

Tutorial 2

Drawing Sketches for Solid Models

Learning Objectives

After completing this tutorial, student will be able to:

- *Understand the importance of sketching environment*
- *Open a new part document*
- *Understand various terms used in the sketching environment*
- *Use various sketching tools*
- *Use the drawing display tools*
- *Delete sketched entities*

THE SKETCHING ENVIRONMENT

Most of the products designed by using SOLIDWORKS are a combination of sketched, placed, and derived features. The placed and derived features are created without drawing a sketch, but the sketched features require a sketch to be drawn first. Generally, the base feature of any design is a sketched feature and is created using the sketch. Therefore, while creating any design, the first and foremost requirement is to draw a sketch for the base feature. Once you have drawn the sketch, you can convert it into the base feature and then add the other sketched, placed, and derived features to complete the design. In this tutorial, you will learn to create the sketch for the base feature using the various sketching tools.

In general terms, a sketch is defined as the basic contour of a feature. For example, the solid model of a spanner shown in Figure 2-1.



Figure 2-1 Solid model of a spanner

This spanner consists of a base feature, cut feature, mirror feature (cut on the back face), fillets, and an extruded text feature. The base feature of this spanner is shown in Figure 2-2. It is created using a single sketch drawn on the **Front Plane**, as shown in Figure 2-3. This sketch is drawn in the sketching environment using various sketching tools. Therefore, to draw the sketch of the base feature, first you need to invoke the sketching environment where you will draw the sketch.



Figure 2-2 Base feature of the Spanner

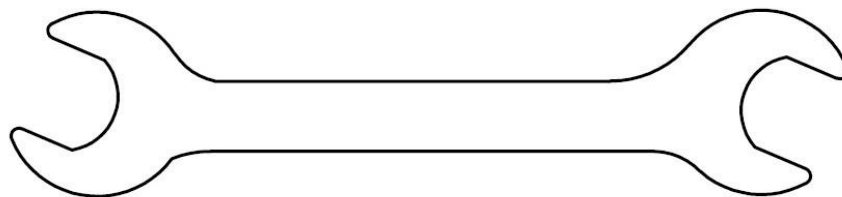


Figure 2-3 Sketch for the base feature of the Spanner

Note

*The sketcher environment of SOLIDWORKS can be invoked at any time in the **Part** or **Assembly** mode. You will learn more about invoking sketcher environment later .*

Tutorial

Tutorial Exercise 1

In this tutorial, you will start a new part document in SOLIDWORKS 2016 and then invoke the sketcher environment to draw the sketch of the solid model shown in Figure 2-4. The sketch of the model is shown in Figure 2-5. While drawing the sketch, you will modify the Snap, Grid, and Units settings for the active document. Note that the solid model and dimensions given in the Figures 2-4 and 2-5 are for your reference only. You will learn to apply dimensions and creating solid models in the later tutorials. **(Expected time: 30 min)**

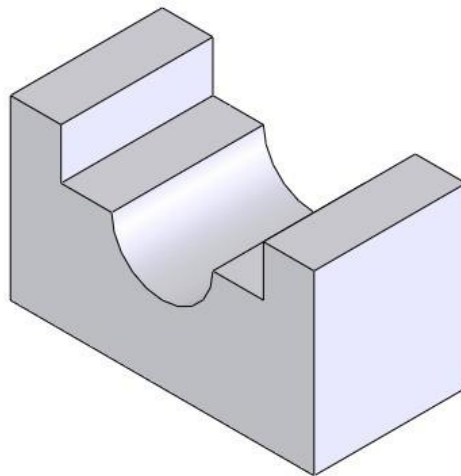


Figure 2-4 Solid model for Tutorial 1

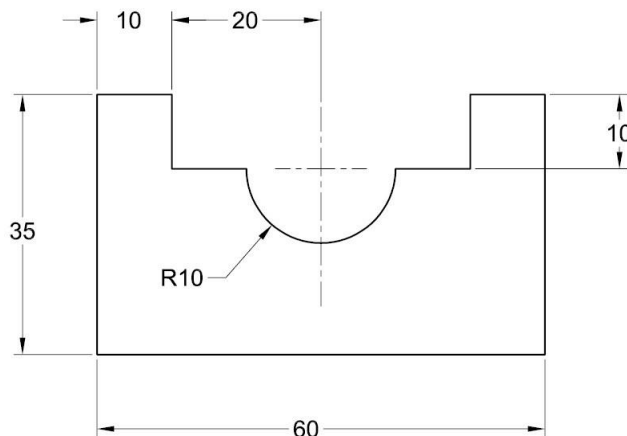


Figure 2-5 Sketch for Tutorial 1

The following steps are required to complete this tutorial:

- a. Start SOLIDWORKS 2016 and then start a new part document.
- b. Invoke the sketching environment.
- c. Modify the settings of the snap, grid, units so that the cursor jumps through a distance of 5 mm.
- d. Draw the sketch using the **Line** tool, refer to Figure 2-15.
- e. Save the sketch and then close the file.

Starting SOLIDWORKS 2016

Once you have installed the SOLIDWORKS 2016, an icon is displayed on the windows desktop and a folder is added to the **Start** menu.

1. Choose **Start > SOLIDWORKS 2016** from the **Start** menu or double-click on the **SOLIDWORKS 2016** icon on the desktop of your computer; the SOLIDWORKS 2016 window will be displayed. If you are starting the SOLIDWORKS application for the first time after installing it, the **SOLIDWORKS License Agreement** dialog box will be displayed, as shown in Figure 2-6. Choose the **Accept** button from the dialog box; the **SOLIDWORKS 2016** interface window will be displayed, as shown in Figure 2-7.

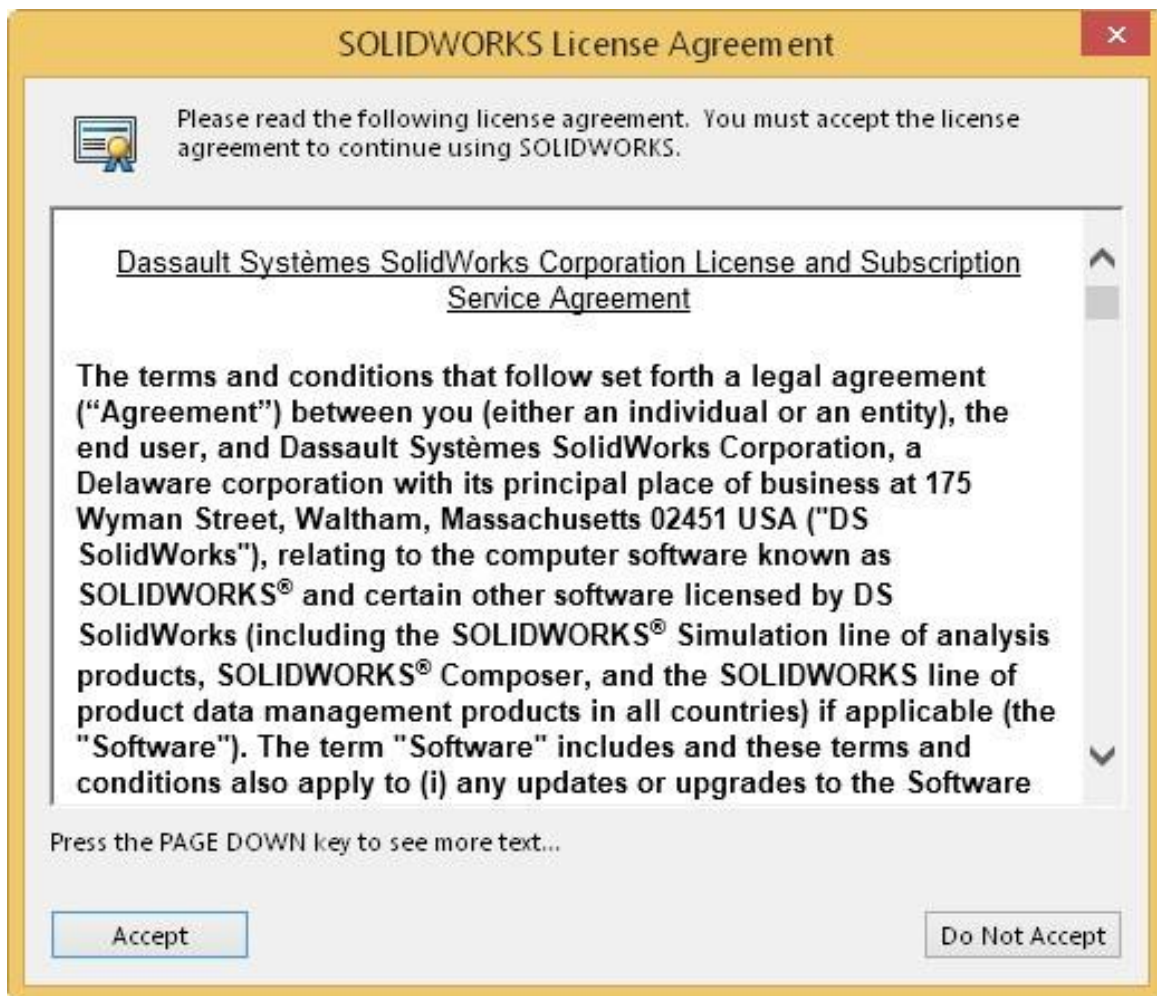


Figure 2-6 The SOLIDWORKS License Agreement dialog box

The initial interface of the **SOLIDWORKS 2016** window consists of SOLIDWORKS menus, Menu Bar, and Task Pane, refer to Figure 2-7.

