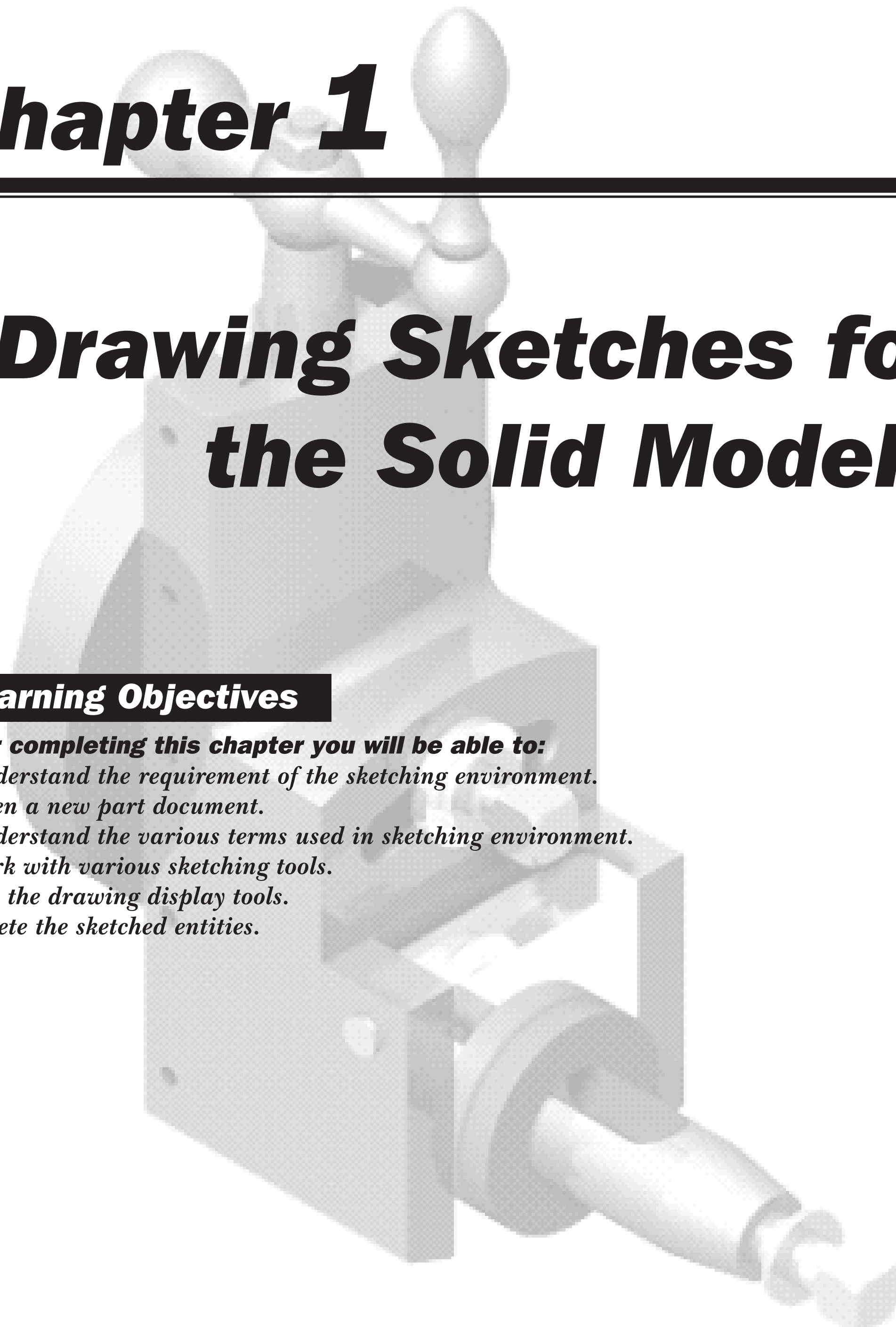
Chapter 1

Drawing Sketches for the Solid Models



Learning Objectives

After completing this chapter you will be able to:

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

THE SKETCHING ENVIRONMENT

Most of the products designed using SolidWorks are a combination of sketched features, placed features, and derived features. The placed and derived features are created without creating a sketch but the sketched features require a sketch to be created first. Generally, the base feature of any design is a sketched feature and is created by drawing the sketch. However, once you are conversant with the various options of SolidWorks, you can also use a derived feature or a derived part as the base feature. Therefore, while creating any design, the first and foremost point is to draw the sketch for the base feature. Once you have drawn the sketch for the base feature, you can convert it into the base feature and then add the other sketched, placed, and derived features to complete the design. In this chapter you will learn to create the sketch for the base feature using the various sketcher entities.

In general terms, a sketch is defined as the basic contour for the feature. For example, consider the spanner shown in Figure 1-1.

Figure 1-1 *Solid model of a spanner*

This spanner consists of a base feature, a cut feature, a mirror feature (cut on the back face), fillets, and an extruded text feature. The base feature of this spanner is shown in Figure 1-2. This base feature is created using a single sketch shown in Figure 1-3. This sketch is drawn in the sketching environment using the various sketching tools. Therefore, to draw the sketch of the base feature, you first need to invoke the sketching environment where you will draw the sketch.

Figure 1-2 *Base feature of the spanner*

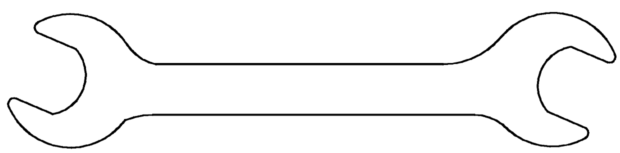


Figure 1-3 *Sketch for the base feature of the spanner*

The sketching environment of SolidWorks can be invoked any time in the **Part** mode, **Assembly** mode, and **Drawing** mode. You just have to specify that you want to draw the sketch of a feature. This is done by choosing the **Sketch** button from the **Sketch** toolbar, see Figure 1-4. This toolbar is available by default on the right of the drawing window. When you choose this button, the sketching environment will be invoked. You can draw the sketch in this environment and then proceed to the part modeling environment for converting the sketch into a solid model.

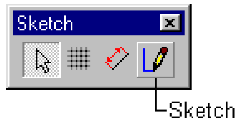


Figure 1-4 *Choosing the* ***Sketch*** *button from the* ***Sketch*** *toolbar to invoke the sketching environment*

OPENING A NEW DOCUMENT

When you start SolidWorks, the **Tip of the day** dialog box will be displayed as shown in Figure 1-5.

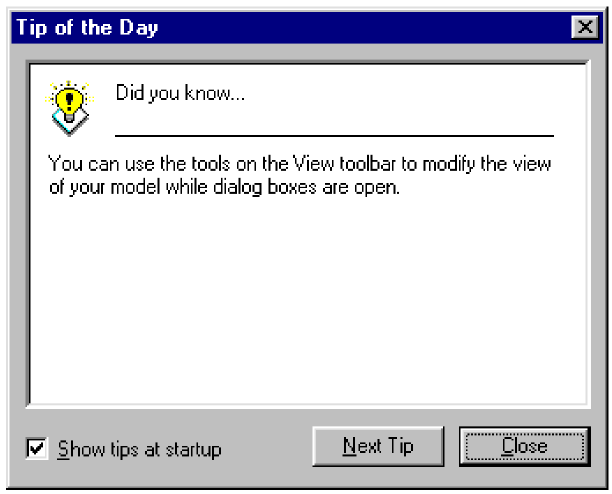


Figure 1-5 Tip of the Day *dialog box*

Tip. *If the* ***Tip of the Day*** *dialog box is not displayed when you start the SolidWorks session then choose* ***Help*** *>* ***Tip of the Day*** *from the menu bar. The* ***Tip of the Day*** *dialog box is displayed. Select the* ***Show tips at startup*** *option from this dialog box. By selecting the* ***Show tips at startup*** *option, the* ***Tip of the Day*** *dialog box will be displayed every time when you start SolidWorks session. You get valuable tips from the* ***Tip of the Day*** *dialog box. The tips displayed in this dialog box are helpful in the full utilization of this CAD package.*

Choose the **Close** button from this dialog box. You will notice that the **Welcome to SolidWorks 2003** window is displayed as shown in Figure 1-6. This window can be used to open a new file, open an existing file, and use the various types of helps available in SolidWorks to start working with this 3D solid modeling tool. This window is also used to visit the website of the SolidWorks partners.

Tip. *When you choose any option from the* ***Welcome to SolidWorks 2003*** *window, it is not closed. It is minimized on the screen and you can restore this window to open an existing file, open a new file, or use any other option.*

***Figure 1-6 Welcome to SolidWorks 2003** window*

Choose **New Document** from the **Welcome to SolidWorks 2003** window. The **New SolidWorks Document** dialog box is displayed as shown in Figure 1-7. The various options available in this dialog box are discussed next.

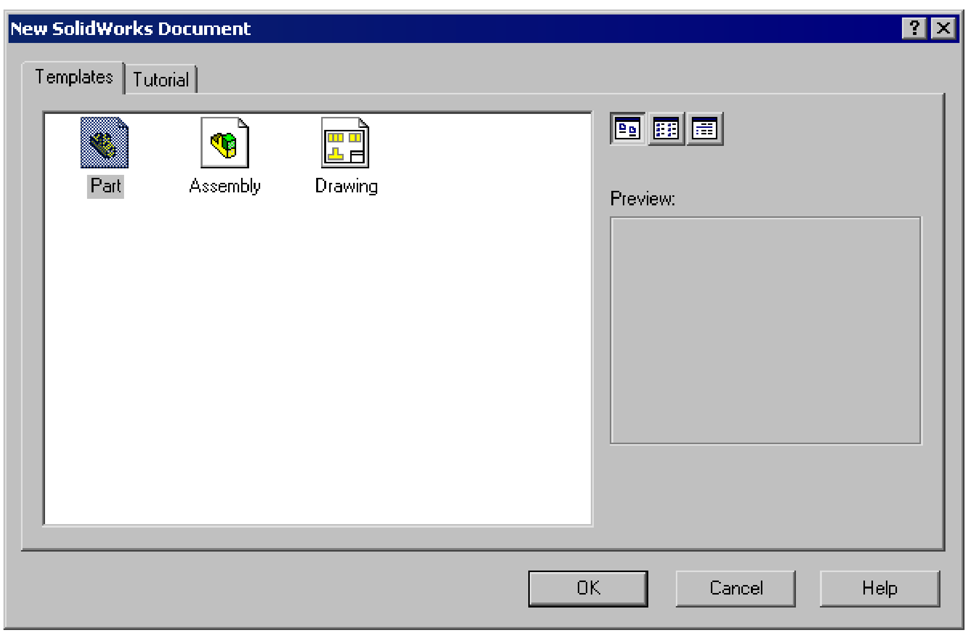


***Figure 1-7 New SolidWorks Document** dialog box*

Template Tab

The **Template** tab displays the three default templates for opening a new part, assembly, or drawing file. These three default templates are discussed next.

Part Template

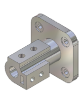
Select the **Part** template and choose **OK** from the **New SolidWorks Document** dialog box to open a new part document for creating the solid models or sheet metal component. When you open a new part document, you will enter the **Part** mode and **Plane 1** is selected by default for sketching. As mentioned earlier, choose the **Sketch** button from the **Sketch** toolbar and you will enter the sketching environment where you can draw the sketch of the base feature.

Assembly Template

Select the **Assembly** template and choose **OK** from the **New SolidWorks Document** dialog box to open a new assembly document. In an assembly document, you will assemble various components created in the various part files. You can also create the components in the assembly document.

Drawing Template

Select the **Drawing** template and choose **OK** from the **New SolidWorks Document** dialog box to open a new drawing document. In a drawing document, you can generate or create the drawing views of the parts created in the part document or the assemblies created in the assembly documents. When you select the **Drawing** template, the **Create RapidDraft Drawing** check box is displayed on the lower left corner of the **New SolidWorks Document** dialog box. This check box is selected to create the rapid draft drawings. A rapid draft drawing is the one that can be opened and edited without loading the part or the assembly file in the memory of SolidWorks.



Note

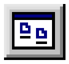
The concept of rapid draft is discussed in detail in later chapters.

Tutorial Area

The **Tutorial** tab also displays the three default templates for opening a new part, assembly, or drawing file. The only difference between the default templates in the **Template** tab and the default templates in the **Tutorial** area is the drawing template. If you choose the drawing template from the **Tutorial** tab, a standard **A-Landscape** format sheet will be displayed in the current document, whereas if you choose the drawing template from the **Template** tab then you can choose a drawing sheet of any size. Therefore, it is recommended that you always select the templates from the **Template** tab.

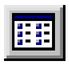
In addition to the **Template** area, this dialog box also provides you with three buttons and the **Preview** area. These options are discussed next.

Large Icon button



The **Large Icon** button is used to display the templates in the **Templates** area in the form of large icons. This button is chosen by default.

List button



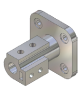
The **List** button is used to list the three templates in the **Template** area in the form of small icons.

List Details button

The **List Details** button is chosen to list the details of the templates in the **Templates** area. When you choose this button, the **Template** area is divided into three columns: **Name**, **Size**, and **Modified**. These columns display the name, size, and the date when the template was modified.

Preview Area

The **Preview** area is used to preview the template to be used.



Note

You can customize the templates in the ***New SolidWorks Document*** *dialog box according to your need and add a tab with the customized templates in this dialog box. Creation and customization of the templates is discussed later in the book.*

THE SKETCHING ENVIRONMENT

When you choose the **Part** template, a new part file will be opened in the part modeling environment. As mentioned earlier, you will have to choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment. The default screen appearance of a SolidWorks part document in the sketching environment is shown in Figure 1-8.

SETTING UP THE DOCUMENT OPTIONS

When you install SolidWorks on your system, you are prompted to specify the dimensioning standards and units for measuring linear distances. You can specify these options at that time. The settings specified at that time are made the default settings and whenever you open a new SolidWorks document, the new file will have those settings. However, if you want to modify these settings for a particular file, you can easily do it using the **Document Properties** dialog box. To invoke this dialog box, choose **Tools > Options**. When you invoke this option, the **System Options** dialog box will be displayed. In this dialog box, choose the **Document Properties** tab. The name of this dialog box is changed to **Document Properties** dialog box.

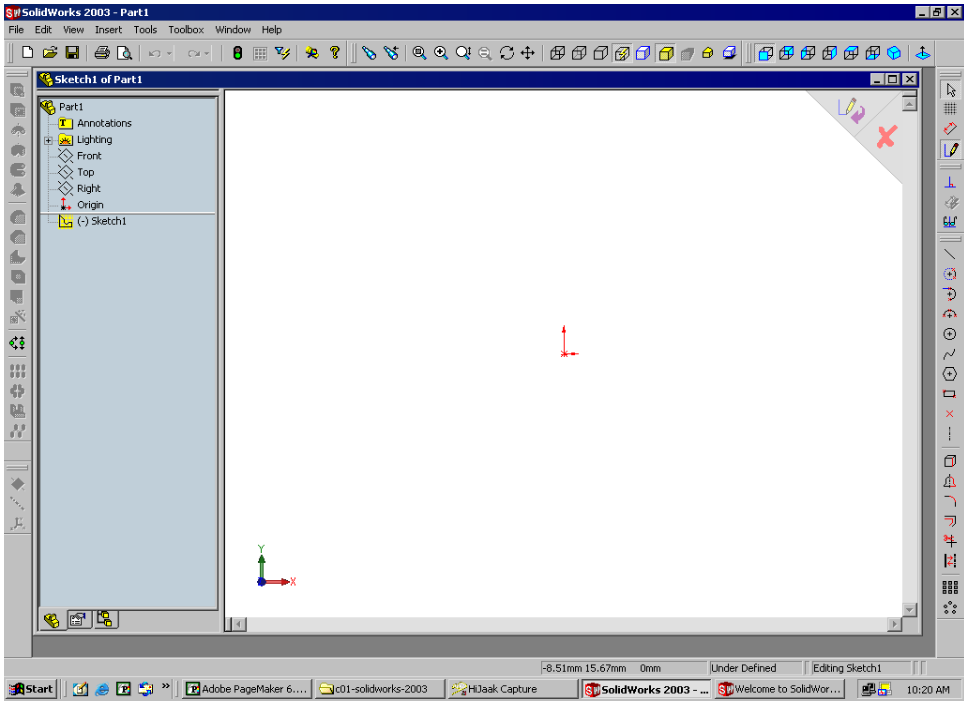
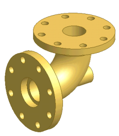


Figure 1-8 *Screen display of a part document in the sketcher environment*



Tip. *When you open a new SolidWorks document, it is not maximized in the SolidWorks window. This is the reason the area of the new document is not maximum. To maximize the document, choose the **Maximize** button provided on the upper right corner of the document. You can also maximize the document window by double-clicking the blue bar on top of the document window.*

Modifying the Dimensioning Standards

To modify the dimensioning standards, invoke the **System Options** dialog box and then choose the **Document Properties** tab. You will notice that the **Detailing** option is selected by default from the area on the left to display the detailing options as shown in Figure 1-9.

The default dimensioning standard that was selected while installing SolidWorks will be selected in the drop-down list provided in the **Dimensioning standard** area. You can select the required dimensioning standard from this drop-down list. The standards that are available in this drop-down list are ANSI, ISO, DIN, JIS, BSI, GOST, and GB. You can select any one of these dimensioning standards for the current document.

