of the dialog box. Set the values **50** and **10** in the **Major grid spacing** and the **Minor-lines per major** spinners, respectively.
3. If on invoking the sketching environment, the grid is displayed, you can turn off its display by clearing the **Display grid** check box in the **Grid** area.
4. Choose the **Go To System Snaps** button and select the **Grid** check box. Next, clear the **Snap only when grid is displayed** check box, if it is already selected.

If you have selected a unit other than millimeter to measure the length while installing SOLIDWORKS, you need to select millimeter as the unit for the current drawing by

following the next two steps.

5. Choose the **Document Properties** tab and select the **Units** option from the area on the left of the **Document Properties - Grid/Snap** dialog box.
6. Select the **MMGS (millimeter, gram, second)** radio button in the **Unit system** area. Next, select **degrees** from the drop-down list in the cell corresponding to the **Angle** row and the **Unit** column.
7. After modifying the necessary settings, choose the **OK** button.

The coordinates displayed close to the lower right corner of the SOLIDWORKS window show an increment of 5 mm when you move the cursor in the drawing area after exiting the dialog box.

Drawing the Centerline and Converting it into a Mirror Line

It is recommended that you draw symmetrical sketches about the centerline that is created by using the **Centerline** tool.

1. Choose the **Centerline** tool from the **Line** flyout in the **Sketch CommandManager**; the **Insert Line PropertyManager** is displayed. Note that the **For construction** check box is selected in this PropertyManager.
2. Move the line cursor toward the origin and specify the start point of the centerline when it snaps to the origin and the symbol of coincident relation is displayed above the cursor.
3. Move the cursor vertically upward and specify the end point of the centerline when the length of the centerline is displayed above the cursor close to 110. You may have to zoom and pan the drawing to draw a line of this length.
4. Press F to fit the drawing on the screen. Next, right-click and choose the **Select** option from the shortcut menu; the line cursor is replaced by the select cursor.
5. Select the centerline and choose **Tools > Sketch Tools > Dynamic Mirror** from the SOLIDWORKS menus; the selected centerline is converted into mirror line and the

dynamic mirror option is activated.

You can confirm the creation of the mirror line and the activation of the dynamic mirror option by observing the symmetrical symbol displayed on both ends of the centerline.

Drawing the Sketch

Next, you need to draw the sketch of the model. You will draw the sketch on the left of the mirror line and the same sketch will be created automatically on the right side of the mirror line. The symmetrical relation is applied between the parent entity and the mirrored entity.

1. Press the L key on the keyboard; the **Line** tool is invoked.
2. Move the line cursor towards the origin and specify the start point of the line when it snaps to the origin and the symbol of coincident relation is displayed above the cursor.
3. Move the cursor horizontally toward the left and specify the endpoint of the line when 75 is displayed above the cursor.

You will notice that as soon as you specify the endpoint of the line, a mirror image is automatically created on the right of the mirror line. The line drawn as the mirrored entity is merged with the line drawn on the left. Therefore, the entire line becomes a single entity. Remember that the lines will merge only if one of the endpoints of the line drawn is coincident with the mirror line.

4. Move the cursor vertically upward and click to specify the endpoint of the line when the length of the line above the cursor shows the value 30.

You will notice that as soon as you specify the endpoint of the line, a mirror image is created automatically on the other side of the mirror line.

5. Move the cursor horizontally toward the right and click to specify the endpoint of the line when the length of the line above the cursor shows the value 50; a mirror image is created automatically on the other side of the mirror line.

6. Move the cursor vertically upward and click to specify the endpoint of the line when the length of the line above the cursor shows the value 70; a mirror image is created automatically on the other side of the mirror line.
7. Move the cursor horizontally toward the right and click to specify the endpoint of the line when the cursor snaps to the centerline; a mirror image is created automatically on the other side of the mirror line. The sketch after drawing the lines is shown in Figure 3-15.
8. Right-click in the drawing area to display a shortcut menu. Next, choose the **Select** option from the shortcut menu to exit the **Line** tool.

Note that the symmetrical symbol is still displayed on both ends of the centerline. It indicates that the dynamic mirror option is still activated. Since this tool is activated, you will create left side circle of the sketch and the right side circle will be created automatically.