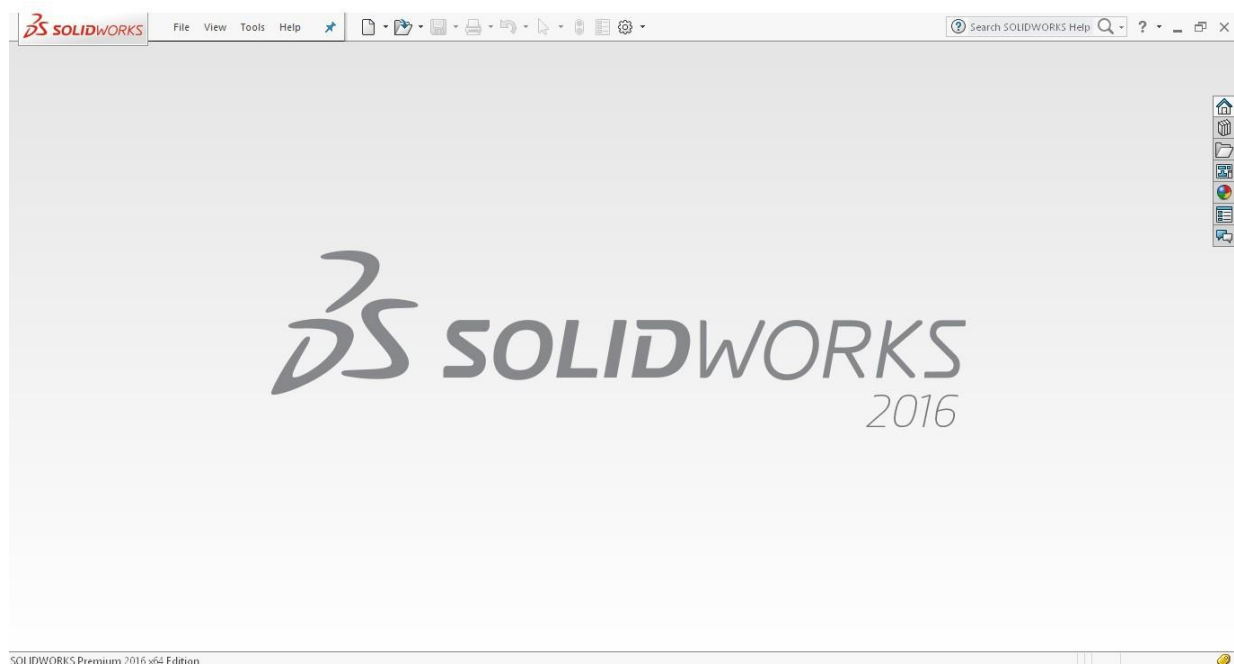


Figure 2-7 The SOLIDWORKS window

The Task Pane is displayed on the right in the **SOLIDWORKS 2016** window. These Task Pane contain various options that are used to start a new file, open an existing file, browse the related links of SOLIDWORKS, and so on. Various Task Pane in SOLIDWORKS are discussed next.

SOLIDWORKS Resources Task Pane

By default, the **SOLIDWORKS Resources** task pane is displayed when you start a SOLIDWORKS session. Different options available in this task pane are discussed next.

Getting Started Rollout: The options in this rollout are used to start a new document, open an existing document, learn the new features in this release of SOLIDWORKS and invoke the interactive help topics. If you are a new user of SOLIDWORKS, choose the **Introducing SOLIDWORKS** option to get an overview of SOLIDWORKS.

SOLIDWORKS Tools Rollout: The options in this rollout are SOLIDWORKS files, SOLIDWORKS Rx, Compare My Score, Copy Settings Wizard and My Products. These options are used to control custom properties and attributes of the files. SOLIDWORKS Rx can be used to help diagnose issues with your computer.

Community Rollout: The options in this rollout are used to invoke various SOLIDWORKS communities such as customer portal, discussion forum, user groups, and so on.

Online Resources Rollout: The options in this rollout are used to invoke the discussion forum of SOLIDWORKS, subscription services, partner solutions, manufacturing network, and print 3D websites.

Subscription Services Rollout: The options in this rollout are used to access various software releases, service packs, and value added resources provided by SOLIDWORKS Corporation and its global network.

Tip of the Day Message Box: The **Tip of the Day** message box provides you with a useful tip that helps you make the full utilization of tools available in SOLIDWORKS. Click on the Next Tip text provided at the lower right corner of the Tip of the Day message box to view the next tip.

Design Library Task Pane

The **Design Library** task pane is invoked by choosing the **Design Library** tab from the Task Panes. This task pane is used to browse the default **Design Library** and the toolbox components available in SOLIDWORKS. Also, it allows you to access the **3D ContentCentral** website. To access the toolbox components, you need to add **Toolbox Add-Ins** in your computer. To do so, choose **Tools > Add-Ins** from the SOLIDWORKS menu; the **Add-Ins** dialog box will be displayed. Select the **SOLIDWORKS Toolbox Library** check box in this dialog box and then choose **OK**. To access the **3D ContentCentral** website, your computer needs to be connected to the Internet.

File Explorer Task Pane

The **File Explorer** task pane is used to explore the files and folders that are saved in the hard disk of your computer.

View Palette Task Pane

The **View Palette** task pane is used to drag and drop the drawing views into a drawing sheet.

Appearances, Scenes, and Decals Task Pane

The **Appearances, Scenes, and Decals** task pane is used to change the appearance of models or drawing display area. On choosing the **Appearances, Scenes, and Decals** tab from the Task Panes; the **Appearances, Scenes, and Decals** task pane will be invoked with three nodes: **Appearances(color)**, **Scenes**, and **Decals**. The **Appearances(color)** node is used to change the appearance of a model. The **Scenes** node is used to change the background of the drawing area and the **Decals** node is used to apply decals to a model.

Custom Properties Task Pane

The **Custom Properties** task pane is displayed on choosing the **Custom Properties** tab from the Task Panes. This task pane is used to view the properties of the files.

In SOLIDWORKS, the tools that are in the **Standard** toolbar are also available in the Menu Bar, as shown in Figure 2-8. This toolbar is available above the drawing area. When you move the cursor over the SOLIDWORKS logo at the top left corner of the display area, the SOLIDWORKS menus will be displayed as a cascading menu, as shown in Figure 2-9. You can also fix this menu by choosing the push-pin button.



Figure 2-8 The Menu Bar



Figure 2-9 The SOLIDWORKS menus

Starting a New Part Document in SOLIDWORKS 2016

1. Select the **New Document** option from **Getting Started** rollout of the **SOLIDWORKS Resources** task pane to start a new part document in SOLIDWORKS 2016; the **New SOLIDWORKS Document** dialog box will be displayed, as shown in Figure 2-10. You can also invoke this dialog box by choosing the **New** button from the Menu Bar.
2. Make sure that the **Part** button is chosen in the **New SOLIDWORKS Document** dialog box. Next, choose the **OK** button to invoke the **Part** mode, refer to Figure 2-11.