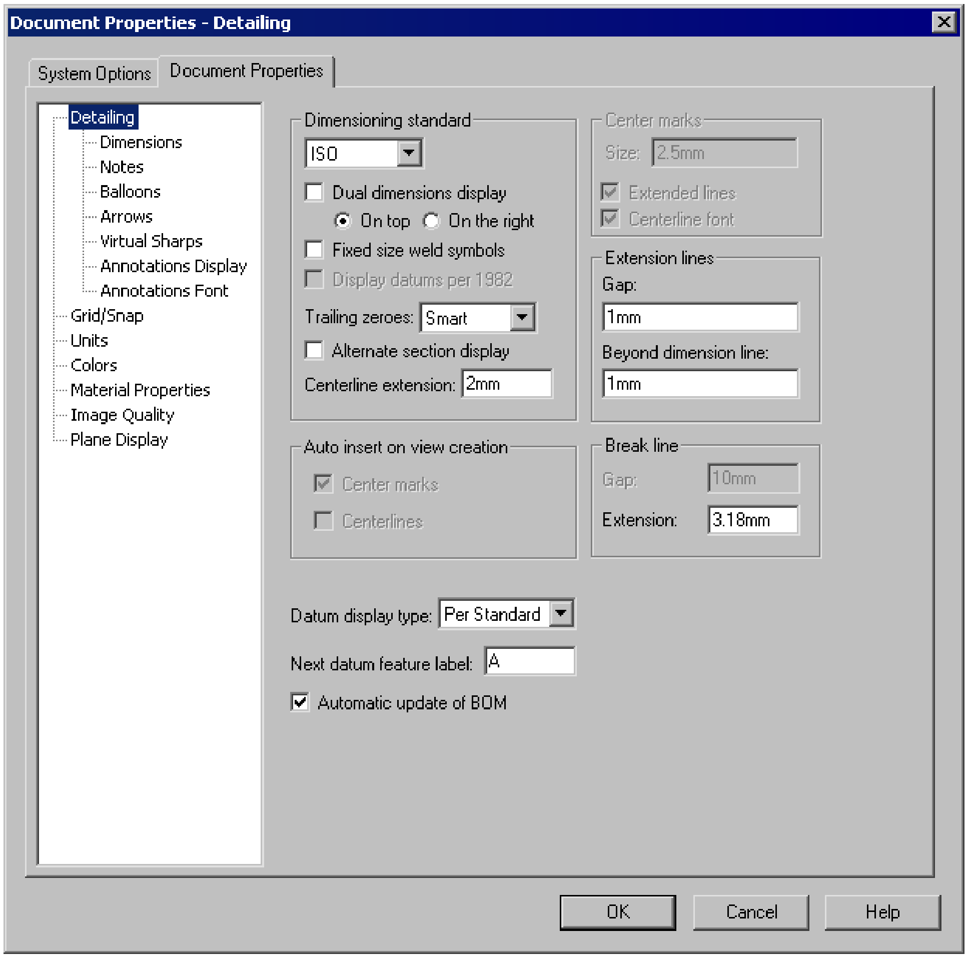


Figure 1-9 *Setting the dimensioning standards*

Modifying the Linear and Angular Units

To modify the linear and angular units, invoke the **System Options** dialog box and then choose the **Document Properties** tab. In this tab, select the **Units** option from the area on the left to display the options related to linear and angular units as shown in Figure 1-10. The default option for measuring the linear distances that was selected while installing SolidWorks will be various in the drop-down list provided in the **Linear units** area. The various types of units that you can select from this drop-down list are angstroms, nanometers, microns, millimeters, centimeters, meters, microinches, mils, inches, feet, and feet & inches. You can also change the units for angular dimensions by selecting it from the drop-down list in the **Angular units** area. The various types of angular units that can be selected from this drop-down list are degrees, deg/min, deg/min/sec, and radians.

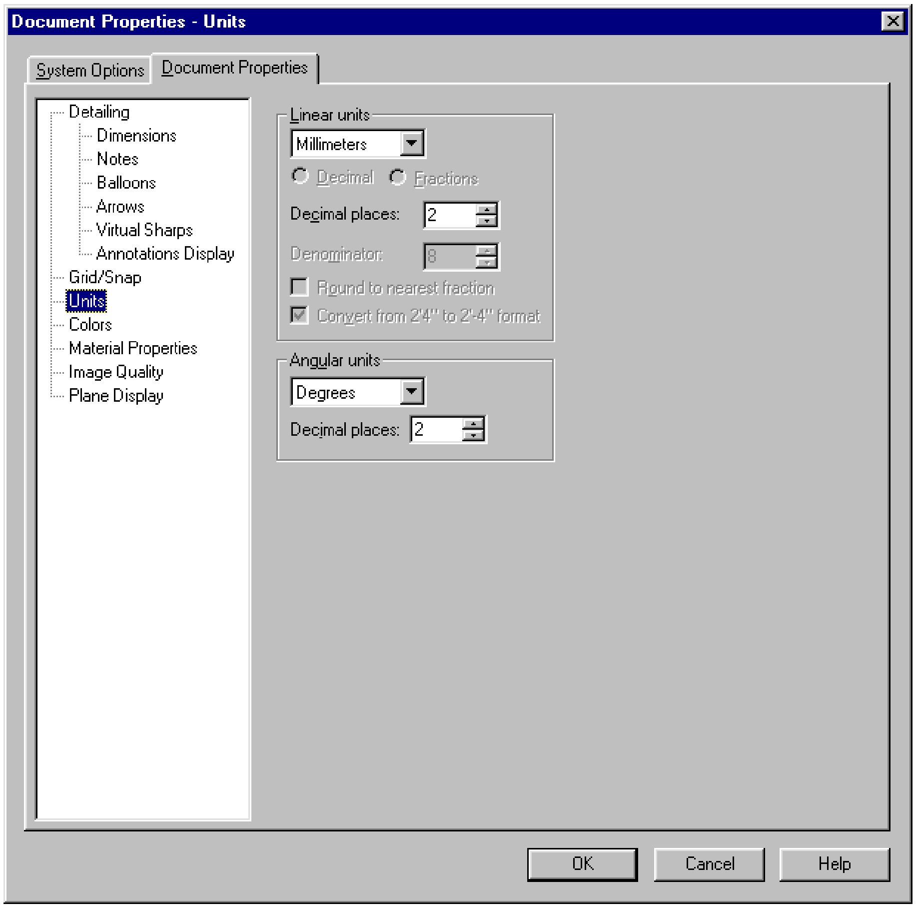
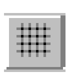


Figure 1-10 *Setting the dimensioning units*

Modifying the Snap and Grid Settings



When you switch to the sketching environment of SolidWorks, you will notice that the cursor jumps through a distance of 10 units. This is evident from the coordinates of the current location of the cursor displayed close to the lower left corner of the SolidWorks window. You will notice that as you move the cursor, the coordinates change. This change in coordinate values is in the increment of 10 units. Therefore, if you draw a sketched entity, its length will change in the increment of 10. The reason for this is that the default ratio between the major and minor grid spacing in the sketching environment is 10. However, if you want that the coordinates should change in any other increment, you will have to modify the ratio of major and minor lines in the grid accordingly. For example, if you want that the coordinates should change in the increment of 5 units, you will have to make the ratio of major and minor lines to 5. To do this, choose the **Grid** button from the **Sketch** toolbar. The **Document Properties - Grid/Snap** dialog box will be displayed as shown in Figure 1-11.

Note that the value of the **Major grid spacing** spinner is **100** and that of the **Minor-lines per**

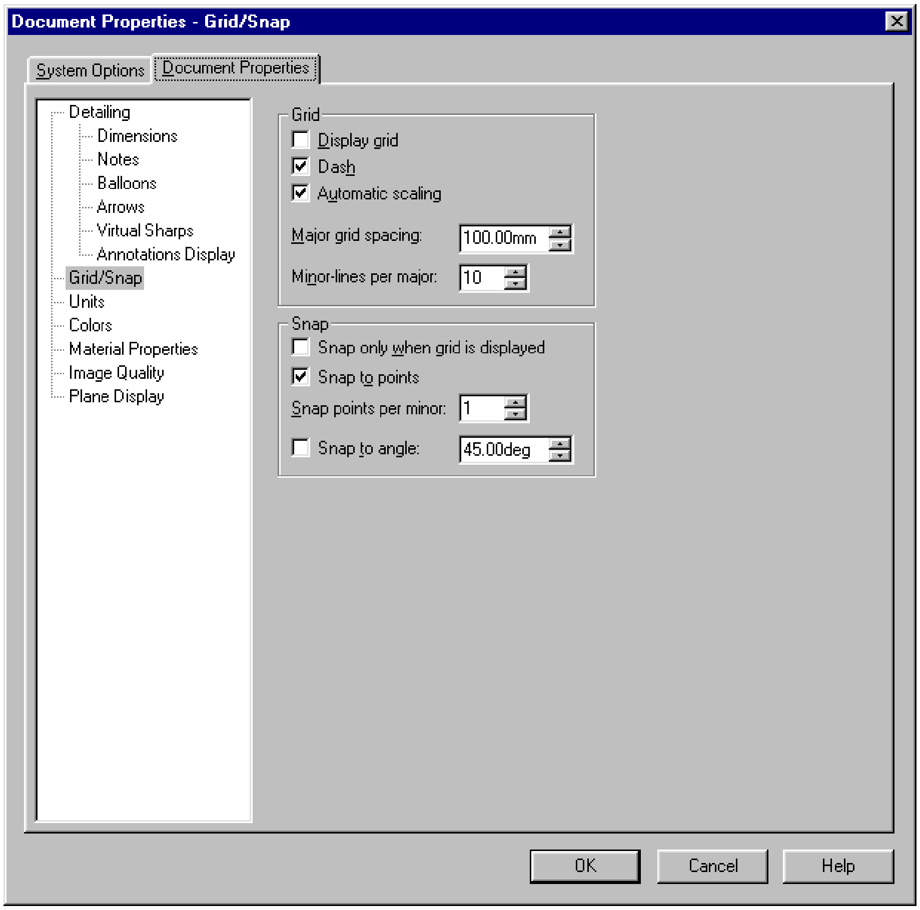
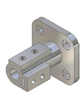


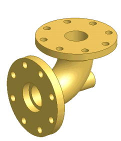
Figure 1-11 *Modifying the Grid and Snap settings*

major spinner is **10**. This means that the ratio between the major and minor lines is 10 because of which the cursor jumps through a distance of 10mm. To make the cursor jump through a distance of 5mm, set the value of the **Major grid spacing** spinner to **50**. The cursor will now jump through a distance of 5mm.



Note

Remember that this setting will only be for the current documents. When you open another document, it will again have the settings that were defined while installing SolidWorks.



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. If you do not want the cursor to snap to a point, clear the* ***Snap to points*** *check box from the* ***Snap*** *area.*

LEARNING ABOUT SKETCHER TERMS

Before you learn about the various sketching tools, it is important for you to understand some terms that are used in the sketching environment. These terms are discussed next.

Origin

The origin is a red color icon that is displayed in the middle of the sketching environment screen. This icon consists of two arrows displaying the X and the Y axis direction. The point of intersection of these two axes is the origin point and the coordinates of this point are 0,0.

Inferencing Lines

The inferencing lines are the temporary lines that are used to track a particular point on the screen. These lines are dashed lines and are automatically displayed when you select a sketching tool in the sketcher environment. These lines are created from the endpoint of a sketched entity or from the origin. For example, if you want to draw a line from the point where two imaginary lines intersect, you can use the inferencing lines to locate the point and then draw the line from that point. Figure 1-12 shows the use of inferencing lines to locate the point of intersection of two imaginary lines.

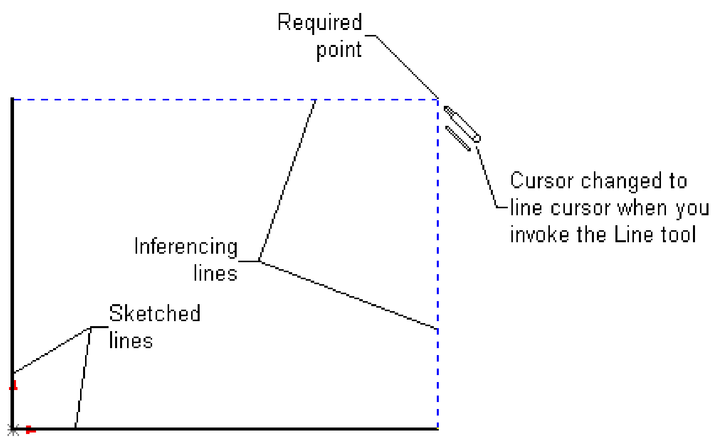


Figure 1-12 *Using inferencing lines to locate a point*

Figure 1-13 shows the use of inferencing lines to locate the center of an arc. Notice that the inferencing lines are created from the endpoint of the line and from the origin.

Note

The inferencing lines that are displayed on the screen will be either blue or brown in color. The blue inferencing lines suggest that the relations are not added to the sketched entity and the brown inferencing lines suggest that the relations are added to the sketched entity. You will learn about various relations in later chapters.

Tip. *You can also turn off the automatic inferencing lines by choosing* ***Tools > Sketch Settings > Automatic Inferencing Lines*** *from the menu bar.*

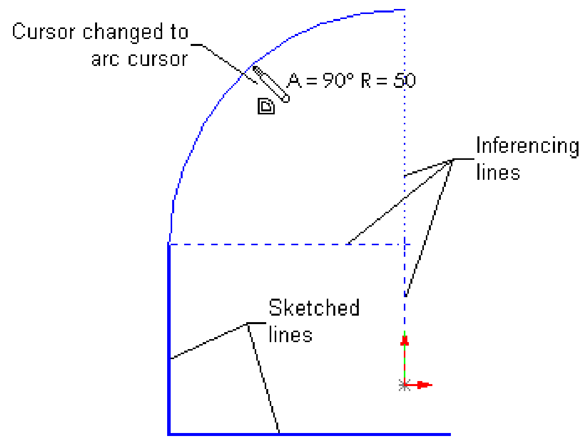


Figure 1-13 *Using inferencing lines to locate the center of an arc*

Select Tool

Menu:	Tools > Select
Toolbar:	Sketch > Select

The **Select** tool is used to select a sketched entity or exit any sketching tool that is active. You can select the sketched entities by picking them one by one using the left mouse button. You can also define a window by holding the left mouse button down and dragging the cursor around the sketched entities to select them. Remember that only those entities will be selected that lie completely inside the window that you have defined.

Note
*You can also invoke the **Select** tool or exit a sketching tool by pressing the ESC key.*

Once you are familiar with these terms, you will learn about various sketcher tools available in SolidWorks. The various sketcher tools are discussed next.

DRAWING LINES

Menu:	Tools > Sketch Entity > Line
Toolbar:	Sketch Tools > Line

The lines are one of the basic sketching tools available in SolidWorks. In general terms, a line is defined as the shortest distance between two points. As mentioned earlier, SolidWorks is a parametric solid modeling tool. This property allows you to draw a line of any length and at any angle and then later force it to the desired length and angle. To draw a line in the sketcher environment of SolidWorks, choose the **Line** tool. You will notice that the cursor that was an arrow earlier is replaced by the line cursor. The line cursor is actually a pencil-like cursor with a small inclined line below the pencil.

In SolidWorks, there are two methods to draw lines. The first method is to draw continuous lines and the second method is to draw individual lines. Both these methods are discussed next.

Drawing Continuous Lines

This is the default method of drawing lines. In this method you just have to specify the startpoint and the endpoint of the line using the left mouse button. As soon as you specify the startpoint of the line, the **Line PropertyManager** will be displayed on the left of the screen. However, when you are drawing continuous lines, the options in the **Line PropertyManager** will not be available.

When you specify the startpoint and the endpoint of the line using the left mouse button, a line will be drawn between the two points. You will notice that the line is green in color and has square boxes at the two ends. The line is displayed in green color because it is still selected. As soon as you draw another line, choose the **Select** tool, or choose another tool, the line turns to blue in color and the new line is displayed in green color.

You will notice that after you have drawn the first line, another line is displayed on the screen. The startpoint of this line is the endpoint of the last line and the length of this line will increase or decrease as you move the mouse. This line is called a rubber-band line and the reason it is a called a rubber-band line is that this line will stretch like a rubber-band as you move the cursor. The point that you specify next on the screen will be taken as the endpoint of the second line and a line will be drawn such that the endpoint of the first line is taken as the startpoint of the new line and the point you specify is taken as the endpoint of the new line. Now, a new rubber-band line is displayed starting from the endpoint of the last line. This is a continuous process and you can draw as many continuous lines as you need by specifying the points on the screen using the left mouse button.

You can exit the continuous line drawing process by pressing the ESC key from the keyboard, by choosing the **Select** tool, or by double-clicking the screen. You can also right-click to display the shortcut menu and choose the **End Chain** option from the shortcut menu. Figure 1-14 shows a sketch drawn using continuous lines.

Note

*When you exit the line drawing process by double-clicking the screen or by choosing **End chain** from the shortcut menu, the current chain is ended but the **Line** tool is still active and you can draw additional lines.*

Drawing Individual Lines

This is the second method of drawing lines. Using this method you can draw individual lines and the startpoint of the next line will not necessarily be the endpoint of the last line. To draw individual lines, you need to press and hold down the left mouse button and drag the cursor from the startpoint of the line to the endpoint. Once you have dragged the cursor to the endpoint, release the left mouse button. A line will be drawn between the point from where you started dragging the mouse and the point where you released the mouse.

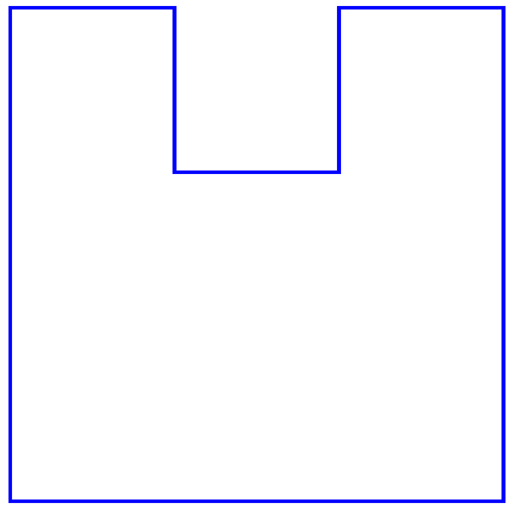
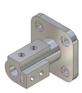


Figure 1-14 *Sketch drawn with the help of continuous lines*

To make the process of sketching in SolidWorks easy, you are provided with the **PropertyManager**. The **PropertyManager** is a table that is displayed on the left of the screen as soon as you select the first point of any sketch entity. The **PropertyManager** has all the parameters related to the sketched entity such as the startpoint, endpoint, angle, length, and so on. You will notice that as you start dragging the mouse, the **Line PropertyManager** is displayed on the left of the drawing window. All the options in the **Line PropertyManager** will be available when you release the left mouse button. Figure 1-15 shows partial display of the **Line PropertyManager**.