9. Choose the **Circle** tool from the **Circle** flyout in the **Sketch CommandManager**; the **Circle PropertyManager** is displayed, as shown in Figure 3-16.

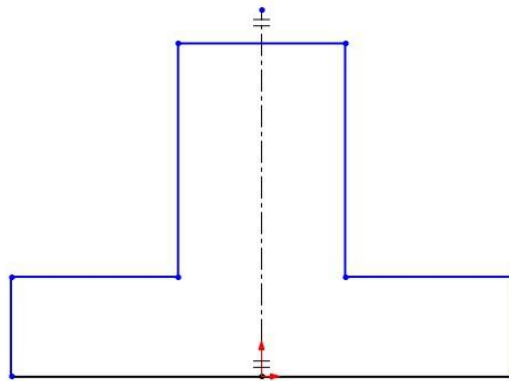


Figure 3-15 Sketch after drawing lines

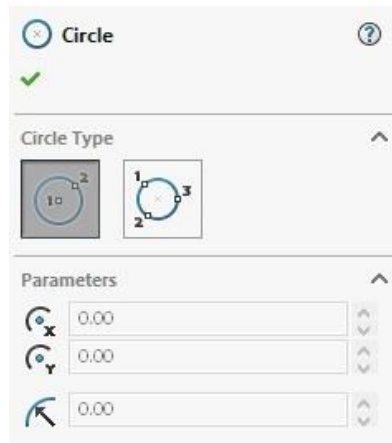


Figure 3-16 *The Circle PropertyManager*

10. Move the cursor to the location whose coordinates are -55 mm, 15 mm, 0 mm and specify the center point of the circle at this location.
11. Move the cursor horizontally toward right to draw a circle and click on the left mouse button when the radius of the circle above the cursor shows 10; a circle of 10 mm radius is drawn. Also, a mirror image is created automatically on the right side of the mirror line, refer to Figure 3-17.

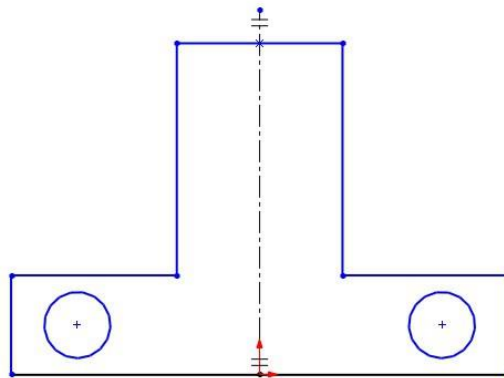


Figure 3-17 *Sketch after drawing circles*

12. Exit the **Circle** tool by pressing the ESC key or choosing the **Select** option from the shortcut menu.

Now, you need to disable the dynamic mirroring.

13. Right-click in the drawing area and then choose **Recent Commands > Dynamic Mirror Entities** from the shortcut menu to disable dynamic mirroring.

Filleting Sketched Entities

Now, you need to fillet the sketched entities by using the **Sketch Fillet** tool.

1. Choose the **Sketch Fillet** tool from the **Sketch CommandManager**; the **Sketch FilletPropertyManager** is displayed, as shown in Figure 3-18. Also, you are prompted to select the sketched entities or sketched vertex to be filleted.

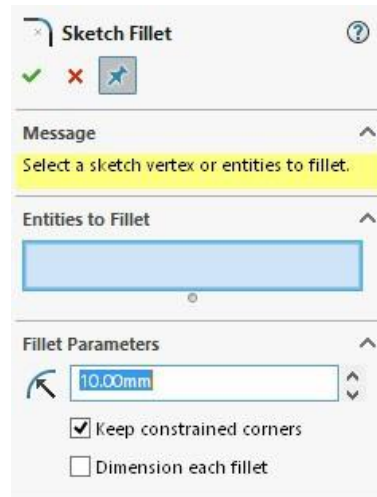


Figure 3-18 The Sketch Fillet PropertyManager

The **Sketch Fillet** tool is used to create a fillet or a tangent arc at the intersection of two sketched entities. It trims or extends the entities to be filleted, depending on the geometry of the sketched entity. You can apply a fillet to two nonparallel lines, two arcs, two splines, an arc and a line, a spline and a line, or a spline and an arc. A fillet between two arcs and a line depends on the compatibility of the geometry to be extended or filleted along the given radius.