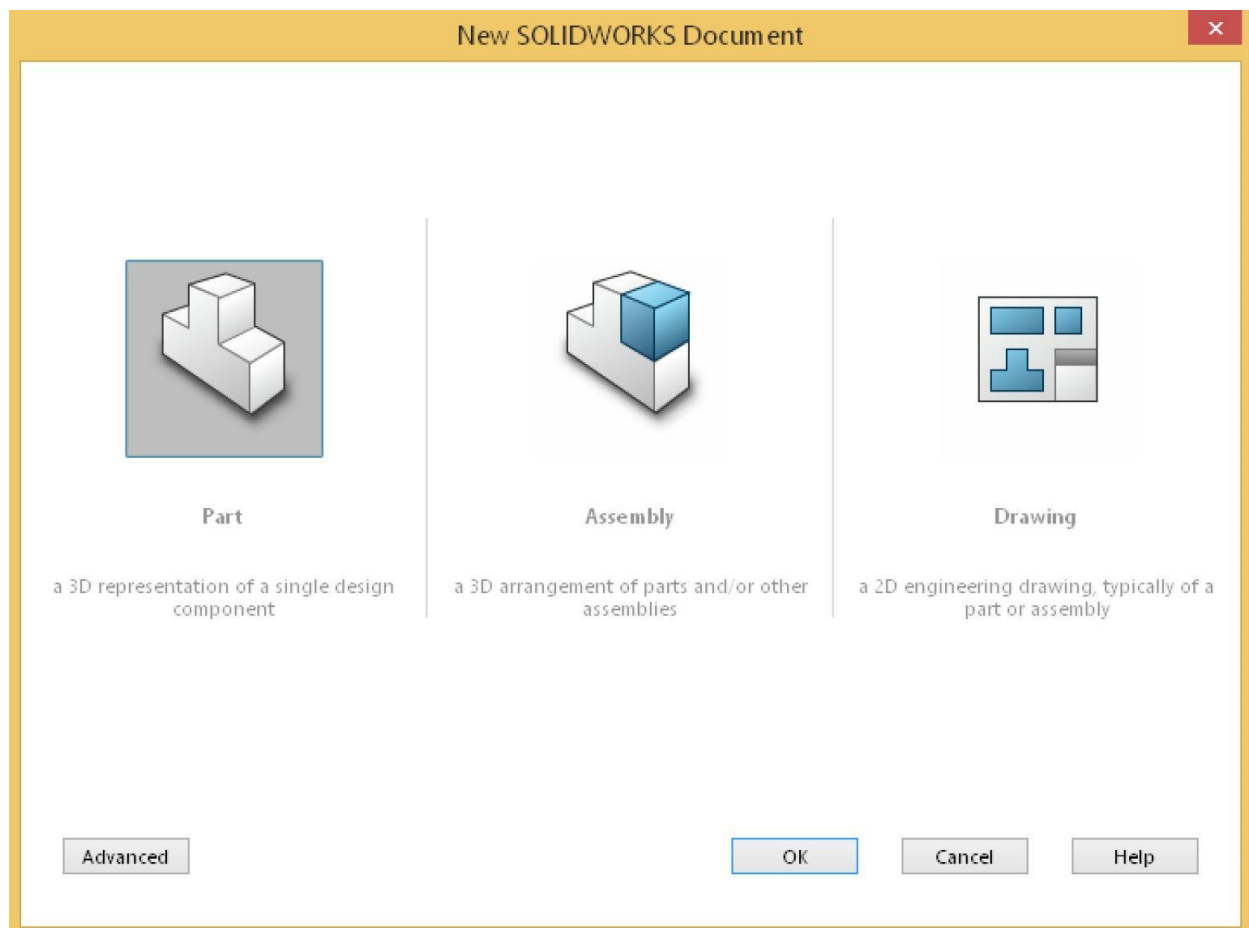


Figure 2-10 *The New SOLIDWORKS Document dialog box*

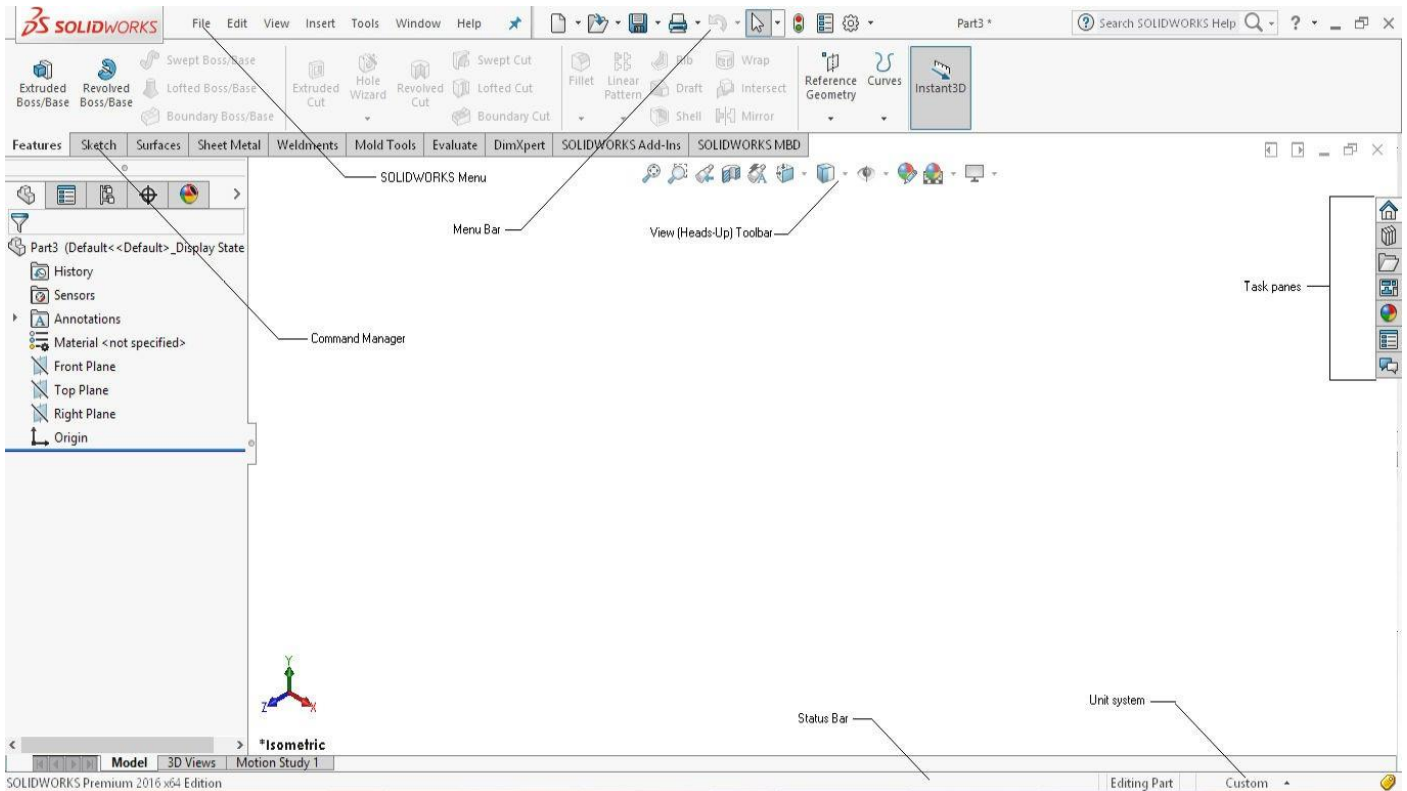


Figure 2-11 *The initial interface displayed on invoking the **Part** document*

Note

To start a new assembly document, you need to choose the **Assembly** button and then the **OK** button from the **New SOLIDWORKS Document** dialog box. In an assembly document, you can assemble the components created in the part documents. You can also create components in the assembly documents. You will learn more about the assembly document in the later tutorials.

To start a new drawing document, you need to choose the **Drawing** button and then the **OK** button from the **New SOLIDWORKS Document** dialog box. In a drawing document, you can generate or create different drawing views of the parts created in the part documents or the assemblies created in an assembly document. You will learn more about drawing document in the later tutorials.

Whenever you start a new part document, by default, you are in the part modeling environment, but you need to start the design by first creating the sketch of the base feature. You need to invoke the sketcher environment to create the sketch of the model shown in the Figure 2-4.

3. Click on the **Sketch** tab of the **CommandManager** to display the **Sketch CommandManager** tools.
4. Choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is displayed on the left in the graphic area and you are prompted to select the sketching plane on which the sketch is to be created. Also, the three default planes (**Front Plane**, **Right Plane**, and **Top Plane**) are available in SOLIDWORKS, and are temporarily displayed on the screen, as shown in Figure 2-12.

You can select a plane to draw the sketch of the base feature depending on the requirement of the design. The selected plane will automatically be oriented normal to the view, so that you can easily create the sketch. Also, the **CommandManager** will display various sketching tools to draw the sketch.

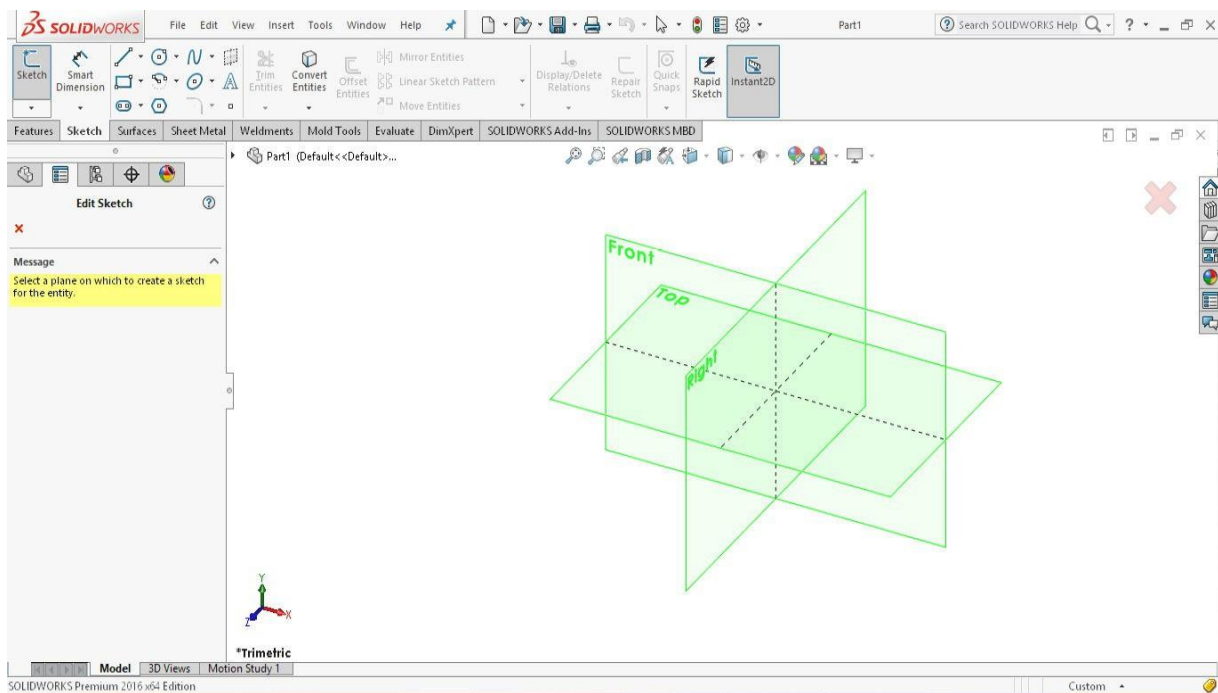


Figure 2-12 The three default planes displayed on the interface

5. Select the **Front Plane** from the graphics area; the sketching environment is invoked and the selected plane gets oriented normal to the view. You will notice that the red colored arrows are displayed at the center of the interface window, indicating that you are in the sketching environment. Also, the confirmation corner with the **Exit Sketch** and **Cancel Sketch** options at the upper right corner in the drawing area is displayed. The window display in the sketching environment of SOLIDWORKS is shown in Figure 2-13.