Note

*The **Line PropertyManager** will also display addition options about relations. You will learn more about relations in later chapters.*

After you have drawn the line, modify the parameters in the **Line PropertyManager** to force the line to the desired length and angle. You can also dynamically modify the line by holding the square boxes at the endpoints of the line and dragging them.

When you draw lines in the sketcher environment of SolidWorks, you will notice that a numeric value is displayed above the line cursor, see Figure 1-16. This numeric value indicates the length of the line that you draw. This value is the same as that in the **Length** spinner of the **Line PropertyManager**. The only difference is that in the **Line PropertyManager**, the value will be displayed with more precision.

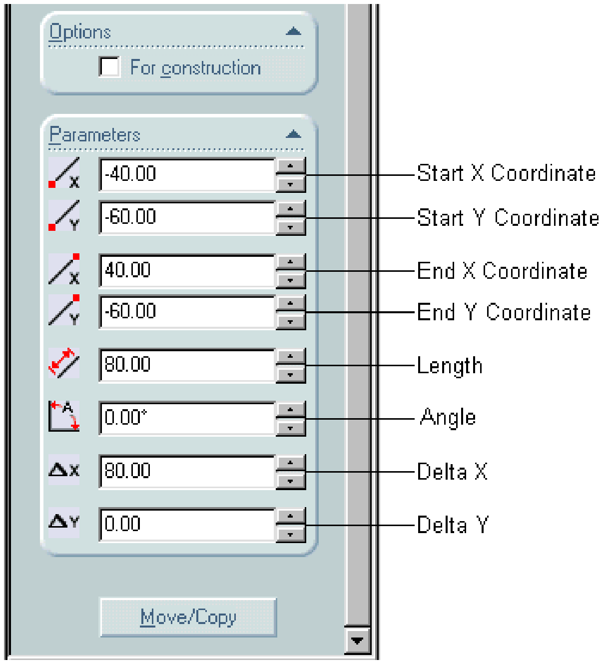


Figure 1-15** Partial display of the **Line PropertyManager

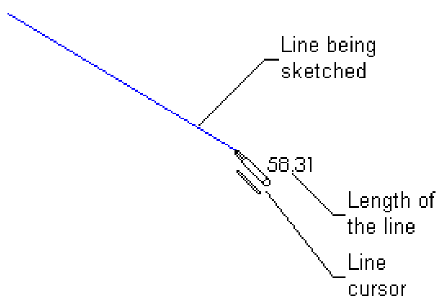


***Figure 1-16** The length of the line displayed on the screen while drawing the line*

The other thing that you will notice while sketching is that sometimes when you are drawing vertical or horizontal lines, a **V** or an **H** symbol is displayed below the line cursor. These are the symbols of the **Vertical** and **Horizontal** relations. SolidWorks automatically applies these relations to the lines. These relations ensure that the lines that you draw are vertical or horizontal and not inclined. Figure 1-17 shows the symbol of the **Vertical** relation on a line and Figure 1-18 shows the symbol of the **Horizontal** relation on a line.

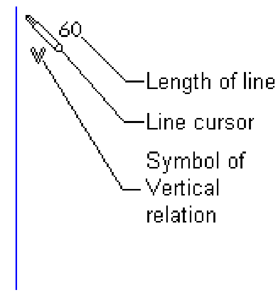


*Figure 1-17 Symbol of the **Vertical** relation*

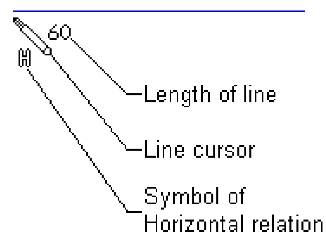
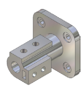


*Figure 1-18 Symbol of the **Horizontal** relation*



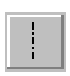
Note

*In addition to the **Horizontal** and **Vertical** relations, you can also apply a number of other relations such as **Tangent**, **Concentric**, **Perpendicular**, **Parallel**, and so on. You will learn about all these relations in later chapters.*

*The other options of the **Line PropertyManager** will be discussed in later chapters.*

Drawing Construction Lines

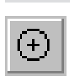
Menu:	Tools > Sketch Entity > Centerline
Toolbar:	Sketch Tools > Centerline



Construction lines are the ones that are drawn only for the aid of sketching. These lines 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. You will notice that when you draw a construction line, the **For Construction** check box in the **Line PropertyManager** is selected. You can also draw a construction line by sketching the line using the **Line** tool and then selecting the **For Construction** check box available in the **Line PropertyManager**. You will notice that when you choose this check box, the line is turned into a centerline.

DRAWING CIRCLES

Menu:	Tools > Sketch Entity > Circle
Toolbar:	Sketch Tools > Circle



In SolidWorks, the circles are drawn by specifying the centerpoint of the circle using the left mouse button and then moving the mouse on the screen to define the radius of the circle. Similar to the lines, as soon as you specify the center of the circle, the **Circle PropertyManager** is displayed. However, note that the options in the **Circle PropertyManager** will be available only after you have defined the radius of the circle. Figure 1-19 shows the **Circle PropertyManager**.

To draw the circle, choose the **Circle** button. You will notice that the arrow cursor is replaced by the circle cursor. The circle cursor consists of a pencil and two concentric circles below the pencil. Specify the centerpoint of the circle and then move the cursor to define the radius of

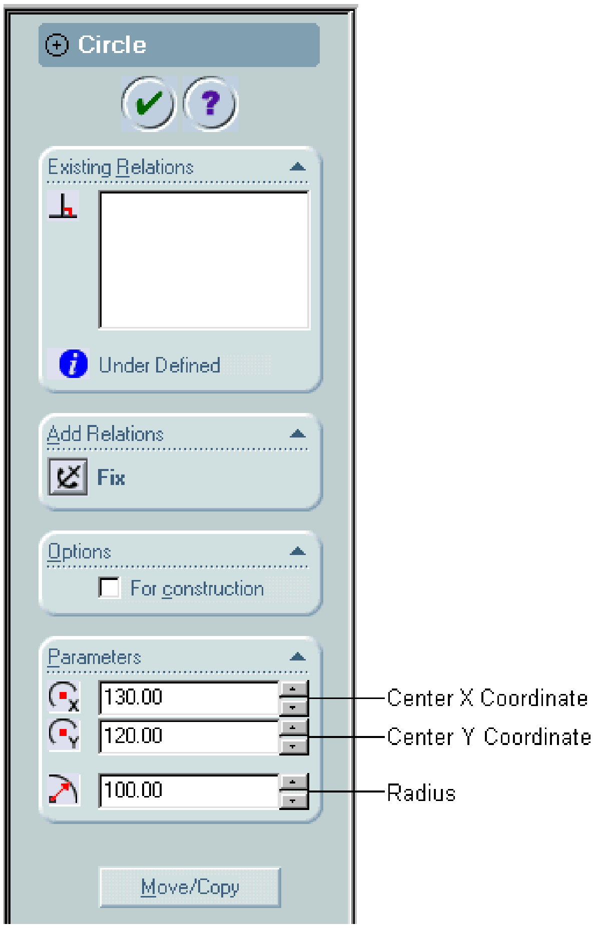
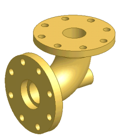


Figure 1-19 Circle PropertyManager

the circle. The current radius of the circle is displayed above the circle cursor. This radius will change as you move the cursor. You can define any arbitrary radius of the circle and then modify it to the desired value by using the **Circle PropertyManager**. Figure 1-20 shows a circle being drawn using the **Circle** tool.

Sketching a Construction Circle

If you want to sketch a construction circle, draw the circle using the **Circle** tool and then select the **For Construction** check box available in the **Circle PropertyManager**.



Tip. *To convert a construction entity back to the sketched entity, invoke the select tool and then select the construction entity. The entity will turn green in color and the **PropertyManager** will be displayed on the left of the drawing window. From the **PropertyManager**, clear the **For Construction** check box. The construction entity will again be changed into a sketched entity and will be displayed with continuous line.*

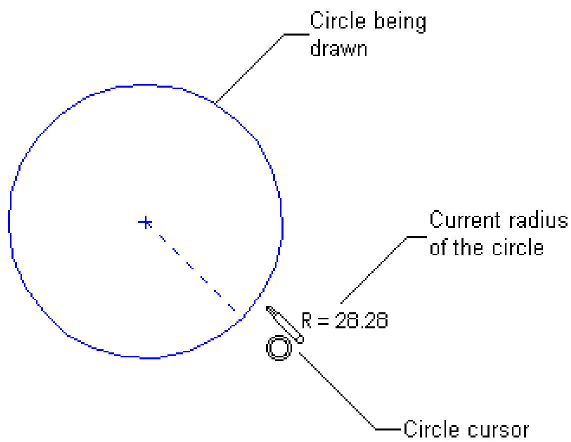


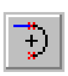
Figure 1-20 Sketching a circle

DRAWING ARCS

In SolidWorks, you can draw the arcs using three methods: **Tangent Arc**, **Centerpoint Arc**, and **3 Point Arc**. All these methods can be invoked separately by choosing their respective buttons from the **Sketch Tools** toolbar. All these three methods to draw arcs are discussed next.

Drawing Tangent Arcs

Menu:	Tools > Sketch Entity > Tangent Arc
Toolbar:	Sketch Tools > Tangent Arc



The tangent arcs are the ones that are drawn tangent to an existing sketched entity. The existing sketched entities include sketched and construction lines, arcs, and splines. As soon as you invoke this tool, the arrow cursor is replaced by the arc cursor. An arc cursor consists of a pencil and an arc below the pencil.

To draw a tangent arc, invoke the **Tangent Arc** tool and then move the arc cursor close to the endpoint of the entity that you want to select as the tangent entity. You will notice that the entity selected as the tangent entity is turned green in color and the color of the pencil in the cursor is changed to yellow. Also, an orange-colored box is displayed below the pencil. This suggests that the endpoint of the entity is selected. Now, press the left mouse button once and move the cursor to size the arc. The arc will start from the endpoint of the tangent entity and its size will change as you move the cursor. Note that the angle and the radius of the tangent arc are displayed above the arc cursor, see Figure 1-21.

As soon as you start moving the cursor, the **Arc PropertyManager** is displayed. However, the options in the **Arc PropertyManager** are not available at this stage. These options are enabled only after you have completed drawing the tangent arc. When you complete a tangent sketch by specifying its endpoint, the **SolidWorks** information box is displayed as shown in Figure 1-22. This dialog box will inform you to select the endpoint of a sketched entity to draw another tangent arc.

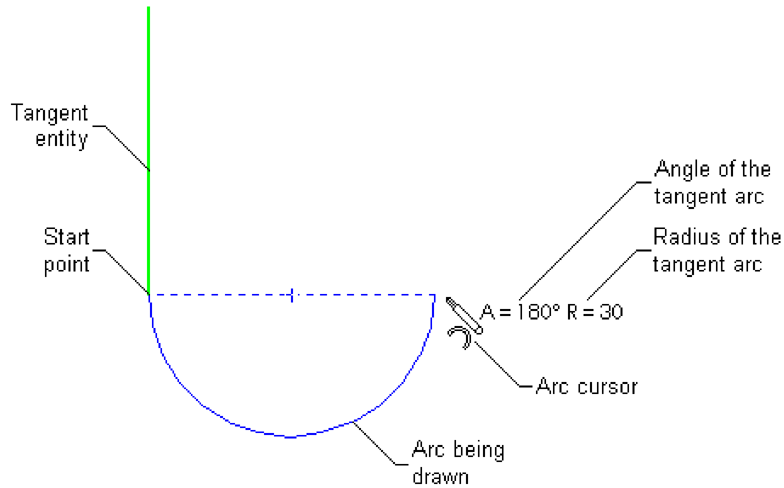


Figure 1-21 *Drawing tangent arc*

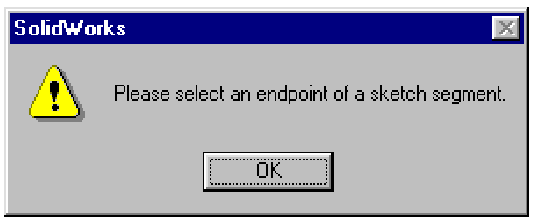
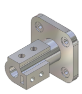


***Figure 1-22 SolidWorks** information box prompting you to select the endpoint of another sketched entity*

You can draw an arbitrary tangent arc and then modify its value using the **Arc PropertyManager**. Figure 1-23 shows a partial view of the **Arc PropertyManager**.

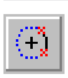


Note

*When you select a tangent entity to draw a tangent arc, the **Tangent** relation is applied between the startpoint of the arc and the tangent entity. Therefore, if you change the coordinates of the startpoint of the arc, the tangent entity will also be modified accordingly.*

Drawing Centerpoint Arcs

Menu:	Tools > Sketch Entity > Centerpoint Arc
Toolbar:	Sketch Tools > Centerpoint Arc



The centerpoint arcs are the ones that are drawn by defining the centerpoint, startpoint, and endpoint of the arc. When you invoke this tool, the arrow cursor is replaced by the arc cursor. As mentioned earlier, an arc cursor consists of a pencil and an arc below the pencil.

To draw a centerpoint arc, invoke the **Centerpoint Arc** tool and then move the arc cursor to the point that you want to specify as the centerpoint of the arc. Press the left mouse button

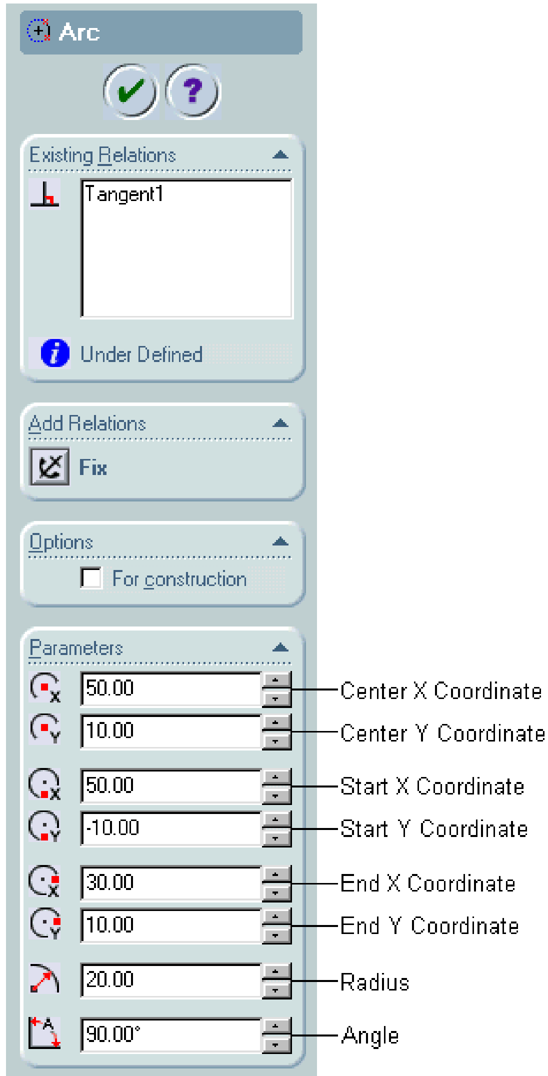


Figure 1-23 Arc PropertyManager

once at the location of the centerpoint and then move the cursor to the point from where you want to start the arc. You will notice that a dotted circle is displayed on the screen. This size of this circle will modify as you move the mouse. This circle is drawn for your reference and the centerpoint of this circle lies at the point that you specified as the center of the arc. Press the left mouse button once at the point that you want to select as the startpoint of the arc. Next, move the mouse to specify the endpoint of the arc. You will notice that the reference circle is no more displayed and an arc is being drawn with the startpoint as the point that you specified after specifying the centerpoint. Also, the **Arc PropertyManager** similar to the one that is shown in the tangent arc is displayed on the left of the drawing window. Note that the options in the **Arc PropertyManager** will not be available at this stage.

If you move the cursor in the clockwise direction, the resultant arc will be drawn in the clockwise direction. However, if you move the cursor in the counterclockwise direction, the resultant arc will be drawn in the counterclockwise direction. Specify the endpoint of the arc using the left mouse button. Figure 1-24 shows the reference circle that is drawn when you move the mouse button after specifying the centerpoint of the arc and Figure 1-25 shows the resultant centerpoint arc.

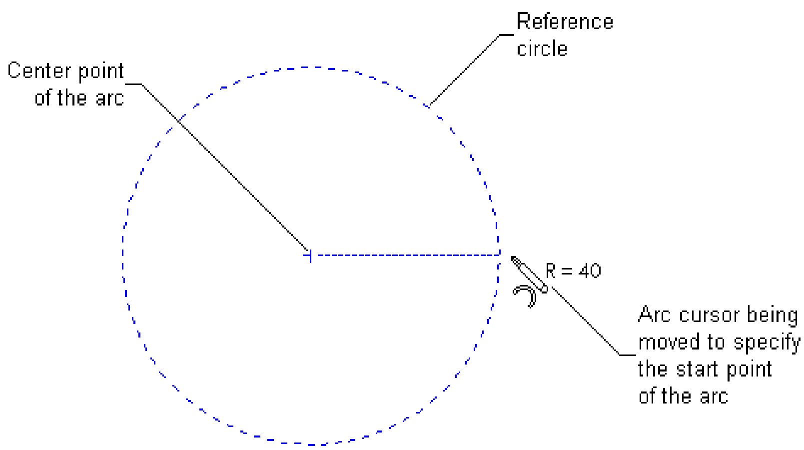


Figure 1-24 *Specifying the centerpoint and the startpoint of the centerpoint arc*

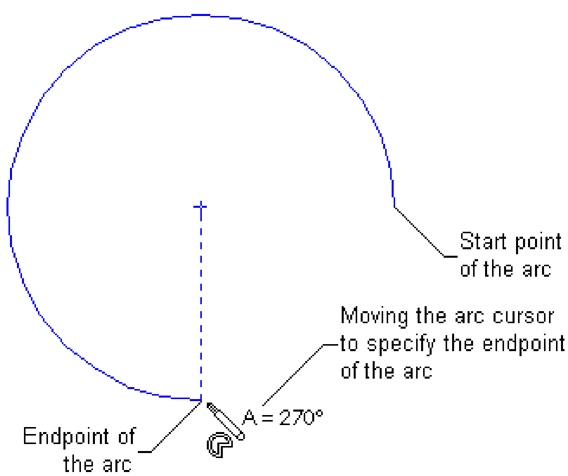
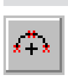


Figure 1-25 *Moving the cursor to specify the startpoint and the endpoint of the arc*

Drawing 3 Point Arcs

Menu:	Tools > Sketch Entity > 3 Point Arc
Toolbar:	Sketch Tools > 3 Pt Arc



The 3 point arcs are the ones that are drawn by defining the startpoint and endpoint of the arc, and a point somewhere on the arc. When you invoke this tool, the arrow cursor is replaced by the arc cursor.