2. Select the first set of entities to be filleted, refer to Figure 3-19; the preview of the fillet with the default radius is displayed. Next, select the second, third, and fourth set of entities to be filleted one by one, refer to Figure 3-19.
3. Set the value of the **Fillet Radius** spinner in the **Fillet Parameters** rollout of the PropertyManager to **10**.
4. Ensure that the **Dimension each fillet** check box in the **Fillet Parameters** rollout of the PropertyManager is cleared.

If the **Dimension each fillet** check box is selected, it will apply dimensions individually to all set of entities to be filleted. Therefore, you can control the dimension of each entities of the selected set individually and create multiple fillets of different radii. However, if this check box is cleared, the dimensions will be applied to the last selected entity only and equal radii relation with the other entities of the selected set to be filleted. Therefore, while modifying the radii of the first set of entities, the radii of all the other set of entities will be modified automatically.

Tip

You can also select the vertex formed at the intersection of the two entities to be filleted.

5. Choose the **OK** button twice from the **Sketch Fillet PropertyManager** to exit from it. The sketch after creating the fillets is shown in Figure 3-20.

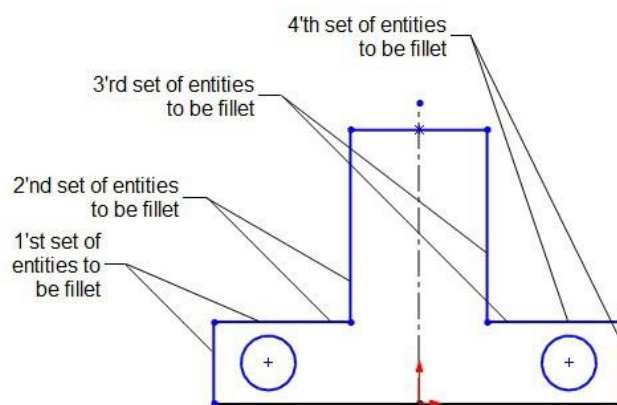


Figure 3-19 Sets of entities to be filleted

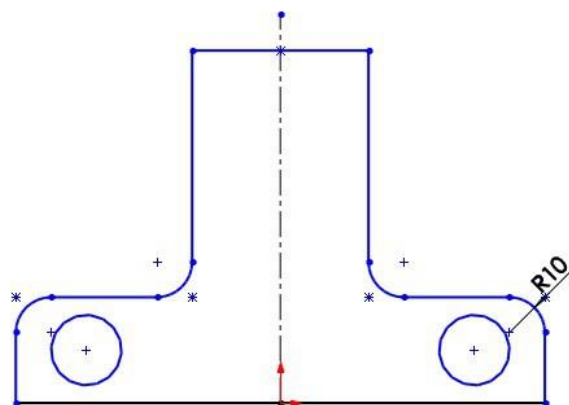


Figure 3-20 Sketch after creating the fillets

Chamfering Sketched Entities

Now, you need to chamfer the sketched entities by using the **Sketch Chamfer** tool.

1. Choose the **Sketch Chamfer** tool from the **Sketch Fillet** flyout in the **Sketch CommandManager**; the **Sketch Chamfer PropertyManager** is displayed, as shown in Figure 3-21.

The **Sketch Chamfer** tool is used to apply a chamfer to the adjacent sketched entities at the point of intersection. A chamfer can be specified by two lengths or angle and length from the point of intersection. You can apply a chamfer between two nonparallel lines that may be intersecting or non-intersecting. The creation of a chamfer between two non-intersecting lines depends on the length of the lines and the chamfer distance.

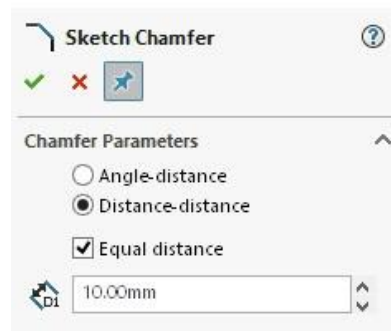


Figure 3-21 The Sketch Chamfer PropertyManager

2. Make sure that the **Distance-distance** radio button and **Equal distance** check box are selected in the **Chamfer Parameters** rollout of the PropertyManager.