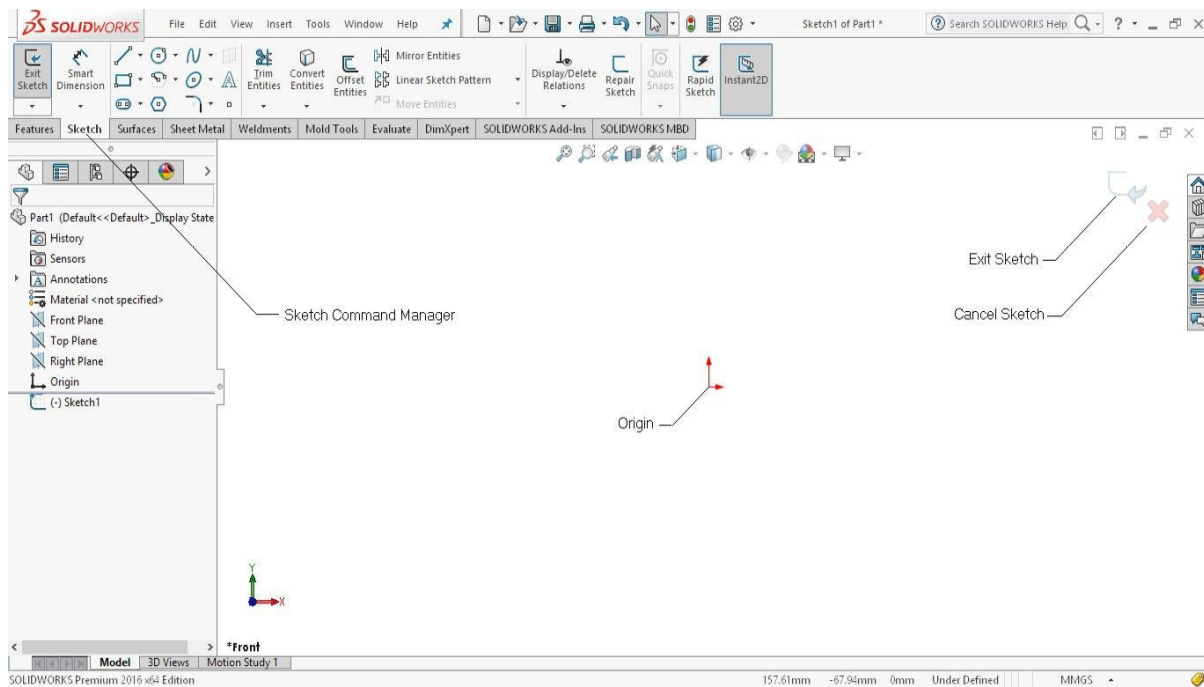


Figure 2-13 Default interface display of a part document in the sketching environment

Modifying the Snap, Grid, and Units 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 5 mm.

1. Choose the **Options** button from the Menu Bar; the **System Options - General** dialog box is displayed.
2. Choose the **Document Properties** tab; the name in the dialog box changes to the **Document Properties - Drafting Standard**.

Note

If you have selected millimeters as the unit of measurement while installing SOLIDWORKS, skip steps 3 and 4 in this section.

3. Select the **Units** option from the area on the left of the dialog box; the options related to linear and angular units are displayed.

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

Tip

*In SOLIDWORKS, you can also change the units for the current document by using the **Unit system** that is located towards right in the Status Bar, refer to Figure 2-11. To do so, click on the **Unit system**; a flyout will be displayed with a tick mark on the left of the activated unit system, refer to Figure 2-14. Now, you can select the required unit system for the activated unit system from this flyout. You can also invoke the **Document Properties-Units** dialog box by selecting the **Edit Document Units** from this flyout.*

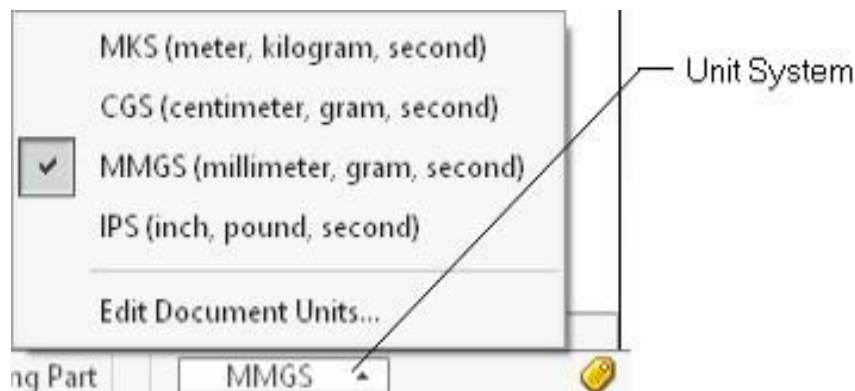


Figure 2-14 The flyout showing the currently selected unit system

Note

*Note that the short form of the current unit system of the activated document will be displayed in the **Status Bar**. For example, if the **MMGS (millimeter, gram, second)** is the current unit system then **MMGS** will be displayed in the **Status Bar**.*

As evident from Figure 2-5, the dimensions in the sketch are multiple of 5. Therefore, you need to modify the grid and snap settings so that the cursor jumps through a distance of 5 mm.

5. Select the **Grid/Snap** option from the area on the left in the dialog box; the options related to the grid and snap are displayed, also the name of the dialog box is modified to **Document Properties - Grid/Snap**.
6. In this dialog box, set the value in the **Major grid spacing** and **Minor-lines per major** spinners to **50** and **10**, respectively.

Note

*The distance through which the cursor jumps depends on the ratio between the values in the **Major grid spacing** and **Minor-lines per major** spinners available in the **Grid** area. For example, if you want the coordinates to increment by 10 mm, you will have to set the ratio of the major and minor lines to 10. This can be done by setting the value of the **Major grid spacing** spinner to 100 and the **Minor-lines per major** spinner to 10. Similarly, to make the cursor jump through a distance of 5 mm, set the value of the **Major grid spacing** spinner to 50 and the **Minor-lines per major** spinner to 10.*

Tip

*If you want to display the grid in the sketching environment, select the **Display grid** check box from the **Grid** area of the **Document Properties - Grid/Snap** dialog box. Alternatively, choose **Hide/Show Items > View Grid** from the **View (Heads-Up)** toolbar.*

While drawing a sketched entity by snapping through grips, the grips symbol will be displayed below the cursor on the right.



7. Make sure that the **Grid** check box is selected in the **System Options - Relations/Snaps** dialog box. To invoke this dialog box, choose the **Go To System Snaps** button from the **Document Properties - Grid/Snap** dialog box.

8. Choose the **OK** button to exit the dialog box.

Now, when you move the cursor, the coordinates displayed close to the lower right corner of the drawing area show an increment of 5 mm.

Drawing the Sketch

The sketch will be drawn using the **Line** tool. The arc in the sketch will also be drawn using the same tool. You need to start the drawing from the lower left corner of the sketch to be drawn.

1. Choose the **Line** tool from the **Sketch CommandManager**; the arrow cursor is changed into line cursor. Also, the **Insert Line PropertyManager** is displayed at the left in the drawing area.

The line cursor is actually a crosshairs-like cursor with a small inclined line below the crosshairs. You can also invoke the **Line** tool by pressing the L key.

The line is the basic sketching entity available in SOLIDWORKS. In general terms, a line is defined as the shortest distance between two points.

In SOLIDWORKS, the **Line** tool is used to draw a chain of continuous lines which is a default method of drawing lines. In this method, you have to specify the start point and the endpoint of the line using the left mouse button. As soon as you specify the start point of the line, the **Insert Line PropertyManager** will disappear and the **Line Properties PropertyManager** will be displayed. However, the options available in this PropertyManager are not activated at this stage. These options will be activated after the line is created. As soon as, you specify the end point of the line by using the left mouse button, a line will be drawn between two points. Now, when you move the cursor away from the end point of the line, you will notice that another line is attached with the cursor. It means that you can create a chain of continuous line one after other. You can end the process of drawing the continuous line by pressing the ESC key, by double-clicking on the screen, or by invoking the **Select** tool from the Menu Bar. You can also right-click to display the shortcut menu and choose the **End chain** or **Select** option to terminate the process of drawing the line.