To draw a 3 point arc, invoke the **3 Pt Arc** tool and then move the arc cursor to the point that you want to specify as the startpoint of the arc. Press the left mouse button once at the location of the startpoint and then move the cursor to the point that you want specify as the endpoint of the arc. As soon as you start moving the cursor after specifying the startpoint, a reference arc will be drawn and the **Arc PropertyManager** will be displayed. However, the options in the **Arc PropertyManager** will not be available at this stage.

Using the left mouse button, specify the endpoint of the arc. You will notice that the reference arc is no more displayed. Instead a solid arc is displayed and the cursor is attached to the arc. As you move the cursor, the arc will also be modified dynamically. Using the left mouse button, specify a point on the screen to create the arc. The last point that you specify will determine the direction of the arc. The options in the **Arc PropertyManager** will be displayed after you have drawn the arc. You can modify the properties of the arc using the **Arc PropertyManager**. Figure 1-26 shows the reference arc that is drawn by specifying the startpoint and the endpoint of the arc and Figure 1-27 shows the resultant 3 point arc.

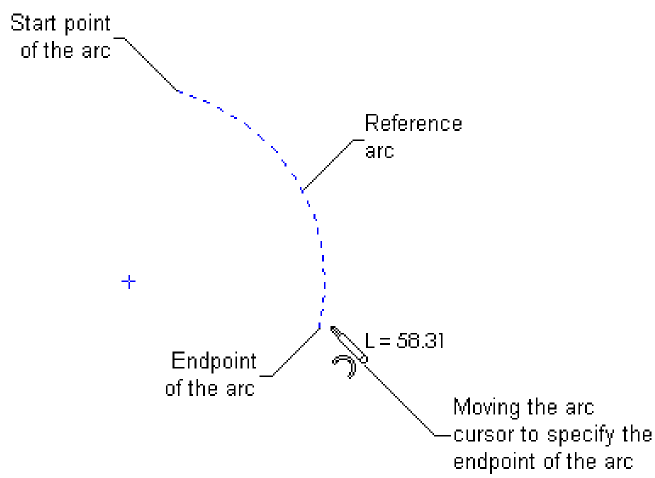


Figure 1-26 *Specifying the startpoint and the endpoint of the arc*

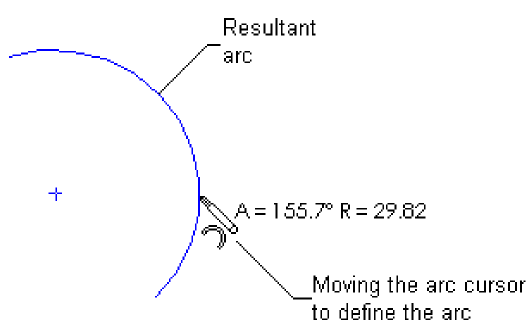


Figure 1-27 *Specifying the point on arc to draw it*

Invoking the Options to Draw an Arc from within the Line Tool

SolidWorks allows you to invoke the options to draw tangent and normal arcs while you are drawing continuous lines using the **Line** tool. Note that the option to draw an arc can be invoked only if at least one sketched line, arc, or spline exists on the screen. To draw an arc tangent to the last line drawn using the current **Line** tool sequence, move the cursor away from the endpoint of the line and then move it back close to the endpoint. You will notice that the line cursor is replaced by the arc cursor and the **Line PropertyManager** is replaced by the **Arc PropertyManager**. Similarly, if you want to draw an arc tangent to an existing sketched entity using the **Line** tool, invoke this tool and select the endpoint of the sketched entity. Now, move the cursor away from the endpoint and then move it back close to the endpoint. You will switch to the arc mode.

Now, if you want to draw an arc tangent to the previous line, move the cursor along the line through a small distance. A dotted line will be drawn. Next, move the cursor in the direction in which the arc should be drawn. You will notice that a tangent arc is drawn. Specify the endpoint of the tangent arc using the left mouse button. Figure 1-28 shows an arc tangent to an existing line.

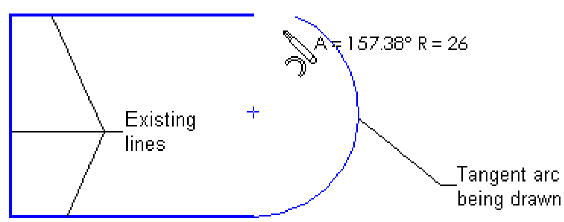


***Figure 1-28** Drawing a tangent arc using the **Line** tool*

If you want to draw an arc normal to the previous line, move the cursor away from the endpoint and then move it back to the endpoint to switch to the arc mode. Next, move the cursor normal to the existing line and then move the cursor in the direction in which the arc should be drawn. You will notice that a normal arc is being drawn. Figure 1-29 shows a normal arc being drawn using the **Line** tool.

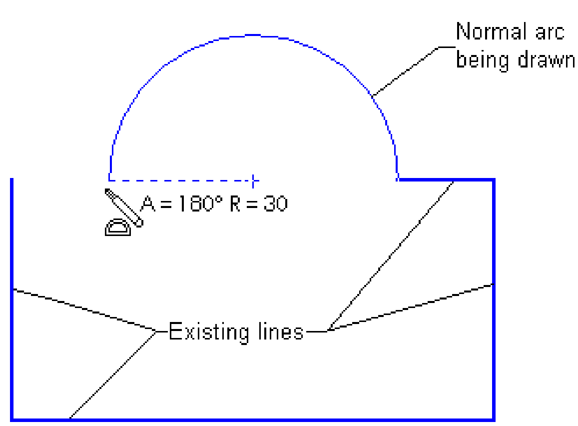
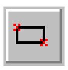


***Figure 1-29** Drawing a normal arc using the **Line** tool*

After drawing an arc using the **Line** tool, the next entity that will be drawn in the same sequence is the line. If you want to draw another arc, use the procedure that was followed while drawing the first arc. Note that if you want to switch back to the line mode, click once at the endpoint of the previous entity using the left mouse button. You can also toggle between the line and the arc mode by pressing the A key from the keyboard or by using the option in the shortcut menu that is displayed when you right-click in the drawing area.

DRAWING RECTANGLES

Menu:	Tools > Sketch Entity > Rectangle
Toolbar:	Sketch Tools > Rectangle



In SolidWorks, the rectangles are drawn by specifying two opposite corners of the rectangle. To draw a rectangle, invoke the **Rectangle** tool. The arrow cursor will be replaced by the rectangle cursor. Move the cursor to the point that you want to specify as the first corner of the rectangle. Press the left mouse button once at the first corner and then move the cursor and specify the other corner of the rectangle using the left mouse button. You will notice that the length and width of the rectangle are displayed above the rectangle cursor. The length is measured along the X axis and the width is measured along the Y axis. Figure 1-30 shows a rectangle being drawn by specifying two opposite corners.

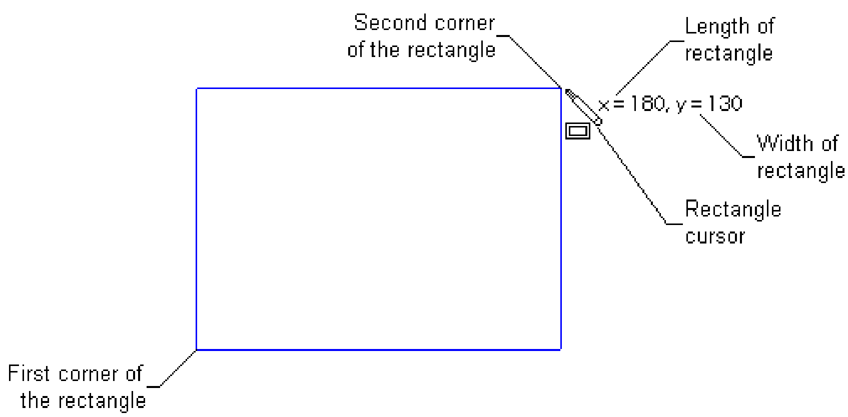


Figure 1-30 *Drawing a rectangle by specifying two opposite corners*

Note

*When you draw a rectangle, the **PropertyManager** will not be displayed. This is because a rectangle is considered as a combination of four individual lines. Therefore, after drawing the rectangle, if you select one of the lines of the rectangle using the **Select** tool, the **Line PropertyManager** will be displayed. You can modify the parameters of the selected line using the **Line PropertyManager**.*

*Remember that since the relations are applied to all the four corners of the rectangle, if you modify the parameters of one of the lines using the **Line PropertyManager**, the other three lines will also be modified accordingly.*

*You can convert a rectangle into a construction rectangle by selecting all the lines together using a window and then selecting the **For Construction** check box from the **PropertyManager**.*

DRAWING PARALLELOGRAM

Menu: Tools > Sketch Entity > Parallelogram

In SolidWorks, the **Parallelogram** tool can be used to draw a parallelogram and also to draw a rectangle at an angle. The methods to draw both these entities are discussed next.

Drawing a Rectangle at an Angle

A rectangle at an angle is drawn by first defining one of the edges of the parallelogram using two points. Since the edge is defined using two points, you can specify the points at an angle, thus forcing the edge to be at an angle. After defining one of the edges at an angle, you will move the cursor to define the width of the rectangle. The width of the rectangle will be defined normal to the first edge. Therefore, you will specify a total of three points in defining a parallelogram.

To draw a rectangle at an angle, invoke the **Parallelogram** tool from the menu bar. The cursor will be replaced by the parallelogram cursor. Move the cursor to the point that you want to select as the startpoint of one of the edges of the rectangle. Press the left mouse button once at this point and move the cursor to size the edge. You will notice that a reference line is being drawn. Based on the current position of the cursor, the reference line will be horizontal, vertical, or aligned. The current length of the edge and its angle will be displayed above the parallelogram cursor. Using the left mouse button, specify the endpoint of the edge such that the resultant reference line is at an angle.

Next, move the cursor to specify the width of the rectangle. You will notice that a reference rectangle is drawn at an angle. Also, irrespective of the current position of the cursor, the width will be specified normal to the first edge, either above or below. Using the left mouse button, specify a point on the screen to define the width of the rectangle. The reference rectangle will be converted into a sketched rectangle. Figure 1-31 shows a rectangle drawn at an angle.

Drawing Parallelograms

The process of drawing a parallelogram is similar to that of drawing a rectangle at an angle.

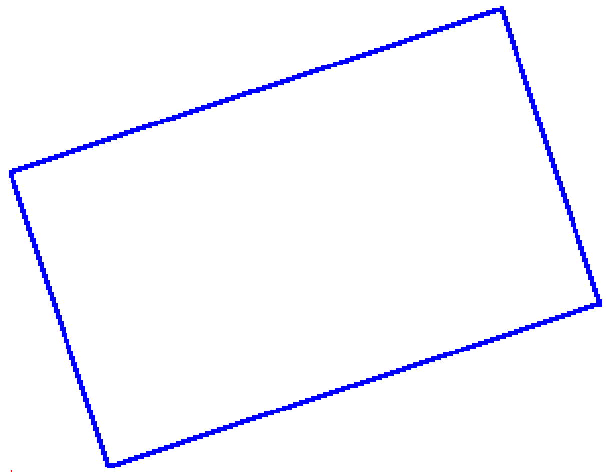
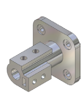


Figure 1-31 *Rectangle at an angle*

The only difference is that while specifying the third point to define the width, press the CTRL key from the keyboard. The width will no more be added normal to the first edge. Therefore, you can draw a parallelogram.

To draw a parallelogram, invoke the **Parallelogram** tool from the menu bar. The cursor will be replaced by the parallelogram cursor. Specify two points on the screen to define one edge of the parallelogram. Next, press the CTRL key from the keyboard once and then move the mouse to define the width of the parallelogram. You will notice that the width is no more added normal to the first edge. As you move the mouse, a reference parallelogram will be drawn. The size and the shape of the reference parallelogram will depend upon the current location of the cursor.

Specify the point on the screen to define the parallelogram. Figure 1-32 shows a parallelogram drawn at an angle.

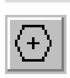


Note

Similar to the rectangles, each edge of a parallelogram is considered as a separate line. Also, in case of parallelograms, the ***PropertyManager*** *is not displayed while you are drawing it.*

DRAWING POLYGONS

Menu:	Tools > Sketch Entity > Polygon
Toolbar:	Sketch Tools > Polygon



A polygon is defined as a multisided geometric figure in which the length of all the sides and the angle between all the sides are the same. In SolidWorks, you can draw a polygon with the number of sides ranging from 3 to 40. The dimensions of a polygon are controlled using the diameter of a construction circle that is either inscribed inside the polygon or circumscribed about the polygon. If the construction circle is inscribed inside the

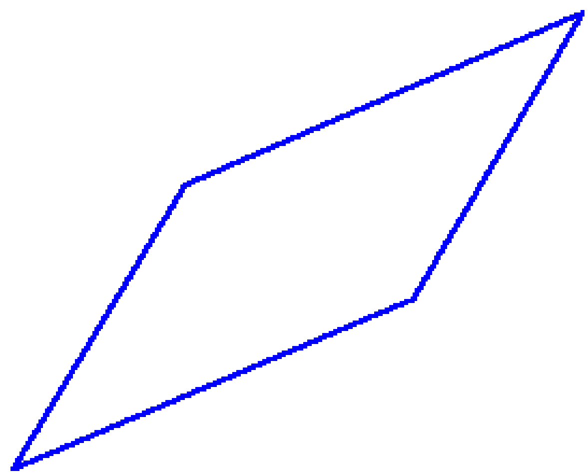


Figure 1-32 *Parallelogram at an angle*

polygon, the diameter of the construction circle is taken from the edges of the polygon. If the construction circle is circumscribed about the polygon, the diameter of the construction circle is taken from the vertices of the polygon.

To draw a polygon, invoke the **Polygon** tool. When you invoke this tool, the **Polygon PropertyManager** will be displayed as shown in Figure 1-33. Set the parameters such as the number of sides, inscribed or circumscribed circle, and so on in the **Polygon PropertyManager**. You can also modify these parameters after drawing the polygon. When you invoke this tool, the arrow cursor will be replaced by the polygon cursor. Press the left mouse button at the point that you want to select as the centerpoint of the polygon and then move the cursor to size the polygon. The length of each side and the rotation angle of the polygon will be displayed above the polygon cursor as you drag it. Using the left mouse button, specify a point on the screen after you get the desired length and rotation angle of the polygon. You will notice that based on whether you selected the **Inscribed circle** or the **Circumscribed circle** radio button in the **Polygon PropertyManager**, a construction circle will be drawn inside or outside the polygon. After you have drawn the polygon, you can modify the parameters such as the centerpoint of the polygon, the diameter of the construction circle, the angle of rotation of the polygon, and so on using the **Polygon PropertyManager**. If you want to draw another polygon, choose the **New polygon** button provided below the **Angle** spinner in the **Polygon PropertyManager**.

Figure 1-34 shows a six-sided polygon with the construction circle inscribed inside the polygon and Figure 1-35 shows a five-sided polygon with the construction circle circumscribed outside the polygon. Notice that the reference circle is retained with the polygon. Remember that this circle will not be considered while converting the polygon into a feature.