The **Distance-distance** radio button allows you to create chamfer by specifying the distance value. If the **Equal distance** check box is selected, only the **Distance 1** spinner is available in the PropertyManager to specify the distance value for creating chamfer. The distance entered in this spinner will also be applied as the distance of the second direction for creating chamfer. However, if you clear this check box, it allows you to specify two different distances for creating a chamfer. As soon as you clear the **Equal distance** check box, the **Distance 2** spinner will also be displayed below the **Distance 1** spinner of the **Sketch Chamfer PropertyManager** to set the value of the distance in the second direction.

Tip

You can also create a chamfer by specifying the angle and distance values. To do so, you need to select the **Angle-distance** radio button from the **Sketch Chamfer PropertyManager**. When you select this radio button, the **Distance 1** and **Distance 1 Angle** spinners will be activated in the PropertyManager where you can specify the distance and angle value for creating chamfer.

3. Set the value of the **Distance 1** spinner to **10**. Next, select the first set of entities to be chamfered, refer to Figure 3-22; the **SOLIDWORKS** message window is displayed with the message that **At least one sketch constraint is about to be lost. Chamfer anyway?**
4. Choose the **Yes** button in the message window; the chamfer between the selected set of entities is created.
5. Select the second set of entities to be chamfered and then exit the PropertyManager by choosing the **OK** button. Figure 3-23 shows the final sketch after creating the chamfers.

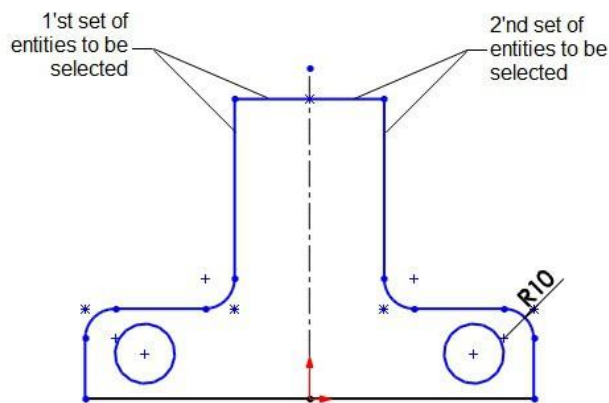


Figure 3-22 Entities to be selected for chamfering

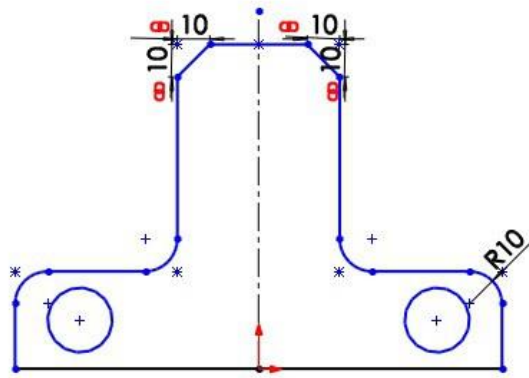


Figure 3-23 Final sketch after creating chamfers

Saving the Sketch

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box.
 2. Browse to the *c03* folder and enter the name of the document as **c03_tut02** in the **File name** edit box and choose the **Save** button.
 3. Press CTRL+W to close the file.
-

Tutorial Exercise 3

In this tutorial, you will create the base sketch of the model shown in Figure 3-24. The sketch of the model is shown in Figure 3-25. The dimensions in Figure 3-25 are given for your reference only. (**Expected time: 30 min**)