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 ends but the **Line** tool still remains active. As a result, you can draw other lines. However, if you choose **Select** from the shortcut menu, the **Line** tool will be deactivated.*

You can also draw an individual line by using the **Line** tool. To do so, invoke the **Line** tool. Next, press and hold the left mouse button to specify the start point of the line and then drag the cursor toward the required direction. After getting the required direction and length, release the left mouse button.

2. Move the line cursor to a point whose coordinates are 30 mm, 0 mm, and, 0 mm.
3. Click at this point to specify the start point of the line; the **Line Properties PropertyManager** will be displayed.
4. Move the cursor horizontally toward the right. Click again when the length of the line

above the line cursor displays 60; a bottom horizontal line of 60 mm length is drawn.

5. Move the line cursor vertically upward and click when the length of the line above the line cursor displays 35.
6. Choose the **Zoom to Fit** button from the **View (Heads-Up)** toolbar to fit the sketch into the screen.

Note

You can invoke the drawing display tools when any other tool is active in the drawing area. After modifying the drawing display area, the tool that was active before invoking the drawing display tool will be restored and you can continue using that tool.

7. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line above the line cursor shows the value 10.
8. Move the line cursor vertically downward and click when the length of the line above the line cursor is displayed as 10.
9. Move the line cursor horizontally toward the left and click when the length of the line above the line cursor is displayed as 10.

Next, you need to draw an arc normal to the last line. As mentioned earlier, you can also draw an arc using the **Line** tool. Drawing arcs using the **Line** tool is a recommended method to draw a sketch that is a combination of lines and arcs. This increases the productivity by reducing the time taken in invoking tools for drawing an arc and then invoking the **Line** tool to draw lines.

10. Move the line cursor away from the endpoint of the last line and then move it back close to the endpoint; the arc mode is invoked. Also, the **Line Properties PropertyManager** disappears and the **Arc PropertyManager** is displayed at the left in the drawing area.
11. Move the arc cursor vertically downward to the next grid point.
12. Move the arc cursor toward the left.

You will notice that a normal arc is being drawn and the angle and radius of the arc are displayed above the line cursor.

13. Move the cursor to the left and click when the angle value on the arc cursor is displayed as 180 and the radius value is displayed as 10. An arc normal to the last line is drawn and again the line mode is invoked automatically.

14. Move the line cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 10.

15. Move the line cursor vertically upward and click when the length of the line on the line cursor is displayed as 10.

16. Move the line cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 10.

17. Move the line cursor to the start point of the first line and click when the orange circle is displayed.

18. Press the ESC key to exit the **Line** tool.

This completes the sketch. However, you need to modify the drawing display area such that the sketch fits the screen.

19. Press the F key; the drawing display area is modified and sketch fits into the screen. The final sketch for Tutorial 1 with the grid display turned off is shown in Figure 2-15.

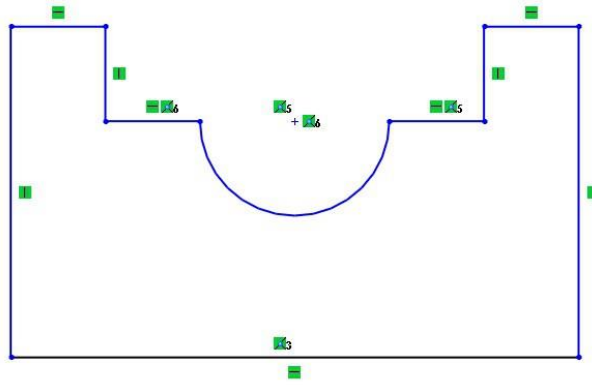


Figure 2-15 Final sketch for Tutorial 1

Note

Figure 2-15 shows the relation symbols that are applied automatically while drawing the sketch. To control the display of relations on and off, choose the **Hide/Show Items** button in the **View (Heads-Up)** toolbar; the flyout will be displayed, refer to Figure 2-16. Next, choose the **View Sketch Relations** button which is a toggle button from this flyout.

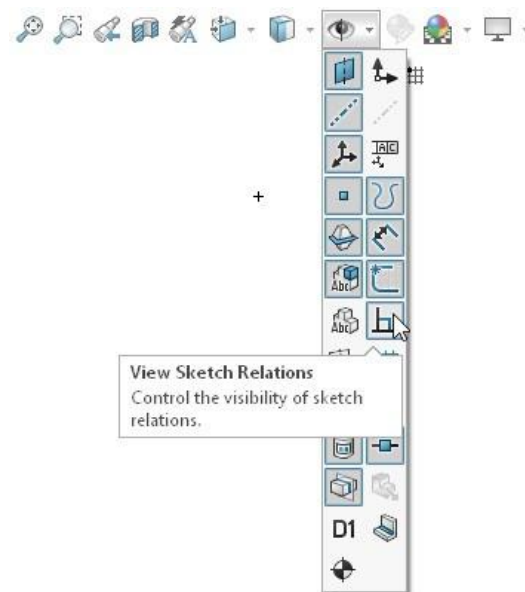


Figure 2-16 The **Hide/Show Items** flyout

Saving the Sketch

It is recommended that you create a separate folder to save the tutorial files of this book. You will create a folder with the name *SOLIDWORKS Tutorials* in the *Documents* folder and then create the sub-folder of each tutorial inside the *SOLIDWORKS Tutorials* folder. Next, you can save the tutorials of a tutorial in the folder of that tutorial.

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box. Create the *SOLIDWORKS Tutorials* folder inside the \Documents folder and then create the *c02* folder inside the *SOLIDWORKS Tutorials* folder.
2. Enter **c02_tut01** as the name of the document in the **File name** edit box. Choose the **Save** button to save the file at the location \Documents\SOLIDWORKS Tutorials\c02.
3. Close the document by choosing **File > Close** from the SOLIDWORKS menus.

Tutorial Exercise 2

In this tutorial, you will draw the basic sketch of the revolved solid model shown in Figure 2-17. The sketch of this model is shown in Figure 2-18. Do not dimension the sketch. The solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

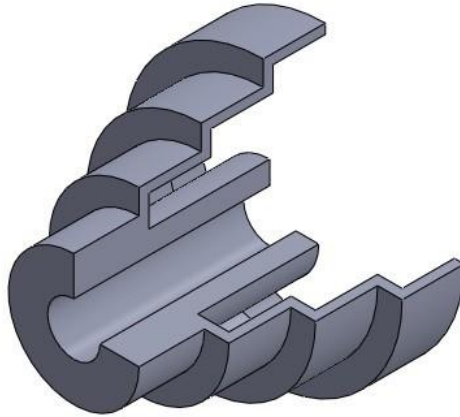


Figure 2-17 Revolved solid model for Tutorial 1

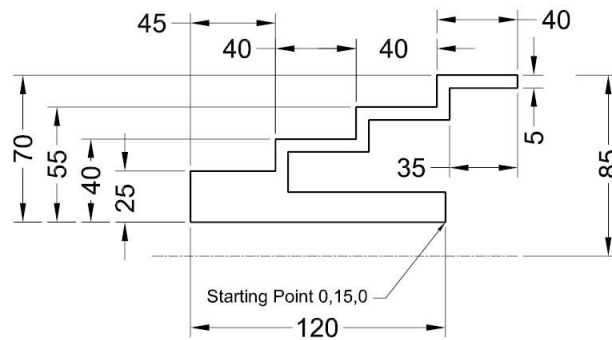


Figure 2-18 Sketch of the revolved solid model

The following steps are required to complete this tutorial:

- Invoke a new part document.
- Invoke the sketching environment.
- Modify the settings of the snap, grid, and units settings so that the cursor jumps through a distance of 5 mm.
- Draw the sketch using the **Line** and **Centerline** tools, refer to Figure 2-21.
- Save the sketch and then close the file.

Invoking a New Part Document

- Choose the **New** button from the Menu Bar to invoke the **New SOLIDWORKS Document** dialog box.
- In the **New SOLIDWORKS Document** dialog box, the **Part** button is chosen by default. Choose the **OK** button in this dialog box; a new SOLIDWORKS part document starts in part modeling environment.

You need to invoke the sketching environment to draw the sketch.

- Choose the **Sketch** tab from the **CommandManager** and then the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is displayed and you are prompted to select a plane on which you want to draw the sketch.

4. Select the **Right Plane** from the drawing area; the sketching environment is invoked and the plane gets oriented normal to the view. You will notice that the red colored arrows are now displayed at the center of the screen indicating that you are in the sketching environment.

Modifying the Snap, Grid, and Units 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 5 mm.

1. Click on the **Unit system** that is located towards right in the Status Bar; a flyout is displayed with a tick mark next to the unit system of the current document, refer to Figure 2-19.