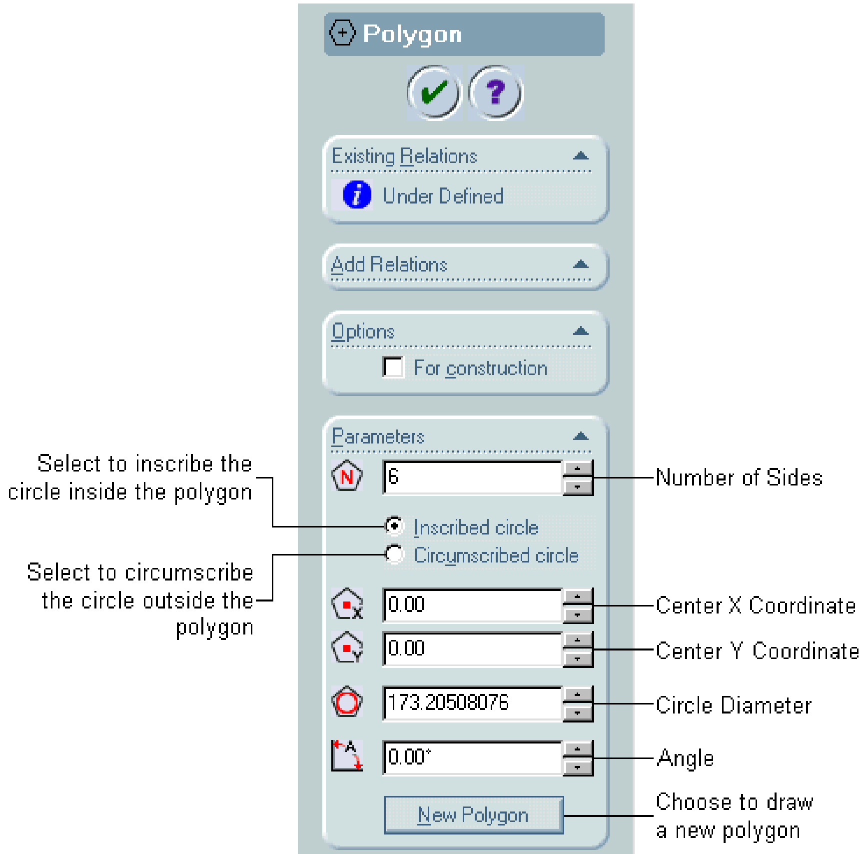


***Figure 1-33** Polygon PropertyManager*

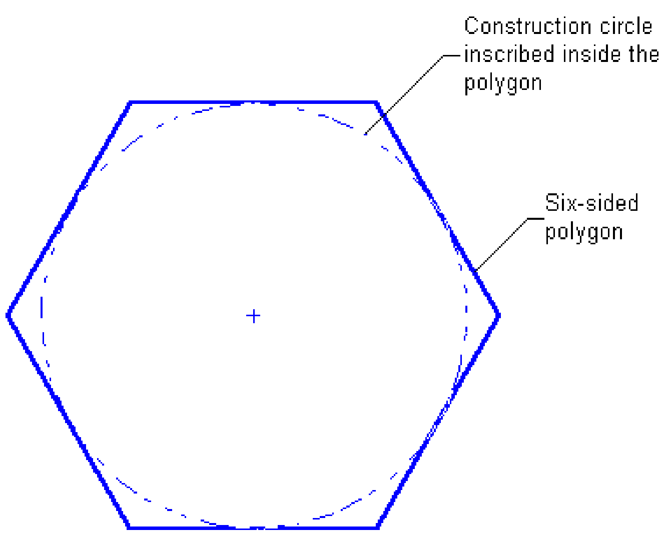


***Figure 1-34** Six-sided polygon with construction circle inscribed inside the polygon*

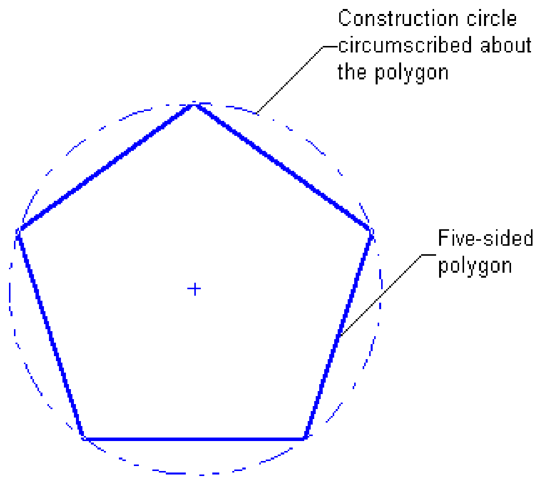
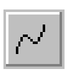


Figure 1-35 *Five-sided polygon with construction circle circumscribed outside the polygon*

DRAWING SPLINES

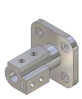
Menu:	Tools > Sketch Entity > Spline
Toolbar:	Sketch Tools > Spline



In SolidWorks, the splines can be drawn using two methods. In the first method, which is the default method, you can draw a spline by continuously specifying the endpoints of the spline segments using the left mouse button. This method of drawing splines is similar to the method of drawing continuous lines.

In the second method of drawing a spline, you have to specify the first point of the spline and then press and hold the left mouse button and drag the cursor to define the second point of the spline. After specifying the second point, release the left mouse button. One segment of the spline will be drawn. To draw the next segment, move the cursor close to the endpoint of the first spline segment. The pencil in the spline cursor will turn yellow in color and an orange-colored box will be displayed below the pencil. This suggests that the endpoint is selected. When the orange box is displayed, press and hold the left mouse button down and drag the cursor. The endpoint of the last segment will be taken as the startpoint of the second segment and the point where you release the cursor will be taken as the endpoint of the second segment. Repeat the procedure to draw as many segments of the spline.

The number of points you specify in a spline are taken as the handles of the spline. Therefore, after drawing a spline, if you select it using the **Select** tool, all the points that you specified are displayed on the spline inside square boxes. These points are called the handles or the control points and you can modify the shape of a spline using these handles. Figure 1-36 shows a sketched spline.



Note

Irrespective of the number of segments in a spline, it will be considered as a single entity.

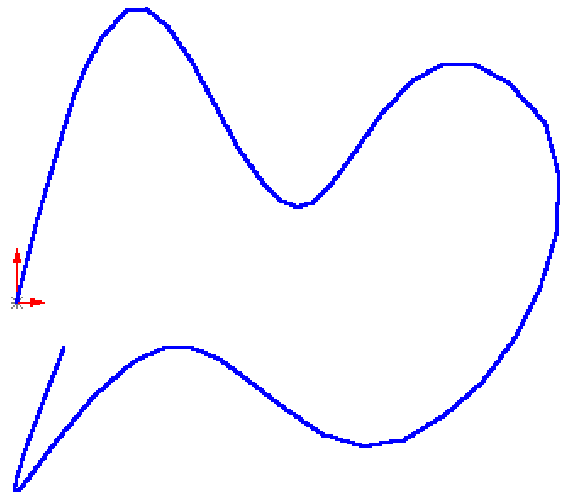
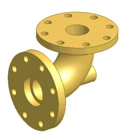


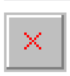
Figure 1-36 Sketched spline with startpoint at the origin



Tip. *As you start drawing a spline, the* ***Spline PropertyManager*** *is displayed. However, the options in it are not available. These options will be available when you select an existing spline using the* ***Select*** *tool. The current handle will be displayed with a filled square and its number and the corresponding X and Y coordinates will be displayed in the* ***Spline PropertyManager****. You can modify these coordinates to modify the selected spline.*

DRAWING POINTS

Menu:	Tools > Sketch Entity > Point
Toolbar:	Sketch Tools > Point



To draw a point, choose the **Point** tool and then specify the point on the screen where you want to place the point. The **Point PropertyManager** will be displayed with the X and Y coordinates of the current point. You can modify the location of the point by modifying its X and Y coordinates in the **Point PropertyManager**.

DRAWING ELLIPSES

Menu:	Tools > Sketch Entity > Ellipse

In SolidWorks, the ellipse is drawn by specifying the centerpoint of the ellipse and then specifying the two ellipse axes by moving the mouse. To draw an ellipse, invoke this tool from the menu bar. The arrow cursor will be replaced by the ellipse cursor. Move the cursor to the point that you want to select as the centerpoint of the ellipse. Press the left mouse button once at the centerpoint of the ellipse and then move the cursor to specify one of the ellipse axis. You will notice that a reference circle is drawn and two values are displayed above the ellipse cursor, see Figure 1-37. The first value that shows R = * is the radius of the first axis that you are defining and the second value that shows r = * is the radius of the other axis.

While you are defining the first axis, the second axis is taken equal to the first axis. This is the reason a reference circle is drawn and not a reference ellipse.

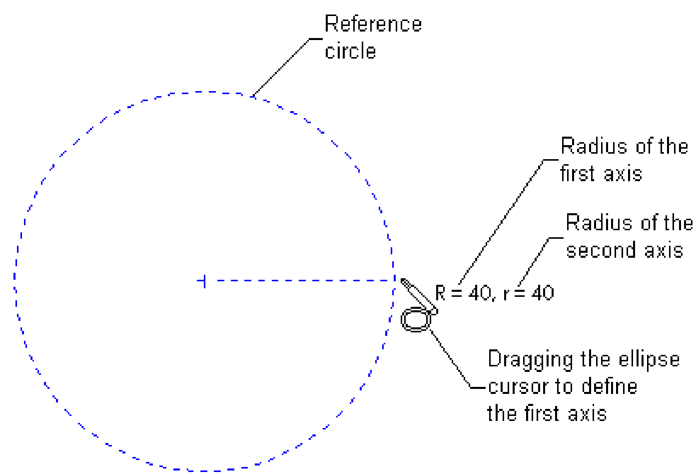


Figure 1-37 *Dragging the cursor to define the first axis*

Specify a point on the screen to define the first axis. Next, move the cursor to size the other ellipse axis. You will notice that the **Ellipse PropertyManager** is displayed. Figure 1-38 shows a partial view of the **Ellipse PropertyManager**.

The second value above the ellipse cursor that shows r = * will change dynamically as you move the cursor on the screen. Using the left mouse button, specify a point on the screen to define the second axis of the ellipse, see Figure 1-39.

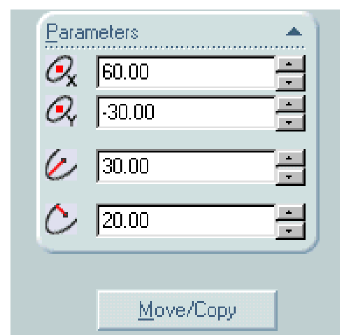


Figure 1-38 *Partial view of the* ***Ellipse PropertyManager***

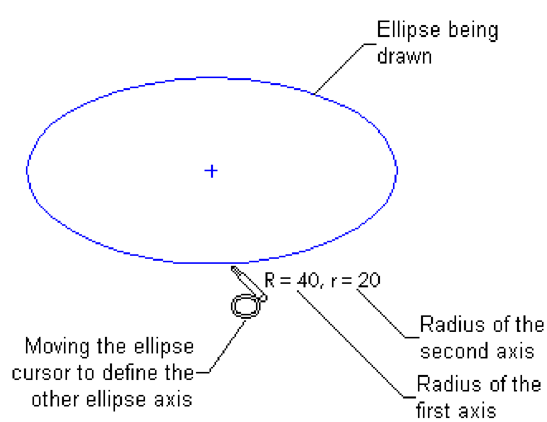


Figure 1-39 *Define the second axis of the ellipse*

DRAWING ELLIPTICAL ARCS

Menu:	Tools > Sketch Entity > Centerpoint Ellipse

In SolidWorks, the process of drawing an elliptical arc is similar to that of drawing an ellipse. You will follow the same process of defining the ellipse first. The point that you specify on the screen to define the other axis of the ellipse is taken as the startpoint of the elliptical arc. You can define the endpoint of the elliptical arc by specifying a point on the screen as shown in Figure 1-40.

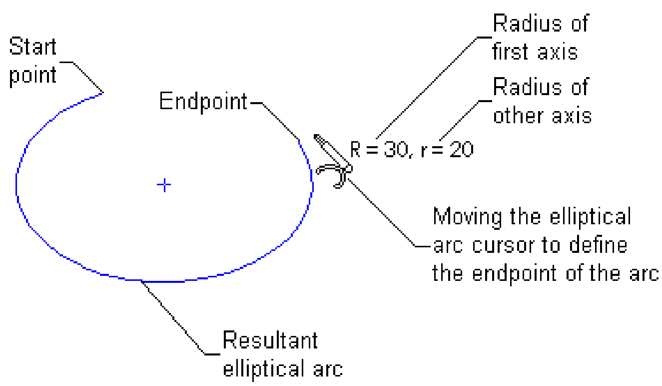


Figure 1-4*0 Drawing the elliptical arc*

You can also set the parameters of the elliptical arc in the **Ellipse PropertyManager** shown in Figure 1-41.

DRAWING A PARABOLIC CURVE

Menu:	Tools > Sketch Entity > Parabola

In SolidWorks, you will draw a parabolic curve by specifying the focal point of the parabola and then specifying two points on the guide lines of the parabolic curve. To draw a parabolic curve, invoke this tool from the menu bar. The cursor will be replaced by the parabola cursor. Move the cursor to the point that you want to specify as the focal point of the parabola. Press the left mouse button once at the focal point and then move the cursor to define the apex point and size the parabola. You will notice that a reference parabolic arc is displayed. As you move the cursor away from the focal point, the parabola is flattened. After you get the basic shape of the parabolic curve, specify a point on the screen using the left mouse button. This point is taken as the apex of the parabolic curve. Next, specify two point on the screen with

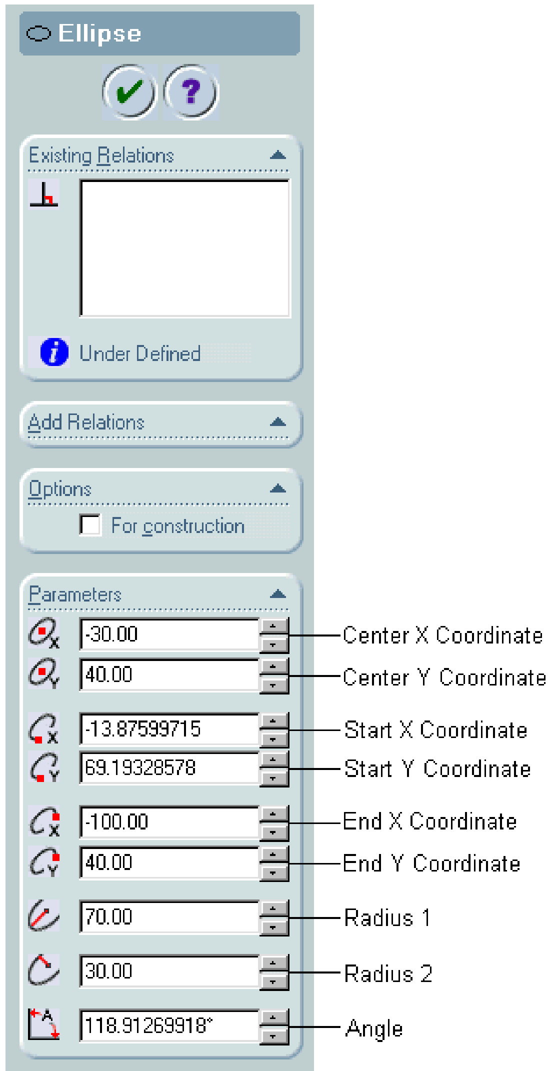


Figure 1-41 Ellipse PropertyManager

respect to the reference parabola to define the guide lines of the parabolic curve, see Figure 1-42.

As you move the mouse after specifying the focal point of the parabola, the **Parabola PropertyManager** will be displayed. But the options in the **Parabola PropertyManager** will not be available. These options will be available only after you have drawn the parabola. Figure 1-43 shows a partial view of the **Parabola PropertyManager**.

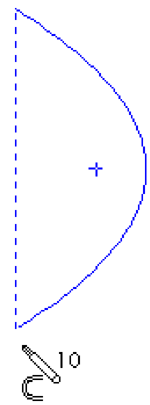


Figure 1-42 Drawing the parabola

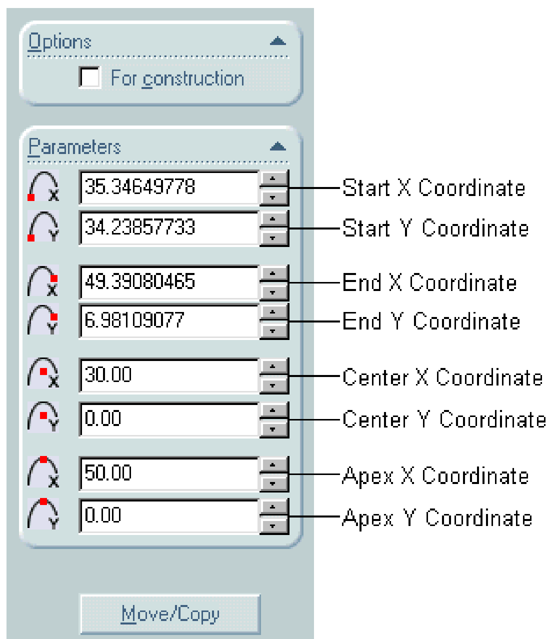


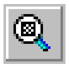
*Figure 1-43 Partial view of the **Parabola PropertyManager***

DRAWING DISPLAY TOOLS

The drawing display tools are one of the most important tools provided in any of the solid modeling software. These tools allow you to modify the display of the drawing by zooming or panning the drawing. Some of the drawing display tools that are available in SolidWorks are discussed in this chapter. The remaining tools are discussed in the later chapters.

Zoom to Fit

Menu:	View > Modify > Zoom to Fit
Toolbar:	View > Zoom to Fit



The **Zoom to Fit** tool is used to increase or decrease the drawing display area so that all the sketched entities or dimensions are fitted inside the current view.

Zoom to Area

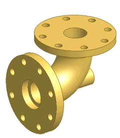
Menu:	View > Modify > Zoom to Area
Toolbar:	View > Zoom to Area

The **Zoom to Area** tool is used to magnify a specified area so that the part of the drawing inside the magnified area can be viewed in the current window. The area is defined by a window that is created by dragging the cursor and specifying two opposite corners of the window. When you choose this tool, the cursor is replaced by a magnifying glass cursor. Press and hold the left mouse button down and drag the cursor to specify two opposite corners of the window. The area enclosed inside the window will be magnified.

Zoom In/Out

Menu:	View > Modify > Zoom In/Out
Toolbar:	View > Zoom In/Out

The **Zoom In/Out** tool is used to dynamically zoom in or out of the drawing. When you invoke this tool, the cursor is replaced by the zoom cursor. To zoom out of a drawing, press and hold the left mouse button down and drag the cursor in the downward direction. Similarly, to zoom in a drawing, press and hold the left mouse button down and drag the cursor in the upward direction. As you drag the cursor, the drawing display will be modified dynamically. After you get the desired view, exit this tool by choosing the **Select** tool from the **Sketch** toolbar. You can also exit this tool by right-clicking and choosing **Select** from the shortcut menu or by pressing the ESC key.



Tip. *You can also use the keyboard shortcuts to invoke some of the drawing display tools. For example, to invoke the* ***Zoom to Fit*** *tool, press the F key. Similarly, to zoom out of a drawing, press the Z key and to zoom in, press SHIFT+Z key.*

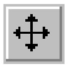
Zoom to Selection

Menu:	View > Modify > Zoom to Selection
Toolbar:	View > Zoom to Selection

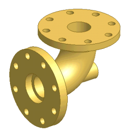
The **Zoom to Selection** tool is used to modify the drawing display area such that the selected entity is fitted inside the current display. This tool will be available only when you select an entity using the **Select** tool. After selecting the entity, choose the **Zoom to Selection** button. The drawing display area will be modified such that the selected entity fits inside the current view.

Pan

Menu:	View > Modify > Pan
Toolbar:	View > Pan



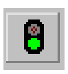
The **Pan** tool is used to drag the view in the current display. This process is similar to changing the view by using the scroll bars available in the drawing window.