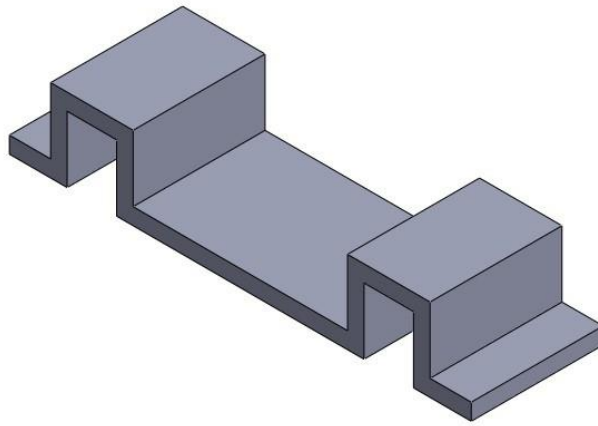


Figure 3-24 Solid model for Tutorial 3

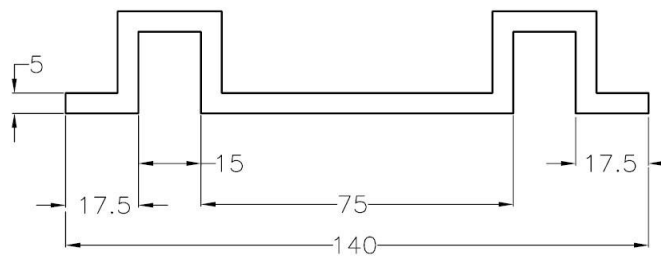


Figure 3-25 Sketch for Tutorial 3

The following steps are required to complete this tutorial:

- Start SOLIDWORKS and then a new part document.
- Invoke the sketching environment.
- Modify the snap, grid, and units settings.
- Draw the sketch, refer to Figure 3-26.
- Offset the entire sketch bidirectionally, refer to Figure 3-28.
- Save the sketch.

Starting SOLIDWORKS and then a New SOLIDWORKS Document

1. Start SOLIDWORKS and then invoke the **Part** environment by choosing the **Part**

button from the **New SOLIDWORKS Document** dialog box.

Invoking the Sketching Environment

Next, you need to invoke the sketching environment by selecting the **Front Plane** as the sketching plane.

1. Choose the **Sketch** tab from the **CommandManager**. Next, choose the **Sketch** tool from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is invoked and you are prompted to select the plane to create the sketch.
2. Select the **Front Plane**; the sketching environment is invoked and the plane is oriented normal to the view.

Modifying the Snap, Grid, and Unit Settings

Before drawing the sketch, you need to modify the grid and snap settings to make the cursor jump through a distance of 5 mm.

1. Invoke the **Document Properties - Grid/Snap** dialog box and then set the value as **50** and **10** in the **Major grid spacing** and **Minor-lines per major** spinners, respectively.
2. Make sure that the **Grid** check box is selected in the **System Option - Relations/Snaps** dialog box. Also, make sure that the unit system is set to **MMGS (millimeter, gram, second)** in the **Document Properties - Units** dialog box.
3. After making the necessary settings, choose the **OK** button from the dialog box.

The coordinates displayed close to the lower right corner of the SOLIDWORKS window show an increment of 5 mm when you move the cursor in the drawing area after exiting the dialog box.

Drawing the Sketch

Next, you need to draw the sketch using the sketch tools. In this tutorial, you will create the sketch by using the **Line** and **Offset Entities** tools. First, you will create a symmetric line for the sketch and then offset it bidirectionally. After offsetting the symmetric line, you need to convert it into a construction line.

1. Invoke the **Line** tool and specify the start point of the line at the origin. Next, move the cursor horizontally toward right and specify the end point of the line when the length of the line displayed above the cursor is 15.
2. Move the cursor vertically upward and specify the end point of the line when the length of the line displayed above the cursor is 20.
3. Move the cursor horizontally toward the right and specify the end point of the line when the length of the line displayed above the cursor is 20.
4. Move the cursor vertically downward and specify the end point of the line when the length of the line displayed above the cursor is 20.
5. Move the cursor horizontally toward the right and specify the end point of the line when the length of the line displayed above the cursor is 70.
6. Move the cursor vertically upward and specify the end point of the line when the length of the line displayed above the cursor is 20.
7. Move the cursor horizontally toward the right and specify the end point of the line when the length of the line displayed above the cursor is 20.
8. Move the cursor vertically downward and specify the end point of the line when the length of the line displayed above the cursor is 20.
9. Move the cursor horizontally toward the right and specify the end point of the line when the length of the line displayed above the cursor is 15. Next, exit the **Line** tool. Figure 3-26 shows the sketch after creating its symmetric line.