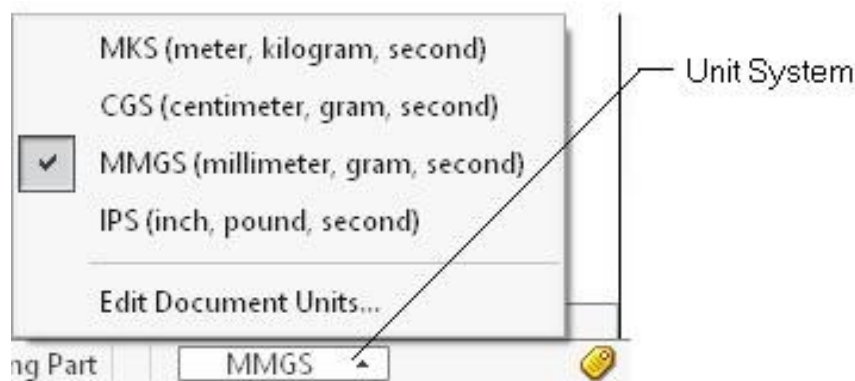


Figure 2-19 The flyout showing the currently selected unit system

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.

3. Select the **MMGS (millimeter, gram, second)** radio button 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 directly for the current document by selecting the respective option from the **unit system** flyout. However, if you are in the sketching environment, on selecting the unit system from the unit system flyout, you will automatically exit the sketching environment.*

As evident from Figure 2-18, the dimensions in the sketch are multiples of 5. Therefore, you need to modify the grid and snap settings so that the cursor jumps through a distance of 5 mm.

4. Select the **Grid/Snap** option from the area on the left of the **Document Properties - Units** dialog box to display the options related to the grid and snap. Also, note that name of the dialog box is modified to **Document Properties - Grid/Snap**.
5. In this dialog box, set the value in the **Major grid spacing** and **Minor-lines per major** spinners to **50** and **10**, respectively.
6. Choose the **Go To System Snap** button from the dialog box; the name of the dialog box is modified to **System Options - Relations/Snaps**.
7. Make sure that the **Grid** check box is selected in this dialog box. Next, choose the **OK** button to exit the dialog box.

Now, when you move the cursor, the coordinates displayed close to the lower right corner of the drawing area show an increment of 5 mm.

Drawing the Sketch

As evident from the Figure 2-18, the sketch will be drawn using the **Line** tool and you need to start drawing the sketch from the lower left corner of the sketch. As the model of the sketch is a revolved feature, you first need to create its axis by using the **Centerline** tool, so that while converting the sketch you can use this as an axis of revolution. You will learn more about the revolve features in later tutorials.

1. Click on the down arrow next to the **Line** tool; the **Line** flyout is displayed, refer to Figure 2-20.

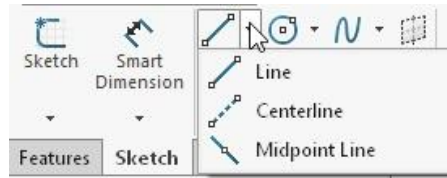


Figure 2-20 *The Line flyout*

2. Choose the **Centerline** button from the **Line** flyout; the **Insert Line PropertyManager** is displayed at the left of the drawing area. Note that in the **Options** rollout of this PropertyManager, the **For construction** check box is selected.

If the **For construction** check box of the **Insert Line PropertyManager** is selected, you can draw a construction line or a centerline. The construction lines or the centerlines are drawn only for the aid of sketching and are not considered while converting the sketches into features. You can draw a construction line similar to the sketched line by using the **Centerline** tool.

3. Move the cursor towards the origin and specify the start point of the centerline when the cursor snaps to the origin and the coincident relation is displayed below the cursor.
4. Move the cursor horizontally toward the left and specify the end point of the centerline when the length of the centerline above the cursor is displayed close to 120. Note that, you may need to scroll the wheel of the mouse to zoom in/out of the drawing. You can also invoke the **Zoom In/Out** tool by choosing the **View > Modify > Zoom In/Out** from the SOLIDWORKS menus to zoom in/out of the drawing.


The **Zoom In/Out** tool is used to dynamically zoom in or out of the drawing. When you invoke this tool, cursor changes to zoom cursor. To zoom out of a drawing, press and hold the left mouse button and drag the cursor to the downward direction. Similarly, to zoom in a drawing, press and hold the left mouse button and drag the cursor in upward direction. As you drag the cursor, the drawing display area will be modified dynamically. After getting the desired view, exit this tool by right-clicking and choosing the **Zoom In/Out** button from the shortcut menu displayed.

5. Right-click in the drawing area; a shortcut menu is displayed. Choose the **Select** option from it to exit the **Centerline** tool.
6. Press the F key; the sketch is zoomed and it fits on the screen.

- 7 . Choose the **Line** button from the **Sketch CommandManager**; the arrow cursor changes to line cursor. Also, the **Insert Line PropertyManager** is displayed on the left of the drawing area.

Tip

*You can also draw a construction line using the **Line** tool. To do so, invoke the **Insert Line PropertyManager** by choosing the **Line** tool. Next, select the **For construction** check box from the **Options** rollout of the **PropertyManager** to draw the construction line.*


8. Move the line cursor to a location whose coordinates are 0 mm, 15 mm, and 0 mm.
9. Left-click at this point and move the cursor horizontally toward the left. You will notice that the symbol of the **Horizontal** relation  is displayed below the line cursor and the length and angle of the line are displayed above the line cursor.

Note

You will learn more about adding relations in the later tutorials.

10. Left-click again when the length of the line above the line cursor is displayed as 120.

The first horizontal line is drawn. As you are drawing continuous lines, the endpoint of the line drawn is automatically selected as the start point of the next line.

11. Move the line cursor vertically upward. The symbol of the **Vertical** relation  is displayed on the right of the line cursor and the length of the line is displayed above the line cursor. Click when the length of the line on the line cursor is displayed as 25.
12. Move the cursor horizontally toward the right and click when the length of the line on the line cursor is displayed as 45.
13. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor is displayed as 15.
14. Move the line cursor horizontally toward the right and click when the length of the line on the line cursor is displayed as 40.

15. Move the line cursor vertically upward and click when the length of the line on the line cursor is displayed as 15.
16. Press F on the keyboard to fit the sketch on the screen.
17. Move the line cursor horizontally toward the right and click when the length of the line on the line cursor is displayed as 40.
18. Move the line cursor vertically upward and click when the length of the line on the line cursor is displayed as 15.
19. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor displays as 40.
20. Move the line cursor vertically downward and click when the length of the line on the line cursor is displayed as 5.
21. Move the line cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 35.
22. Move the line cursor vertically downward and click when the length of the line on the line cursor is displayed as 15.
23. Move the line cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 40.
24. Move the line cursor vertically downward and click when the length of the line on the line cursor is displayed as 15.
25. Move the line cursor horizontally toward the left and click when the length of the line on the line cursor is displayed as 40.
26. Move the line cursor vertically downward and click when the length of the line on the line cursor is displayed as 20.

27. Move the line cursor horizontally toward the right and click when the length of the line on the line cursor is displayed as 70.
28. Move the line cursor vertically downward to the start point of the first line. Click when an orange or yellow circle is displayed; the final sketch for Tutorial 2 is created, as shown in Figure 2-21. In this figure, the grid display is turned off for clarity.
29. Right-click and then choose **Select** from the shortcut menu displayed to exit the **Line** tool.

Note

*The display of relations shown in Figure 2-21, can be turned on or off by choosing the **View Sketch Relations** button from the **View (Heads-Up) > Hide/Show items** flyout.*

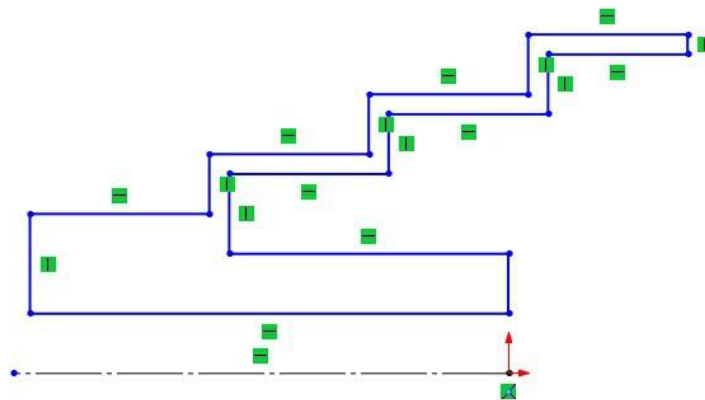


Figure 2-21 Final sketch for Tutorial 2

Tip

*In case, you have drawn wrong sketch entities, you can delete them. To do so, select the entities to be deleted by using the **Select** tool (cursor) and then press the **DELETE** key. You can select the entities individually or more than one by defining a window around the entities. When you select the entities, they turn light blue. When they turn light blue, press the **DELETE** key. You can also delete the sketched entities by selecting them and choosing the **Delete** option from the shortcut menu that is displayed on right-clicking.*

Saving the Sketch

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box. Create the *SOLIDWORKS Tutorials* folder inside the \Documents folder and then create the *c02* folder inside the *SOLIDWORKS Tutorials* folder, if it is not created in the Tutorial 1 of this tutorial.
2. Enter **c02_tut02** as the name of the document in the **File name** edit box and choose the **Save** button. The document is saved at the location \Documents\SOLIDWORKS Tutorials\c02.
3. Close the document by choosing **File > Close** from the SOLIDWORKS menus.