Tip. *You can also invoke the* ***Pan*** *tool using the CTRL key and the arrow keys on the keyboard. For example, to pan toward the right, press the CTRL key and then press the right arrow key a few times. Similarly, to pan upward, press the CTRL key and then press the up arrow key a few times.*

Redraw

Menu:	View > Redraw
Toolbar:	Standard > Rebuild



The **Redraw** tool is used to refresh the screen. Sometimes when you draw a sketched entity, some unwanted elements remain on the screen. To remove these unwanted elements from the screen, choose this tool. The screen will be refreshed and all the unwanted elements will be removed. You can also invoke this tool by pressing CTRL+R keys from the keyboard.

DELETING THE SKETCHED ENTITIES

You can delete the sketched entities by selecting them using the **Select** tool and then pressing the DELETE key from the keyboard. You can select the entities by picking them individually or select more than one entity by defining a window around the entities. When you select the entities, they turn green in color. When they turn green, press the DELETE key from the keyboard. You can also delete the sketched entities by selecting them and choosing the **Delete** option from the shortcut menu that is displayed upon right-clicking.

TUTORIALS

Tutorial 1

In this tutorial you will draw the sketch of the model shown in Figure 1-44. The sketch is shown in Figure 1-45. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

The steps that will be followed to complete this tutorial are listed below:

a. Start SolidWorks and then open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Draw the sketch of the model using the **Line** and the **Circle** tools, refer to Figure 1-48 through Figure 1- 50.
d. Save the sketch and then close the file.

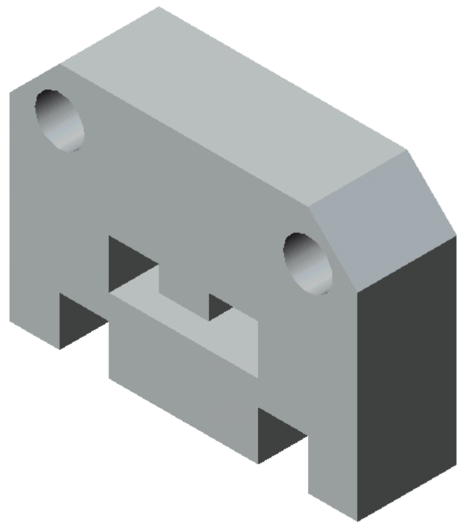


Figure 1-44 *Solid model for Tutorial 1*

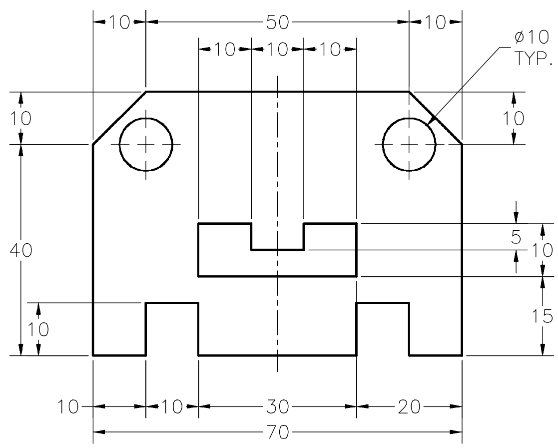


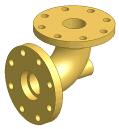
Figure 1-45 *Sketch of the model*

Starting SolidWorks and Opening a New Part Document

1. Start SolidWorks by choosing **Start > Programs > SolidWorks 2003 > SolidWorks 2003** or by double-clicking the shortcut icon of SolidWorks available on the desktop of your computer.

   The **Tip of the Day** dialog box will be displayed. If the **Tip of the Day** dialog box is not displayed when you start the SolidWorks session then you need to set the option to display the **Tip of the Day** dialog box when you start the new SolidWorks session.

2. Choose **Help** > **Tip of the Day** from the menu bar. The **Tip of the Day** dialog box is displayed. Select the **Show tips at startup** check box from this dialog box.



Tip. *If the shortcut icon of SolidWorks is not created automatically on the desktop of your system when you install SolidWorks, you can create it manually. To create the shortcut icon on the desktop, choose* ***Start > Programs > SolidWorks 2003*** *to display the SolidWorks cascading menu. Right-click* ***SolidWorks 2003*** *in the cascading menu and then choose* ***Send To > Desktop (create shortcut)*** *from the shortcut menu.*

By selecting the **Show tips at startup** check box, the **Tip of the Day** dialog box will be displayed every time you start the SolidWorks session. You can get many valuable tips from the **Tip of the Day** dialog box. The tips displayed in this dialog box are helpful to make the full utilization of this CAD package.

3. Close the **Tip of the Day** dialog box by choosing the **Close** button. The **Welcome to SolidWorks 2003** window is displayed. Choose the **New Document** option from this window.

   The **New SolidWorks Document** dialog box is displayed.

4. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box as shown in Figure 1-46.

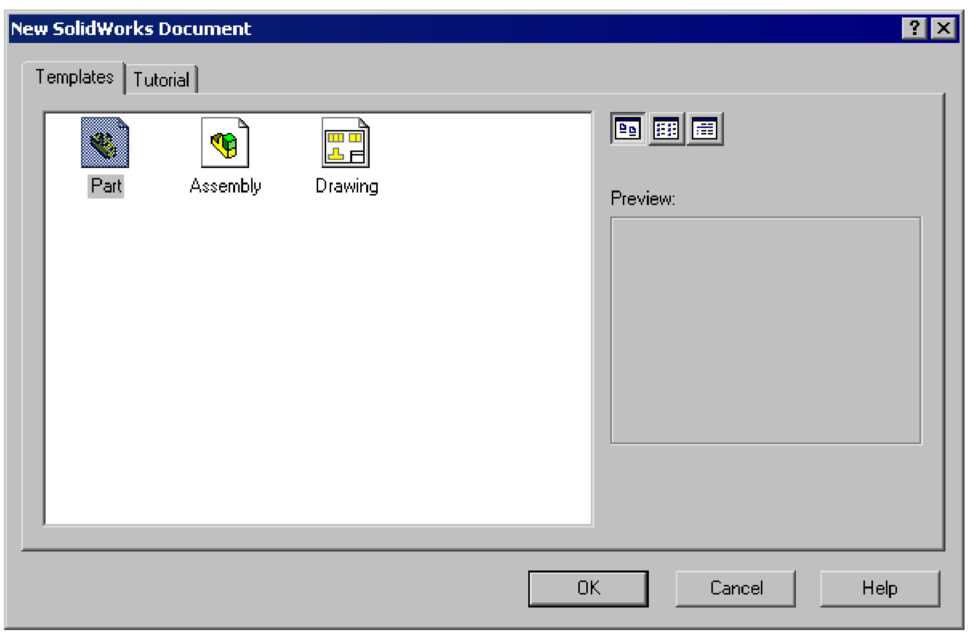


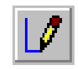
Figure 1-46 New SolidWorks Document *dialog box*

A new SolidWorks part document will be opened. But the part document window will not be maximized in the SolidWorks window.

5. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.

   When you open a new part document, the part modeling environment is active by default. Since you first need to draw the sketch of the feature, you need to invoke the sketching environment.

6. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.



   You will notice that the origin that was displayed with gray color earlier is changed to red color, indicating the sketching environment. The default screen appearance of the sketching environment of SolidWorks is shown in Figure 1-47.

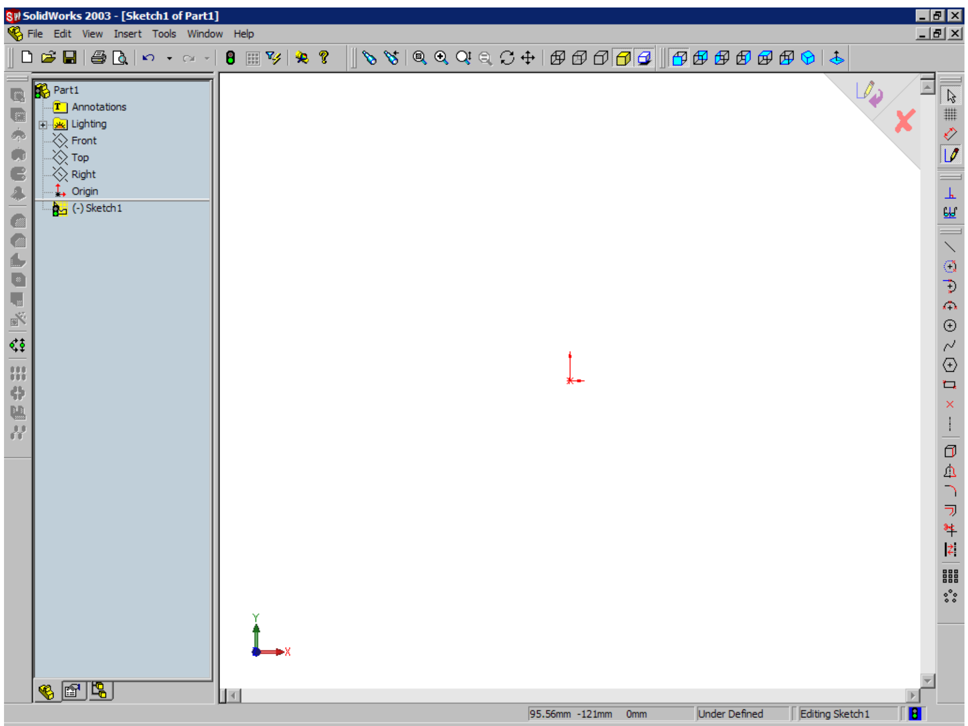
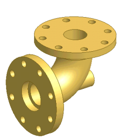


***Figure 1-47** Screen display in the sketcher environment*



Tip. *If the grid is displayed on the screen when you invoke the sketching environment for the first time, then you can set the option to turn off the grid display. Choose **Tools > Options** from the menu bar. The **System Options - General** dialog box is displayed. Choose the **Document Properties** tab from this dialog box. Select the **Grid/Snap** option from the left of this dialog box. Clear the **Display grid** check box from the **Grid** area and choose the **OK** button from this dialog box.*

Setting the Units and Grid

It is assumed that while installing SolidWorks, you selected the option of measuring the length in millimeters. This is the reason the length will be measured in millimeter in the current file. However, if you selected some other units, you need to make some initial settings of changing the linear and angular units before you proceed with drawing the sketch.

1. Choose **Tools > Options** from the menu bar to invoke the **System Options - General** dialog box.

2. Choose the **Document Properties** tab. The name of the dialog box will be changed to the **Document Properties - Detailing** dialog box.

3. Select the **Units** option from the area on the left to display the options related to linear and angular units.

4. Select **Millimeters** from the drop-down list available in the **Linear units** area. Also, select the **Degrees** option from the drop-down list provided in the **Angular units** area.

Note

*If you selected **Millimeters** as the units while installing SolidWorks, you can skip the points discussed earlier in this section.*

5. Select **Grid/Snap** from the area on the left. Set the value of the **Major grid spacing** spinner to **100** and the value of the **Minor-lines per major** spinner to **10**.

6. Select the **Snap to points** check box if it is cleared. Choose **OK** to exit the dialog box.

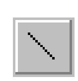
Drawing the Outer Loop of the Sketch

It is a good practice to draw the sketch on one side of the origin, preferably in the first quadrant. This is because while generating the part program for manufacturing the part, you will have a reference for work origin in advance.

The sketch of the model consists of an outer loop, two circles inside the outer loop, and a cavity. Therefore, it will be drawn using the **Line** and the **Circle** tools. You will first draw the outer loop and then the inner entities. Note that in the sketcher environment, the lower right corner of the SolidWorks window displays three areas. The first area displays the X, Y, and Z coordinates of the current location of the cursor. These coordinates will modify as you move the cursor around the drawing area. You will use the coordinate display to draw the sketch of the model.

You will start drawing the sketch from the lower left corner of the sketch and the outer loop will be drawn using the continuous lines.

1. Choose the **Line** button from the **Sketch Tools** toolbar to invoke the **Line** tool.



The arrow cursor will be replaced by the line cursor.

2. Move the cursor in the first quadrant close to the origin. The coordinates of the point will be displayed close to the lower right corner of the screen.

3. Press the left mouse button at the point whose coordinates are 10mm 10mm 0mm and then move the cursor horizontally toward the right.

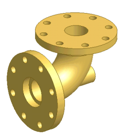
   You will notice that the symbol of **Horizontal** relation is displayed below the line cursor and the length of the line is displayed above the line cursor.

   Since the length of the first horizontal line at the lower left corner is 10mm, you will move the mouse until the length of the line is shown as 10 above the line cursor.

4. Press the left mouse button when the length of the line that is displayed above the line cursor shows a value of 10.

   The first horizontal line is drawn. Since you are drawing continuous lines, the endpoint of the last line will be automatically selected as the startpoint of the next line.

5. Move the line cursor vertically upward. The symbol of **Vertical** relation will be displayed below the line cursor and the length of the line will be displayed above the line cursor.



Tip. *If by mistake you invoke the arc mode while drawing lines, move the cursor back to the endpoint of previous line and press the left mouse button. The line mode will be invoked again.*

6. Press the left mouse button when the length of the line displayed above the line cursor shows a value of 10.

   A vertical line of length 10mm will be drawn and will be displayed in green color. Also, since this is the line that is selected, the previous line will no more be highlighted and therefore will be displayed in blue color.

7. Move the line cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of 10.

   This draws the next horizontal line of 10mm length.

8. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor shows a value of 10.

9. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of 30.

10. Move the line cursor vertically upwards and press the left mouse button when the length of the line on the line cursor shows a value of 10.

11. Move the line cursor horizontally toward the right and press the left mouse button when

the length of the line on the line cursor shows a value of 10.

12. Move the line cursor vertically downwards and press the left mouse button when the length of the line on the line cursor shows a value of 10.

13. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of 10.

14. Move the line cursor vertically upwards and press the left mouse button when the length of the line on the line cursor displays a value of 40.

    The next line that you will draw is an inclined line that makes an angle of 135-degree. To draw this line, you will move the cursor in a direction that makes an angle of 135-degrees. It is not necessary to make sure that the line you draw is exact. You can draw a line at some other angle and then modify the values from the **Line PropertyManager**.

    The aligned length of the line is not given for this line. Instead, the delta X and delta Y values are given. Therefore, you will draw a line at this point such that the delta X and delta Y value of this line is 10mm. These values can be viewed in the **Line PropertyManager**. The last spinners in the **Line PropertyManager** are for the delta values.

15. Move the line cursor such that the line is drawn at an angle of 135-degree. The current angle will be displayed in the spinner above the delta spinners in the **Line PropertyManager**.

16. Press the left mouse button when the delta X value in the **Delta X** spinner shows a value of 10 and the value of delta Y in the **Delta Y** spinner displays a value of 10 in the **Line PropertyManager**.

    The length of the line will be displayed as 14.14 above the line cursor and also in the **Length** spinner in the **Line PropertyManager**.

17. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 50.

    You will notice that some blue and brown inferencing lines are displayed when you move the cursor.

18. Move the line cursor in the direction of 225-degree angle.

    You will notice that two blue inferencing lines are displayed. The first one originates from the startpoint of the first inclined line and the other one from the startpoint of the first line of the sketch.

19. Press the left mouse button at the point where the two inferencing lines intersect.

    You will notice that at this point, the length of the line on the line cursor shows a value of

14.14. Also, the delta X and delta Y values in the **Line PropertyManager** are displayed as 10 each and the angle is shown as 225.00-degree.

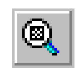
20. Move the cursor vertically downwards to the startpoint of the first line.

    You will notice that when you move the cursor close to the startpoint of the first line, the line cursor turns yellow in color and an orange-colored box is displayed below the line cursor. Also, the length of the line shows a value of 40.

21. Press the left mouse button when the line cursor turns yellow in color. Right-click to display the shortcut menu and choose the **Select** option to exit the **Line** tool.

    This completes the sketch of the outer loop. You will notice a the sketch of a very small size is displayed since the display area shows the area in all the four quadrants. Therefore, you need to modify the drawing display area such that the sketch fits the screen. This is done using the **Zoom to Fit** tool.

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



The outer loop of the sketch is completed and is shown in Figure 1-48.

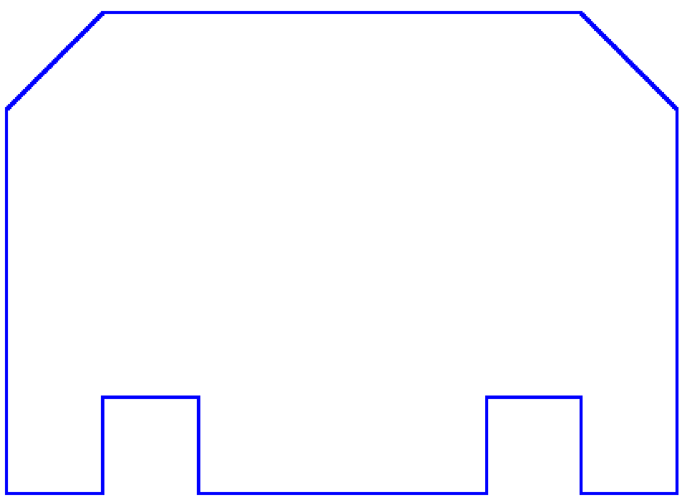
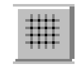


Figure 1-48 *Outer loop of the sketch*

Drawing Circles

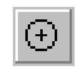
The circles will be drawn using the **Circle** tool. You will use the inferencing line generating from the endpoints of the inclined lines to specify the centerpoint of the circles. The point where the inferencing lines meet is the centerpoint of the circle. As mentioned earlier, the cursor in the sketching environment jumps through a distance of 10mm by default. Therefore, when you move the cursor to define the radius of the arc, the minimum value that can be set is 10mm. This is the reason you need to modify the document settings so that the cursor jumps through a distance of 5mm.



1. Choose the **Grid** button from the **Sketch** toolbar. The **Document Properties - Grid/Snap** dialog box will be displayed.