Figure 3-26 The sketch after creating its symmetric line

Offsetting the Entities

Next, you need to offset the sketch in both directions of the sketched entities using the **Offset Entities** tool.

1. Choose **Offset Entities** tool from the **Sketch CommandManager**; the **Offset Entities PropertyManager** is displayed, as shown in Figure 3-27. Also, you are prompted to select faces, edges, or curves to offset.

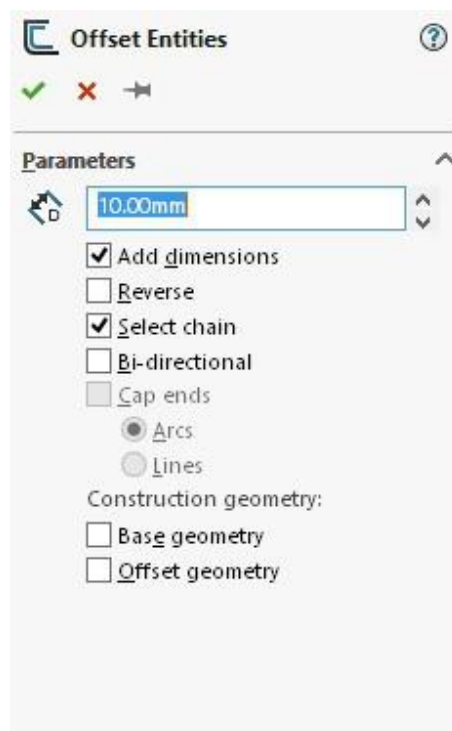


Figure 3-27 The Offset Entities PropertyManager

The **Offset Entities** tool is used to offset selected entities such as sketches, edges, or faces by a specified distance. This tool cannot be used on fit splines, already offset splines, or if the resultant is an intersecting geometry.

2. Set the value of the **Offset Distance** spinner to **2.5** and then press ENTER.
3. Make sure that the **Select chain** check box is selected in the **Parameters** rollout of the PropertyManager.

The **Select chain** tool is used to select the entire chain of continuous sketched entities that are in contact with the selected entity. When you invoke the **Offset Entities PropertyManager**, the **Select chain** check box is selected by default. If you clear this check box, only the selected sketched entity will be offset.

4. Select any one entity of the sketch from the drawing area; the entire sketch gets selected and the preview of the offset sketch is displayed in the drawing area. Note that, by default, the offset is in one direction.

Now, you need to change the direction of the offset to both sides of the sketched entities.

5. Select the **Bi-directional** check box from the **Parameters** rollout of the **Offset Entities PropertyManager**; the **Cap ends** check box is enabled in the PropertyManager. Also, the preview of the offset sketch is modified accordingly.

The **Bi-directional** check box is used to create the offset of a selected entity in both directions of the sketched entity. If you select this check box, the **Reverse** check box that is used to flip the direction of the offset entity will not be displayed in the **Parameters** rollout of the PropertyManager.

As is evident from Figure 3-25 that the ends of the sketch are closed with lines. Therefore, you need to close the ends of the sketch.

6. Select the **Cap ends** check box from the **Parameters** rollout of the PropertyManager; the **Arcs** and **Lines** radio buttons are enabled below the check box.

The **Cap ends** check box is available only if the **Bi-directional** check box is selected. This check box is used to close the ends of the bidirectionally offset entities. The **Arcs** and **Lines** radio buttons are used to specify the type of cap required to close the ends.

7. Select the **Lines** radio button from the **Parameters** rollout of the PropertyManager; the

preview of the offset sketch is modified accordingly.

As discussed earlier, you need to convert the parent sketch into a construction sketch.

8. Select the **Base geometry** check box under **Construction geometry** in the **Parameters** rollout of the PropertyManager.

The **Base geometry** check box is used to convert the parent entity into a construction entity.

9. Choose the **OK** button from the PropertyManager. The final sketch is shown in Figure 3-28.



Figure 3-28 The final sketch

Saving the Sketch

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box.
2. Browse to the *c03* folder and enter the name of the document as **c03_tut03** in the **File name** edit box and choose the **Save** button.

The document is saved at the location */Documents/SOLIDWORKS Tutorials/c03*.

3. Close the file by choosing **File > Close** from the SOLIDWORKS menus.