Tip

*If you save a file in the sketching environment and then open it next time by using the **Open** button available in the Menu Bar, it will open in the sketcher environment only.*

Tutorial Exercise 3

In this tutorial, you will draw the basic sketch of the model shown in Figure 2-22. The sketch to be drawn is shown in Figure 2-23. Do not dimension the sketch; the solid model and its dimensions are given for your reference only. **(Expected time: 30 min)**

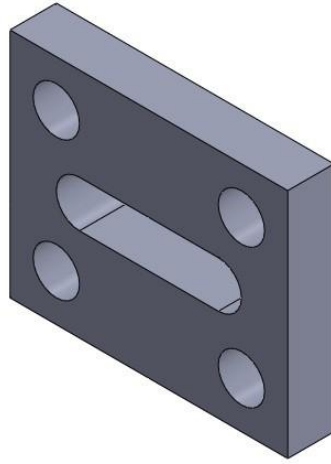


Figure 2-22 Solid model for Tutorial 3

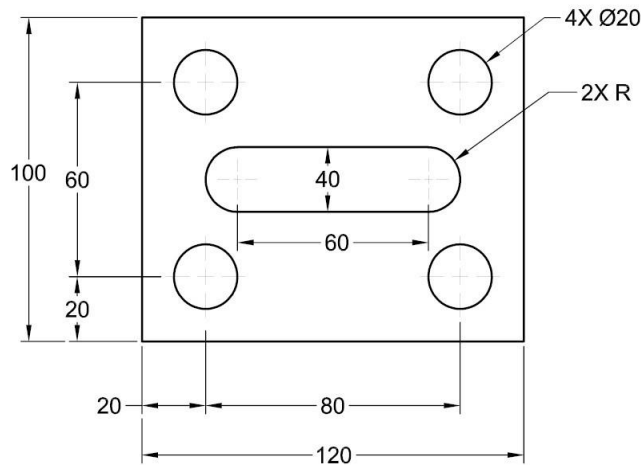


Figure 2-23 Sketch for Tutorial 3

The following steps are required to complete this tutorial:

- Invoke a new part document.
- Modify the settings of the snap, grid, and units settings so that the cursor jumps through a distance of 10 mm.
- Invoke the sketching environment.
- Draw the sketch using the sketching tools, refer to Figures 2-26 through 2-31.
- Save the sketch and then close the file.

Invoking a New Part File

- Choose the **New** button from the Menu Bar; the **New SOLIDWORKS Document** dialog box is displayed with the **Part** button chosen by default.
- Choose the **OK** button from this dialog box; a new SOLIDWORKS part document is started.

Now, you need to invoke the sketching environment, to draw the sketch of the model.

- Choose the **Sketch** button from the **Sketch CommandManager**; the **Edit Sketch PropertyManager** is displayed.
- Select the **Front Plane** from the drawing area; the sketching environment is invoked.

Modifying the Snap, Grid, and Units Settings

As the dimensions in the sketch are multiple of 10, you need to modify the grid and snap settings so that the cursor jumps through a distance of 10 mm.

1. Click on the **Unit system** that is located in the Status Bar; a flyout is displayed with a tick mark next to the unit system of the current document, refer to Figure 2-24.

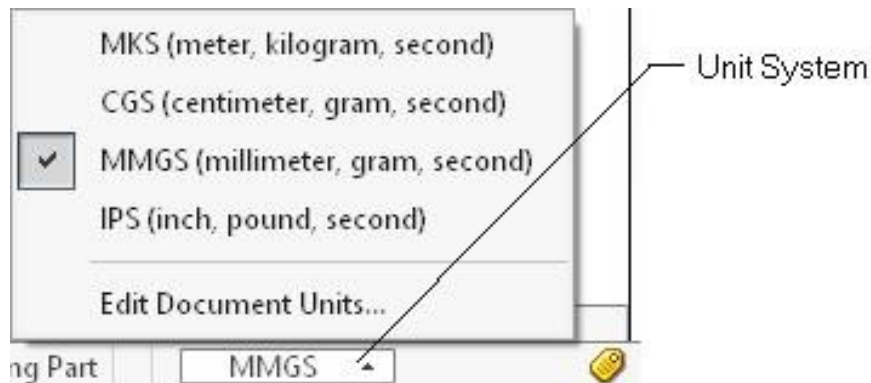


Figure 2-24 The flyout showing the currently selected unit system

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

Note

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

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

Note

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

As evident from Figure 2-23, the dimensions in the sketch are multiple of 10. Therefore, you need to modify the grid and snap settings so that the cursor jumps through a distance of 10 mm.

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

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

Drawing the Sketch

In this tutorial, you need to draw the sketch in two parts. Initially, you need to draw the outer loop of the sketch which is a rectangle. Next, you need to draw the inner loops of the sketch which consist of four simple holes and an elongated hole.

1. Choose the **Corner Rectangle** tool from the **Rectangle** flyout of the **Sketch CommandManager** to draw a rectangle by specifying the two diagonally opposite corners; the **Rectangle PropertyManager** is displayed on the left of the drawing area. Also, the arrow cursor changes to the rectangle cursor.

In SOLIDWORKS, the tools that are used to draw rectangles are grouped together in the **Rectangle** flyout. To invoke this flyout, click on the down arrow located on the right of the **Rectangle** flyout, refer to Figure 2-25. From this flyout, you can select an appropriate method to draw a rectangle.

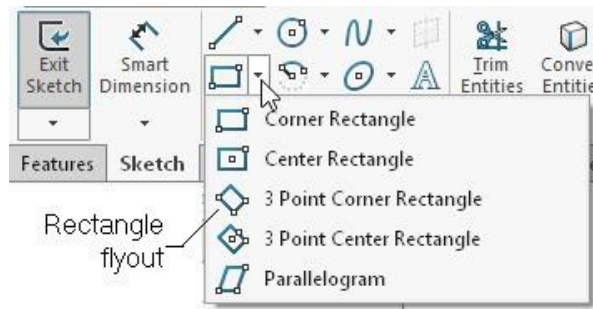


Figure 2-25 *The **Rectangle** flyout*

The **Corner Rectangle** tool is used to draw a rectangle by specifying the two diagonally opposite corners. To draw a rectangle by specifying the center and one of the corners, use the **Center Rectangle** tool. To draw a rectangle at an angle by specifying its three corners, use the **3 Point Corner Rectangle** tool. In this case, the first two corners will define the length and angle of the rectangle and the third corner will define the width of the rectangle. Similarly, the **3 Point Center Rectangle** tool of the **Rectangle** flyout is use to draw a centerpoint rectangle at an angle. The last tool of the **Rectangle** flyout is the **Parallelogram** tool. This tool is used to draw a parallelogram.

Note

*You can also select an appropriate method to draw a rectangle from the **Rectangle Type** rollout of the **Rectangle PropertyManager**.*

2. Move the rectangle cursor toward the origin and specify the first corner of the rectangle when it snaps to the origin and the symbol of coincident relation is displayed below the cursor.

Note

You will learn more about adding relations to the sketch in the later tutorials.

3. Move the cursor toward right direction and specify the second corner of the rectangle when the X and Y length value of the rectangle displayed above the cursor are 120 and 100. Note that, you may need to scroll the wheel of the mouse to zoom in/out of the drawing. You can also invoke the **Zoom In/Out** tool by choosing the **View > Modify > Zoom In/Out** from the SOLIDWORKS menus to zoom in/out of the drawing.

The **Zoom In/Out** tool is used to dynamically zoom in or out the drawing. When you invoke this tool, cursor changes to by zoom cursor. To zoom out of a drawing, press and hold the left mouse button and drag the cursor to downward direction. Similarly, to zoom in a drawing, press and hold the left mouse button and drag the cursor to upward direction.

Now, when you drag the cursor, the drawing display area gets modified dynamically. After getting the desired view, exit this tool by right-clicking and choosing the **Zoom In/Out** button from the shortcut menu displayed.

4. Exit the **Corner Rectangle** tool by right-clicking and choosing the **Select** option from the shortcut menu, displayed. Figure 2-26 shows the outer loop of the sketch drawn using the **Corner Rectangle** tool.

You have drawn the outer loop of the sketch. Now, you need to draw the inner loops, which consist of four holes and an elongated hole. First, draw an elongated hole by using the **Straight Slot** tool and then the four holes by using the **Circle** tool.

5. Choose the **Straight Slot** button from the **Slot** flyout of the **Sketch CommandManager**; the **Slot PropertyManager** is displayed on the left of the drawing area, refer to Figure 2-27.

In SOLIDWORKS, the tools that are used to draw a slot profile are grouped together in the **Slot** flyout. To display this flyout, click on the down arrow located next to the **Straight Slot** tool; the **Slot** flyout will be displayed; as shown in Figure 2-28. From this flyout, you can choose the appropriate method to draw a slot profile. Alternatively, invoke the **Slot PropertyManager** by choosing the **Straight Slot** button from the **Sketch CommandManager**. Next, select an appropriate method to draw a slot profile from the **Slot Type** rollout.