2. Set the value of the **Major grid spacing** spinner to **50** and choose **OK**.

   Setting this value to 50 will force the cursor to jump through a distance of 5mm instead of 10mm. Therefore, when you move the cursor now, the values shown on the cursor and the coordinates of the points shown close to the lower right corner of the SolidWorks window will be in the increment of 5mm.



3. Choose the **Circle** button from the **Sketch Tools** toolbar to invoke the **Circle** tool.

   Since the **Select** tool was active earlier, the cursor earlier was the arrow cursor. But when you invoke the **Circle** tool, the arrow cursor will be replaced by the circle cursor.

4. Move the circle cursor close to the lower endpoint of the right inclined line and then move it toward the left. Remember that you will not press the left mouse button at this moment.

   The inferencing line will be displayed generating from the lower endpoint of the right inclined line. When you move the cursor toward the left, you will notice that at the point where the cursor is vertically in line with the upper endpoint of the right inclined line, another inferencing line will be generated from the upper endpoint of the right inclined line. This inferencing line will intersect the inferencing line generated from the lower endpoint of the inclined line.

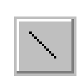
5. Press the left mouse button at the point where the inferencing lines from both the endpoints of the inclined lines meet and then move the circle cursor toward the left to define a circle.

6. Press the left mouse button when the radius of the circle displayed above the circle cursor shows a value of 5.

7. Similarly, draw the circle on the left using the inferencing lines generating from the endpoints of the left inclined line. The sketch after drawing the two circles inside the outer loop is shown in Figure 1-49. Exit the **Circle** tool.

Drawing the Sketch of the Inner Cavity

Next, you will draw the sketch of the inner cavity. To draw the sketch of the inner cavity, you will start drawing with the lower horizontal line.



1. Choose the **Line** button from the **Sketch Tools** toolbar.

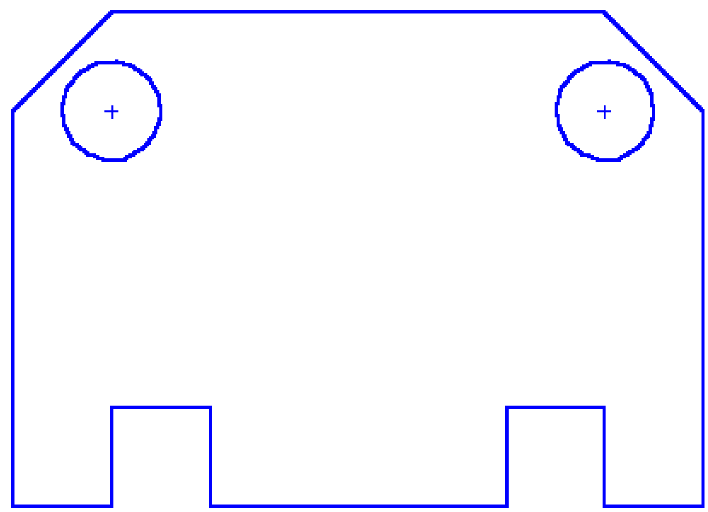


Figure 1-49 *Sketch after drawing the two inner circles*

The arrow cursor will be replaced by the line cursor.

2. Move the line cursor to a location whose coordinates are 30mm 25mm 0mm.

3. Press the left mouse button at this point and move the cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of 30.

4. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 10.

5. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 10.

6. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

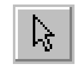
7. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 10.

8. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

9. Move the line horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 10.

10. Move the line cursor vertically downward to the startpoint of the first line. Press the left mouse button when the line cursor turns yellow in color. The length of the line at this point will show a value of 10.

11. Choose the **Select** button from the **Sketch** toolbar. This completes the sketch for Tutorial 1. The final completed sketch for Tutorial 1 is shown in Figure 1-50.



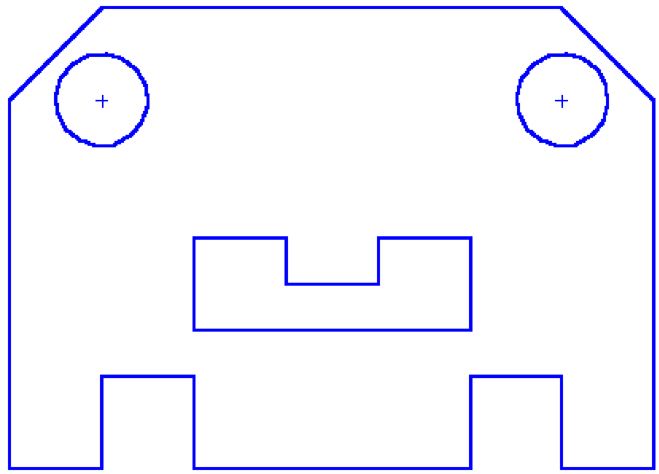
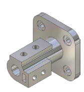


Figure 1-50 *Final sketch for Tutorial 1*



Note

*By default, the entity points are displayed at the endpoints when you sketch an entity. The entity points are dots at the end points of lines, arcs, splines etc. If you do not want the entity points to be displayed then choose **Tools > Options** from the menu bar to invoke the **System Options - General** dialog box. Select the **Sketch** option from the left of this dialog box to display the option related to sketch. Select the **Display entity points in part/assembly sketches** check box and choose the **OK** button from this dialog box.*

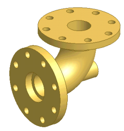
Saving the Sketch

It is recommended that you create a separate directory for saving the tutorial files of this book. When you invoke the option to save a document, the default directory that is displayed is */My Documents*. You will create a directory with the name **SolidWorks** in this directory and then create the directories of each chapter inside the *SolidWorks* directory. As a result, you can save the tutorials of a chapter in the directory of that chapter.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create *SolidWorks* directory inside the */My Documents* directory and then create *c01* directory inside the *SolidWorks* directory.

2. Enter the name of the document as *c01-tut01.sldprt* in the **File name** edit box and choose the **Save** button. The file will be saved in the */My Documents/SolidWorks/c01* directory.

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



Tip. *If you open a document that was saved in the sketching environment, it will be opened in the sketching environment only and not in the part modeling environment.*

Tutorial 2

In this tutorial you will draw the basic sketch for the revolved solid model shown in Figure 1-51. The sketch for the revolved solid model is shown in Figure 1-52. Do not dimension the sketch as the solid model and the dimensions are given only for your reference.

(Expected time: 30 min)

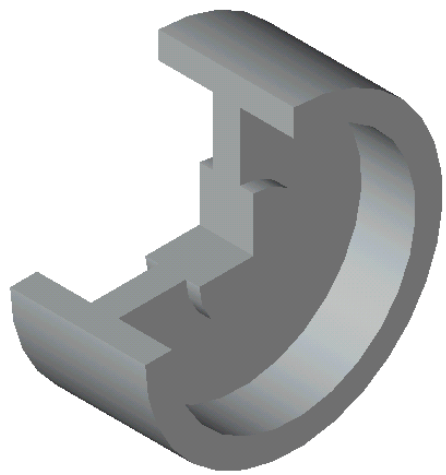


Figure 1-51 *Revolved model for Tutorial 2*

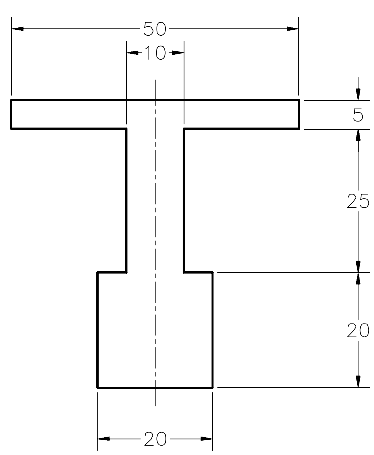
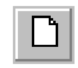


Figure 1-52 *Sketch for the revolved model*

The steps that will be followed to complete this tutorial are listed below:

a. Open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5mm instead of 10mm.
d. Draw the sketch of the model using the **Line** tool, refer to Figure 1-53.
e. Save the sketch and then close the file.

Opening a New File



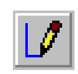
1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.

2. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box.

   A new SolidWorks part document will be opened. But the part document window will not be maximized in the SolidWorks window.

3. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.

   As mentioned earlier, when you open a new part document, the part modeling environment is active by default. However, since you first need to draw the sketch of the revolved model, you need to invoke the sketching environment.

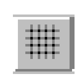


4. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.

   The origin will turn red in color and the **Sketch Tools** toolbar will be displayed below the **Sketch** toolbar. Also, the confirmation corner will be displayed with the **Exit Sketch** and the **Delete Sketch** options on the upper right corner of the drawing area. This suggests that the sketching environment is activated.

Modifying the Snap and Grid Settings and the Dimensioning Units

Before you proceed with drawing the sketch, you need to modify the grid and snap settings so that you can make the cursor jump through a distance of 5mm instead of 10mm, which is the default value.



1. Choose the **Grid** button from the **Sketch** toolbar. The **Document Properties - Grid/Snap** dialog box is displayed.

2. Set the value of the **Major grid spacing** spinner to **50**. Make sure the value of the **Minor-lines per major** spinner is **10**.

   The coordinates displayed close to the lower left corner of the SolidWorks window will show an increment of 5mm instead of the default increment of 10mm when you exit the dialog box.

3. Make sure the **Snap to point** check box in the **Snap** area is selected.

   If you selected units other than millimeter to measure the length while installing SolidWorks, you need to select the units for the current drawing.

4. Select the **Units** option from the area on the left of the **Document Properties - Grid/ Snap** dialog box.

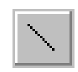
5. Select **Millimeter** from the drop-down list available in the **Linear units** area and **Degrees** from the drop-down list available in the **Angular units** area.

6. Choose the **OK** button after making the necessary settings.

Drawing the Sketch

As evident from Figure 1-52, the sketch will be drawn using the **Line** tool. You will start drawing the sketch from the lower left corner of the sketch.

1. Choose the **Line** button from the **Sketch Tools** toolbar. The arrow cursor will be replaced by the line cursor.



2. Move the line cursor to a location whose coordinates are 40mm 0mm 0mm.

   An inferencing line originating from the origin will be displayed.

3. Press the left mouse button down at this point and move the cursor horizontally toward the right. Press the left mouse button again when the length of the line above the line cursor shows a value of 20.

4. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 20.

5. Move the cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 5.

6. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 25.

7. 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 a value of 20.

8. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

9. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 50.

10. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

11. 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 a value of 20.

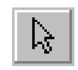
12. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of 25.

13. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 5.

14. Move the line cursor vertically downward to the startpoint of the first line. Press the left mouse button when the line cursor turns yellow in color.

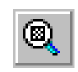
    The length of the line at this point will be 20mm.

15. Choose the **Select** button from the **Sketch** toolbar.



    The sketch is completed but does not fit the screen. Therefore, you need to modify the display area such that the sketch fits the screen.

16. Choose the **Zoom to Fit** button from the **View** toolbar to fit the sketch on the screen. The completed sketch for Tutorial 2 is shown in Figure 1-53.



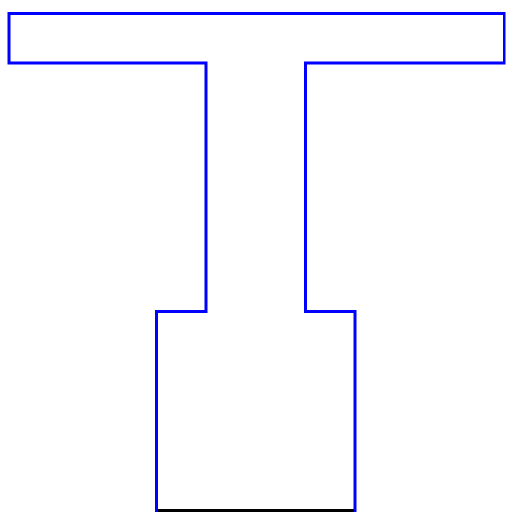
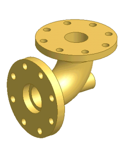
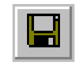


Figure 1-53 *Final sketch for Tutorial 2*



Tip. *You will notice that the bottom horizontal line in the sketch is black in color and the remaining lines are blue in color. In the next chapter, you will learn about the reason why some entities in the sketch are different in color.*

Saving the Sketch



1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter the name of the document as *c01-tut02.sldprt* in the **File name** edit box and choose the **Save** button.

   The document will be saved in the */My Documents/SolidWorks/c01* directory.

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

Tutorial 3

In this tutorial you will draw the basic sketch of the model shown in Figure 1-54. The sketch to be drawn is shown in Figure 1-55. Do not dimension the sketch as the solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

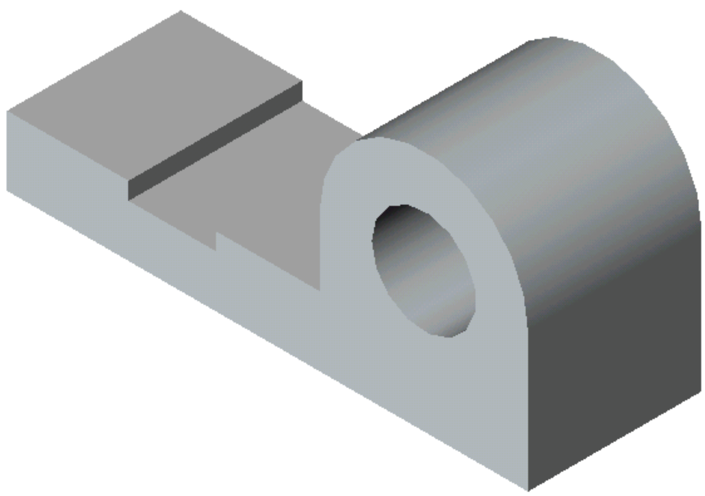


Figure 1-54 *Solid model for Tutorial 3*

The steps that will be followed to complete this tutorial are listed below:

a. Start SolidWorks and then open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5mm instead of 10mm.
d. Draw the outer loop of the sketch using the **Line** and the **Tangent Arc** tool, refer to Figure 1-56.
e. Draw the inner circle using the **Circle** tool, refer Figure 1-57
f. Save the sketch and then close the file.

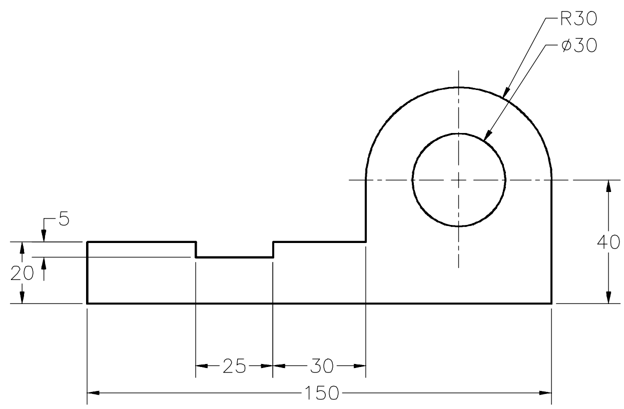
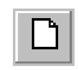


Figure 1-55 *Sketch for Tutorial 3*

Opening a New File



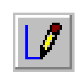
1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.

2. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box.

   A new SolidWorks part document will be opened. However, as mentioned earlier, the part document window will not be maximized in the SolidWorks window.

3. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.

   Since you first need to draw the sketch of the revolved model, you need to invoke the sketching environment.

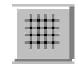


4. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.

   The origin turns red in color and the **Sketch Tools** toolbar is displayed below the **Sketch** toolbar. Also, the confirmation corner with the **Exit Sketch** and **Delete Sketch** options will be displayed on the upper right corner of the drawing area. This suggests that the sketching environment is activated.

Modifying the Snap and Grid Settings and Dimensioning Units

Since the dimensions in the sketch are in the multiples of 5mm, you need to modify the grid and snap settings so that you can make the cursor jump through a distance of 5mm instead of 10mm.



1. Choose the **Grid** button from the **Sketch** toolbar. The **Document Properties - Grid/Snap** dialog box will be displayed.

2. Set the value of the **Major grid spacing** spinner to **50** and make sure that the value of the **Minor-lines per major** is **10**. Also make sure that the **Snap to points** check box in the **Snap** area is selected.

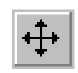
   The coordinates displayed close to the lower left corner of the SolidWorks window will show an increment of 5mm instead of the default increment of 10mm when you close the dialog box.

3. If you selected units other than millimeter to measure the length while installing SolidWorks, select the **Units** option from the area on the left of the **Document Properties - Grid/Snap** dialog box. Select **Millimeter** from the drop-down list available in the **Linear units** area and **Degrees** from the drop-down list available in the **Angular units** area.

4. Choose **OK** to close the dialog box.

Drawing the Outer Loop

As evident from Figure 1-55, the sketch will be drawn using the **Line**, **Tangent Arc**, and the **Circle** tools. You will start drawing from the lower left corner of the sketch. Since the length of the lower horizontal line is 150mm, therefore, you will have to modify the drawing display area such that the drawing area in the first quadrant is increased. This can be done using the **Pan** tool.

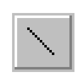


1. Choose the **Pan** button from the **View** toolbar. The arrow cursor will be replaced by the pan cursor.

2. Press and hold the left mouse button down and drag the cursor toward the bottom left corner of the screen.

   You will notice that the origin also moves toward the bottom left corner of the screen, thus increasing the drawing area in the first quadrant.

3. After dragging the origin close to the lower left corner, release the left mouse button.



4. Choose the **Line** button from the **Sketch Tools** toolbar.

   The pan cursor will be replaced by the line cursor.

5. Move the line cursor to a location whose coordinates are 40mm 0mm 0mm.

   An inferencing line originating from the origin will be displayed.

6. Press the left mouse button down at this point and move the cursor horizontally toward the right. Press the left mouse button again when the length of the line above the line cursor shows a value of 150.

7. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 40.

   The next entity that has to be drawn is a tangent arc. As mentioned earlier, you can draw the tangent arc using the **Line** tool also. Drawing arcs from within the **Line** tool is a recommended method when you need to draw a sketch that is a combination of lines and arcs. This increases the productivity by reducing the time taken in invoking the tools for drawing an arc and then invoking the **Line** tool to draw lines.

8. Move the line cursor away from the endpoint of the last line and then move it back to the endpoint.

   The arc mode will be invoked and the line cursor will be replaced by the arc cursor. Also, the **Arc PropertyManager** will be displayed in place of the **Line PropertyManager**.

9. Move the arc cursor vertically upward to a small distance. A dotted reference line will be displayed.

10. When the dotted line is displayed, move the cursor toward the left.

    You will notice that a tangent arc is being drawn. The angle of the tangent arc and its radius will be displayed above the arc cursor.

11. Press the left mouse button when the angle value above the arc cursor shows a value of 180 and the radius shows a value of 30 to complete the arc.

    The required tangent arc is drawn. As mentioned earlier, the line mode is automatically invoked after you have drawn the arc using the **Line** tool. Therefore, the arc cursor will be replaced by the line cursor and the **Arc PropertyManager** will be replaced by the **Line PropertyManager**.

12. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of 20.

13. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 30.

14. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

15. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 25.

16. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of 5.

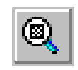
17. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of 35.

18. Move the line cursor to the startpoint of the first line. Press the left mouse button when the line cursor turns yellow in color.

    The length of the line at this point will be 20mm.

19. Right-click and choose **Select** from the shortcut menu to exit the **Line** tool.

20. Choose the **Zoom to Fit** button to fit the sketch on the screen. This completes the outer loop of the sketch. The sketch after drawing the outer loop is shown in Figure 1-56.



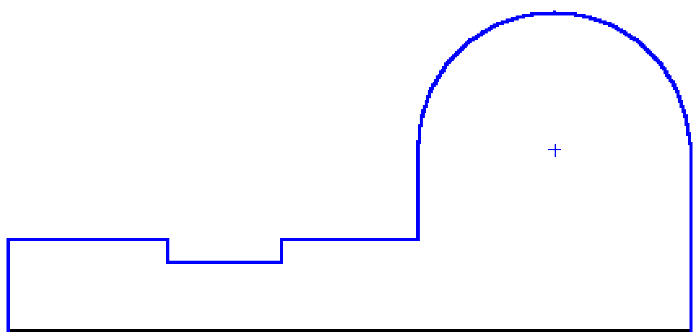
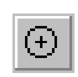


Figure 1-56 *Sketch after drawing the outer loop*

Drawing the Circle

The circle in the sketch will be drawn using the **Circle** tool. The centerpoint of the circle will be the centerpoint of the arc, which is displayed by a cross. This cross is automatically drawn when you draw the arc. You can select this centerpoint to draw the circle.

1. Choose the **Circle** button from the **Sketch Tools** toolbar to invoke the **Circle** tool.



   The arrow cursor will be replaced by the circle cursor.

2. Move the circle cursor close to the centerpoint of the arc and when the pencil in the circle cursor turns yellow in color, press the left mouse button.

3. Move the cursor toward the left and when the radius of the circle above the circle cursor shows a value of 15, press the left mouse button.

A circle of 15mm radius will be drawn.

4. This completes the sketch for Tutorial 3. Right-click and choose the **Select** option from the shortcut menu to exit the **Circle** tool.

   The final sketch for Tutorial 3 is shown in Figure 1-57.

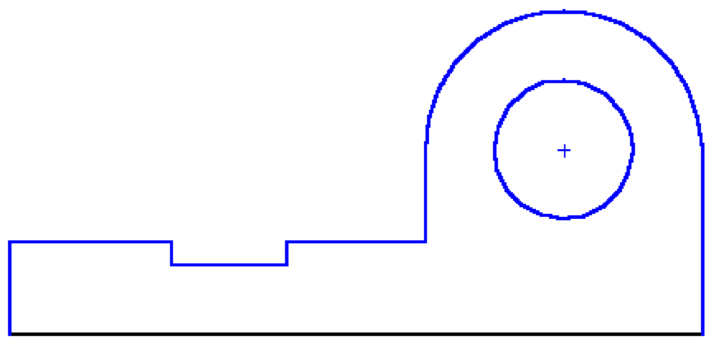


Figure 1-57 *Final sketch for Tutorial 3*

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter the name of the document as *c01-tut03.sldprt* in the **File name** edit box and choose the **Save** button.

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

Tutorial 4

In this tutorial you will draw the sketch of the model shown in Figure 1-58. The sketch for the model is shown in Figure 1-59. Do not dimension the sketch as the dimensions and the solid model are given only for your reference. **(Expected time: 30 min)**

The steps that will be followed to complete this tutorial are listed below:

a. Start SolidWorks and then open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Modify the settings of the snap and grid so that the cursor jumps through a distance of 5mm instead of 10mm.
d. Draw the outer loop of the sketch using the **Line** and the **Tangent Arc** tool, refer to Figure 1-60.
e. Save the sketch and then close the file.

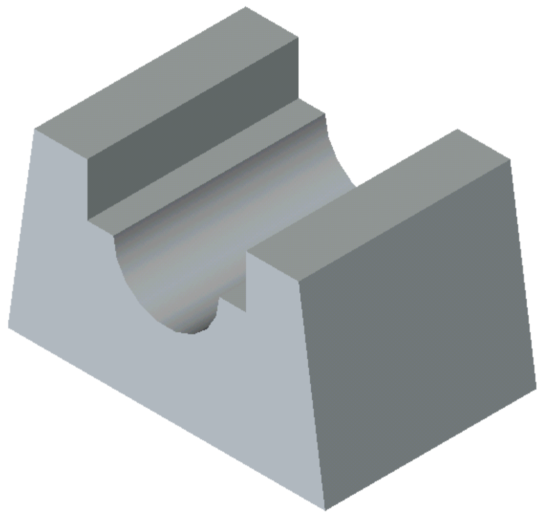


Figure 1-58 Model for Tutorial 4

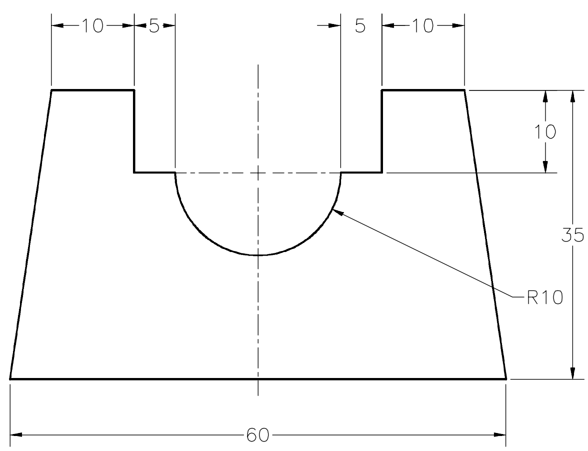
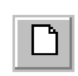


Figure 1-59 Sketch for Tutorial 4

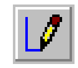
Opening a New File



1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.
2. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box.
3. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.

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

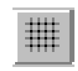
4. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.



Modifying the Snap and Grid Settings and Dimensioning Units

As evident in Figure 1-59, the dimensions in the sketch are in the multiples of 5mm. Therefore, you need to modify the grid and snap settings so that the cursor jumps through a distance of 5mm instead of 10mm.

1. Choose the **Grid** button from the **Sketch** toolbar. The **Document Properties - Grid/Snap** dialog box will be displayed.



2. Set the value of the **Major grid spacing** spinner to **50** and the **Minor line per major** spinner to **10**. Also, select the **Snap to points** check box if not selected.

   You will notice that the coordinates displayed close to the lower left corner of the SolidWorks window shows an increment of 5mm instead of the default increment of 10mm.

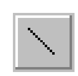
3. If you selected units other than millimeter to measure the length while installing SolidWorks, select the **Units** option from the area on the left of the **Document Properties - Grid/Snap** dialog box. Select **Millimeter** from the drop-down list available in the **Linear units** area and **Degrees** from the drop-down list available in the **Angular units** area.

4. Choose **OK** to close the dialog box.

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 will start drawing from the lower left corner of the sketch.

1. Choose the **Line** button from the **Sketch Tools** toolbar.



   The arrow cursor will be replaced by the line cursor.

2. Move the line cursor to a location whose coordinates are 30mm 0mm 0mm.

   An inferencing line originating from the origin will be displayed.

3. Press the left mouse button at this point and move the cursor horizontally toward the right. Press the left mouse button again when the length of the line above the line cursor shows a value of 60.

   The bottom horizontal line of 60mm length will be drawn.

4. Choose the **Zoom to Fit** button from the **View** toolbar to increase the display of the line that is drawn.

   As mentioned earlier, you can invoke the drawing display tools while you are inside a tool also. After modifying the drawing display, the tool that was active before invoking the drawing display tool will be restored and you can continue using that tool. Therefore, after the drawing display area is modified, the **Line** tool will be restored and you can continue drawing lines.

5. Move the line cursor along a direction that makes an angle close to 98-degree with the positive X axis direction. The angle can be checked from the spinner above the **Delta X** spinner in the **Line PropertyManager**.

6. When the **Delta X** spinner shows a value of **5** and **Delta Y** spinner shows a value of **35**, press the left mouse button.

   The length of the line at this point will be 35.36.

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 a value of 10.

8. Move the line cursor vertically downward and press the left mouse button when the length of the line above the line cursor shows a value of 10.

9 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 a value of 5.

   Next, you need to draw the arc that is normal to the last line.

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 will be invoked and the line cursor will be replaced by the line cursor. Also, the **Line PropertyManager** will be replaced by the **Arc PropertyManager**.

11. Move the arc cursor vertically downward through a small distance.

    A dotted line will be displayed.

12. When the dotted line is displayed, move the arc cursor toward the left.

    You will notice that a normal arc is being drawn.

13. Press the left mouse button when the angle value on the arc cursor is 180 and the radius value is 10.

    An arc normal to the last line will be drawn and the line mode will be activated.

14. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor shows a value of 5.

15. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor shows a value of 10.

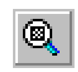
16. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor shows a value of 10.

17. Move the line cursor to the startpoint of the first line and when the line cursor turn yellow in color, press the left mouse button.

18. Right-click to display the shortcut menu and choose **Select** 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. Choose the **Zoom to Fit** button from the **View** toolbar to modify the drawing display area. The final sketch for Tutorial 4 is shown in Figure 1-60.



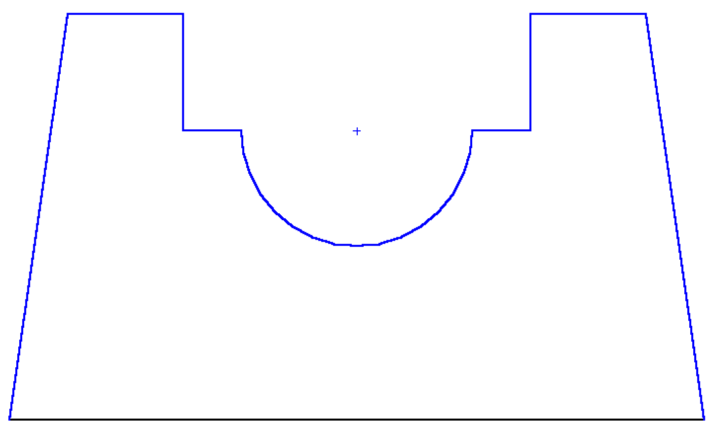


Figure 1-60 *Final sketch for Tutorial 4*

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter the name of the document as *c01-tut04.sldprt* in the **File name** edit box and choose the **Save** button.

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

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with the answers given at the end of this chapter.

1. The base feature of any design is a sketched feature and is created by drawing the sketch. (T/F)

2. You can also invoke the **3Pt Arc** tool from within the **Line** tool. (T/F)

3. By default, the cursor jumps through a distance of 5mm. (T/F)

4. When you save a file in the sketching environment, it is opened in the part modeling environment when you open it for the next time. (T/F)

5. You can convert a sketched entity into a construction entity by selecting the __________ check box provided in the **PropertyManager**.

6. To draw a rectangle at an angle, you need to use the __________ tool.

7. The __________ are the temporary lines that are used to track a particular point on the screen.

8. You can also invoke the __________ tool or exit a sketching tool by pressing the ESC key.

9. When you select a tangent entity to draw a tangent arc, the __________ relation is applied between the startpoint of the arc and the tangent entity.

10. Irrespective of the number of segments in a spline, it will be considered as a __________ entity.

REVIEW QUESTIONS

Answer the following questions:

1. The 3 point arcs are the ones that are drawn by defining the startpoint of the arc, the endpoint of the arc, and a point on the arc. (T/F)

2. You can also delete the sketched entities by selecting them and choosing the **Delete** option from the shortcut menu that is displayed upon right-clicking. (T/F)

3. The origin is a blue-colored icon that is displayed in the middle of the sketcher screen. (T/F)

4. In SolidWorks, the circles are drawn by specifying the centerpoint of the circle and then pressing and entering the radius of the circle in the dialog box that is displayed. (T/F)

5. When you open a new SolidWorks document, it is not maximized in the SolidWorks window. (T/F)

6. In SolidWorks, a rectangle is considered as a combination of which of the following entities.

   (a) Lines (b) Arcs
   (c) Splines (d) None

7. Which one of the following options is not displayed in the **New SolidWorks Document** dialog box?

   (a) **Part** (b) **Assembly**
   (c) **Drawing** (d) **Sketch**

8. Which one of the following entities will not be considered while converting a sketch into a feature?

   (a) Sketched Circles (b) Sketched Lines
   (c) Construction Lines (d) None

9. When you select a line of the rectangle, which one of the following **PropertyManager** will be displayed?

   (a) **Line PropertyManager** (b) **Line/Rectangle PropertyManager**
   (c) **Rectangle PropertyManager** (d) None

10. While drawing an elliptical arc, which one of the following **PropertyManager** will be displayed?

   (a) **Arc PropertyManager** (b) **Ellipse PropertyManager**
   (c) **Elliptical Arc PropertyManager** (d) None

EXERCISES

Exercise 1

Draw the sketch of the model shown in Figure 1-61. The sketch to be drawn is shown in Figure 1-62. Do not dimension the sketch as the solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

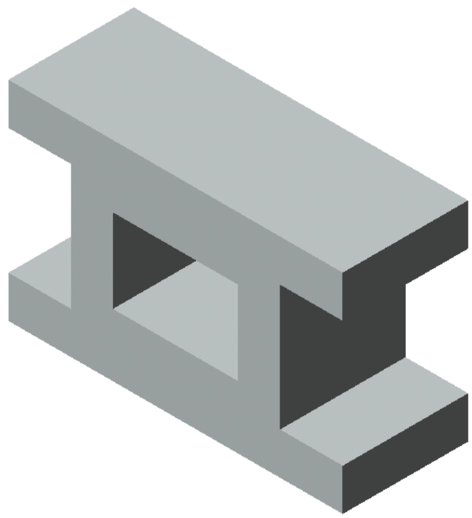


Figure 1-61 *Solid model for Exercise 1*

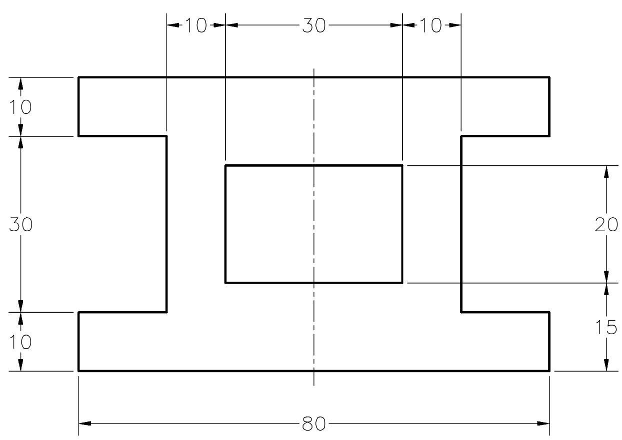


Figure 1-62 *Sketch for Exercise 1*

Exercise 2

Draw the sketch of the model shown in Figure 1-63. The sketch to be drawn is shown in Figure 1-64. Do not dimension the sketch as the solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

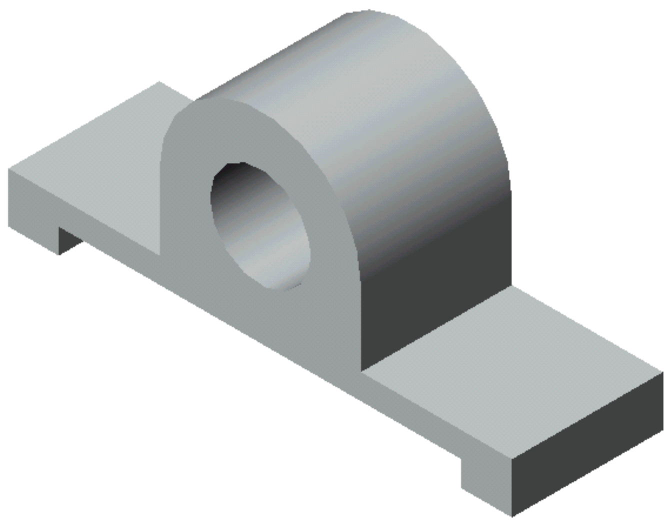


Figure 1-63 *Solid model for Exercise 2*

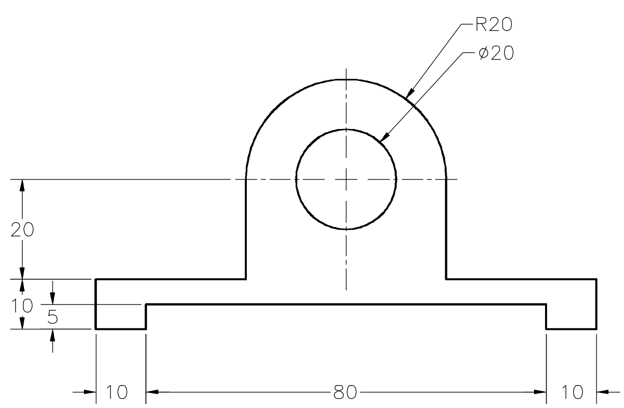


Figure 1-64 *Sketch for Exercise 2*

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** F, **4.** F, **5. For Construction**, **6. Parallelogram**, **7.** inferencing lines, **8. Select**, **9. Tangent**, **10.** single