Figure 2-26 The outer loop of the sketch

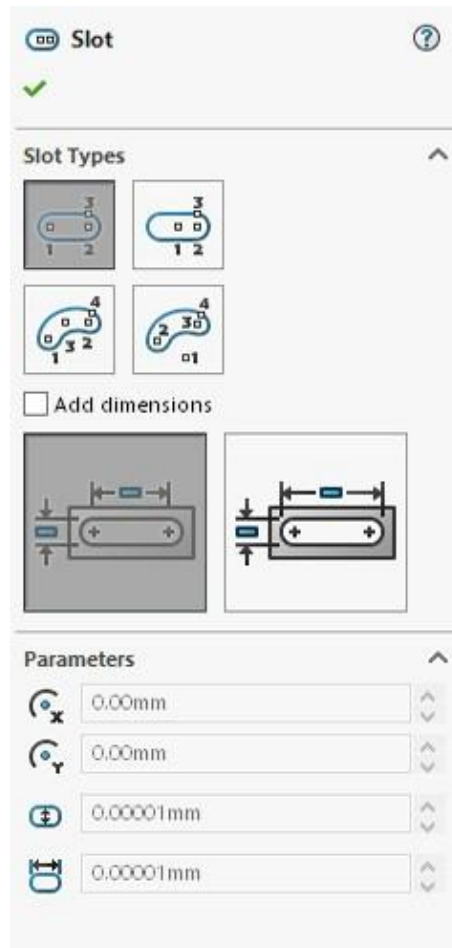


Figure 2-27 The Slot PropertyManager

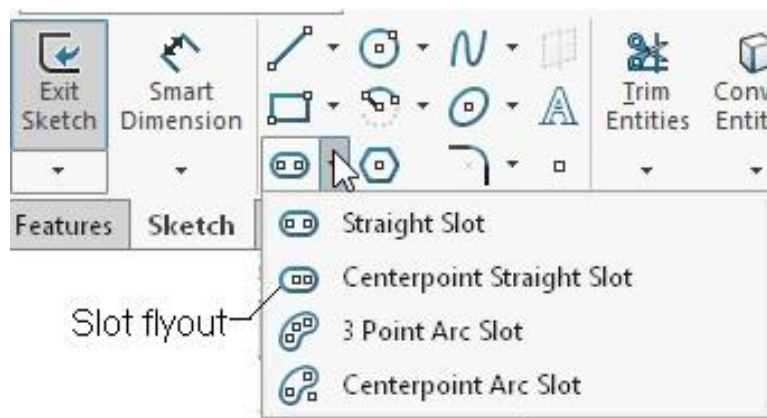


Figure 2-28 The Slot flyout

Tip

*The tool selected from the **Slot** flyout will be displayed as default tool in the **Sketch CommandManager**.*

6. Make sure that the **Straight Slot** and **Center to Center** buttons are chosen in the **Slot Type** rollout of the **Slot PropertyManager**.

The **Slot Type** rollout of the PropertyManager displays different types of methods to create a slot profile. You can choose the appropriate method from this rollout to draw a slot profile. The **Center to Center** button of this rollout is used to measure the slot length from its center to center. You can also choose the **Overall Length** button from this rollout to measure the overall length of the slot. Note that, by default the **Add dimensions** check box of this rollout is clear. If you select this check box, the current dimensions of the slot will be added to it automatically in the drawing area.

7. Move the cursor to the location whose coordinates are 30 and 50 and then specify the first point of the slot at this location.

Note

The coordinates are displayed in the Status Bar that is located at the bottom of the drawing area.

8. Move the cursor horizontally toward right. You will notice that the symbol of the horizontal relation is displayed below the cursor and the length and angle values are displayed above the cursor.
9. Left-click when the length above the cursor shows the value 60; a reference slot is attached to the cursor.
10. Specify a point in the drawing area, when the width value of the slot is displayed closer to 20 mm in the **Parameters** rollout of the **Slot PropertyManager**.

The **Parameters** rollout of the **Slot PropertyManager** displays the parameters of the slot. The parameters displayed in this rollout will be enabled for modification once the slot is created.

11. Set the value of the **Slot Width** spinner in the **Parameters** rollout of the PropertyManager to **20 mm**.
12. Exit the tool by right-clicking and choosing the **Select** option from the shortcut menu displayed. Figure 2-29 shows the sketch after creating the sketch of the elongated hole by using the **Straight Slot** tool.

13. Choose the **Circle** tool from the **Circle** flyout of the **Sketch CommandManager**; the **Circle PropertyManager** is displayed on the left side of the drawing area. Also, the cursor is changed to a circle cursor.

In SOLIDWORKS, there are two methods to draw circles. The first method is by specifying the center point of a circle and then defining its radius. The second method is drawing a circle by defining the three points that lie on its periphery. The tools used to create circle are grouped together in the **Circle** flyout in the **Sketch CommandManager**. To invoke this flyout, click on the down arrow located next to the **Circle** tool, refer to Figure 2-30.

Note

*You can also select an appropriate method to draw a circle from the **Circle Type** rollout of the **Circle PropertyManager**.*

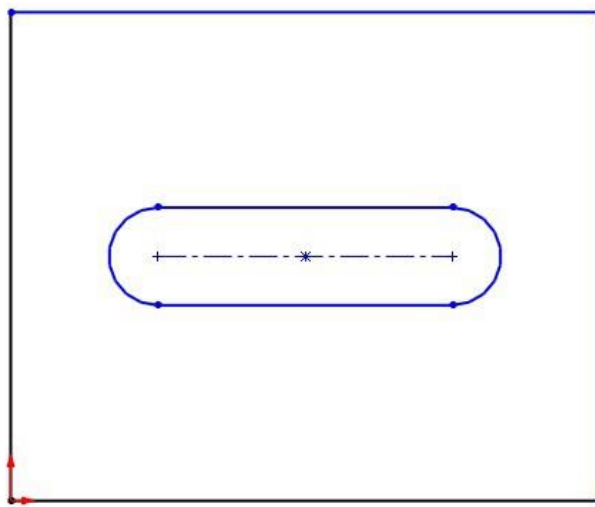


Figure 2-29 The sketch after creating the elongated hole

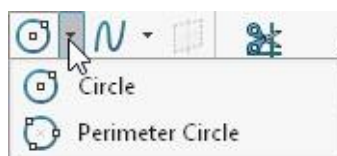


Figure 2-30 The **Circle** flyout

14. Move the circle cursor to the location whose coordinates are 20 and 20 and then

specify the center point of the circle at this location; the rubber band circle is attached with the circle cursor. The circle cursor consists of a crosshairs and a circle with snap grid below it.

15. Move the circle cursor horizontally toward the right and click when the radius of the circle on the circle cursor is displayed as 10. The circle of radius 10 mm has been drawn. Note that the **Circle** tool is still active.
16. Move the circle cursor to the location whose coordinates are 100 and 20 and then specify the center point of the circle at this location; the rubber band circle is attached with the circle cursor.
17. Move the circle cursor horizontally toward the right and click when the radius of the circle on the circle cursor is displayed as 10. The circle of radius 10 mm has been drawn. Note that the circle tool is still active.
18. Similarly, draw the other two circles. The coordinates of the center point of the other two circles are 20, 80 and 100, 80.
19. Right-click in the drawing area; a shortcut menu is displayed. Choose the **Select** option from the shortcut menu to exit the **Circle** tool.
20. Choose the **Zoom to Fit (F)** button from the **View (Heads-Up) > Hide/Show items** toolbar to fit the sketch on the screen. The final sketch with the display of relations turned on, is shown in Figure 2-31.

Note

*To turn ON/OFF the display of relations, choose the **View Sketch Relations** button from the **View (Heads-Up)** flyout.*

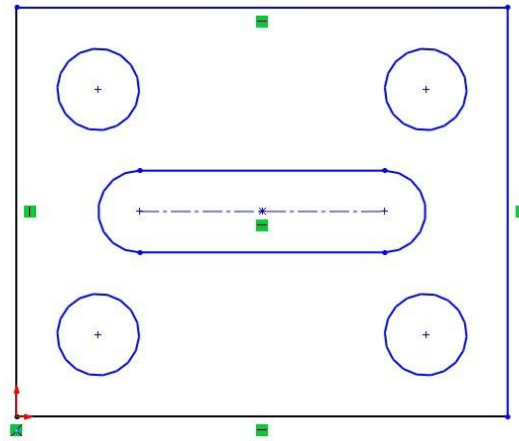


Figure 2-31 *Final sketch for Tutorial 3*

Saving the Sketch

After creating the sketch, save it in the *SOLIDWORKS Tutorials* folder.

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box. Browse to the *c02* folder inside the *SOLIDWORKS Tutorials* folder.
2. Enter **c02_tut03** as the name of the document in the **File name** edit box and choose the **Save** button. The document is saved at the location *\Documents\SOLIDWORKS Tutorials\c02*.
3. Close the document by choosing **File > Close** from the SOLIDWORKS menus.