



Chapter 1

Drawing Sketches for Solid Models

Learning Objectives

After completing this chapter you will be able to:

- *Understand the need for the sketching environment.*
- *Understand the base reference planes that can be selected to create sketches.*
- *Understand various drawing display tools.*
- *Understand various sketching tools.*
- *Use various selection methods.*
- *Delete sketched entities.*

THE SKETCHING ENVIRONMENT

Most of the designs created in a solid modeling tool consist of the profile-based features, placed features, and reference features. A profile is a combination of a number of two-dimensional (2D) entities such as lines, arcs, circles, and so on. The profile-based features are the features that are created using these entities. A profile-based feature is the base feature or the first feature in most designs. For example, refer to the solid model shown in Figure 1-1.

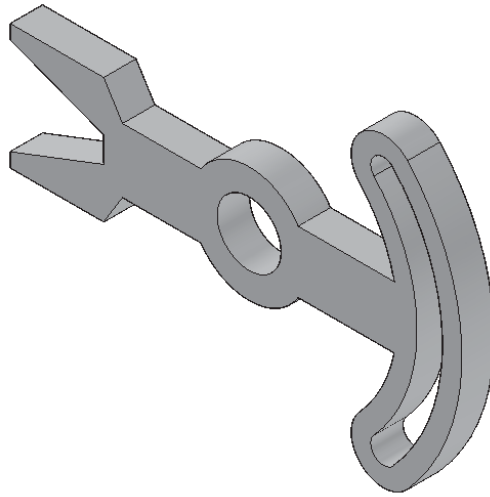


Figure 1-1 Solid model

This model is created using the profile shown in Figure 1-2.

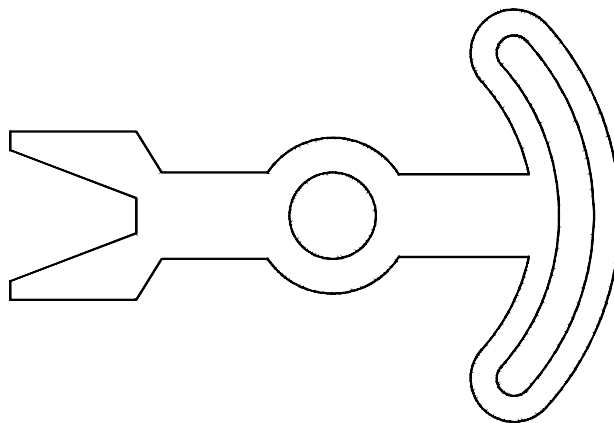


Figure 1-2 Profile of the solid model shown in Figure 1-1

In most of the designs, you first need to invoke the sketching environment and then create the profile of the model in it. After creating the profile, exit the sketching environment and then use the solid modeling tools to complete the design. You can invoke the sketching environment in the **Part** environment of Solid Edge.

There are two methods of starting a new document in the **Part** environment: starting Solid Edge in the **Part** environment and starting a new part document using the **New** dialog box. These methods are discussed next.

Starting Solid Edge in the Part Environment

In most of the solid modeling programs, you need to start the program and then select the option to start a new part document. However, Solid Edge can be directly started in the **Part** environment with a default part document. This can be done using the taskbar menu. Choose the **Start** button available on the lower left corner of the screen to invoke the menu and then choose **All Programs > Solid Edge V16 > Part**, as shown in Figure 1-3.

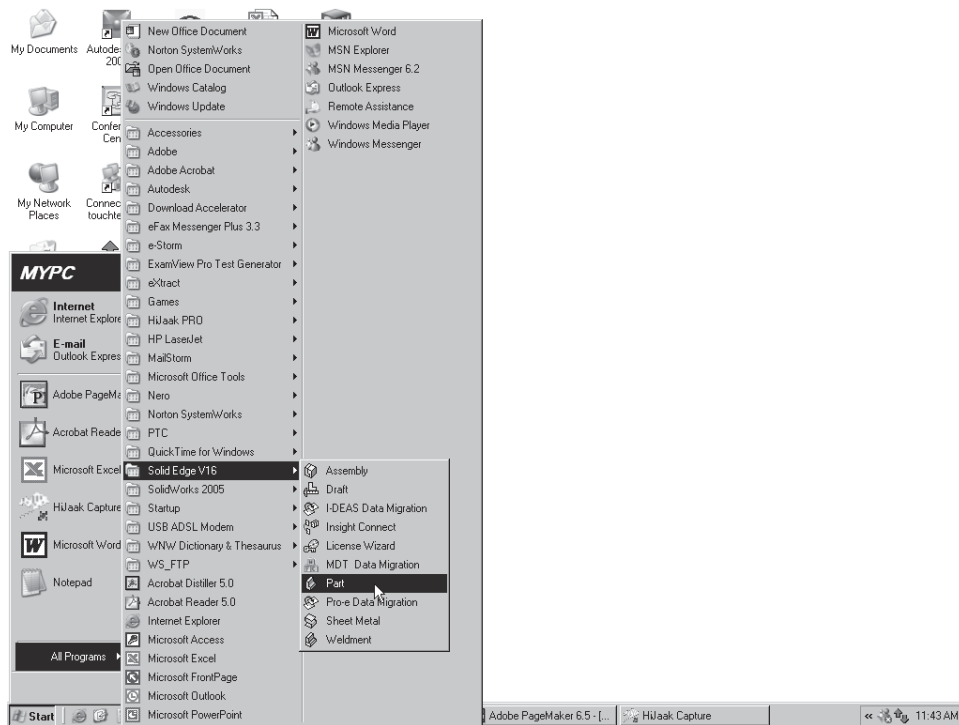


Figure 1-3 Starting Solid Edge in the **Part** environment

The system will prepare to start Solid Edge in the **Part** environment. Once all files are loaded, the Solid Edge window will be displayed and a default part document with the name **Part1** will be opened.

Note that whenever you start Solid Edge in the **Part** environment or open a new part document, the **Protrusion** tool will be active and you will be prompted to select a planar face or

a reference plane to create profile for the protrusion feature. But all designs may not necessarily have protrusion as the first feature. Therefore, it is recommended that whenever you start a new part file, you should first exit the **Protrusion** tool by choosing the **Select Tool** button from the **Features** toolbar. You can also press the ESC key to exit the tool.



Note

You will learn more about protrusion in later chapters.

Starting a New Part Document Using the New Dialog Box

You can also start a new part document using the **New** dialog box. Choose the **New** button from the **Main** toolbar to display the **New** dialog box, as shown in Figure 1-4. The options available in this dialog box are discussed next.

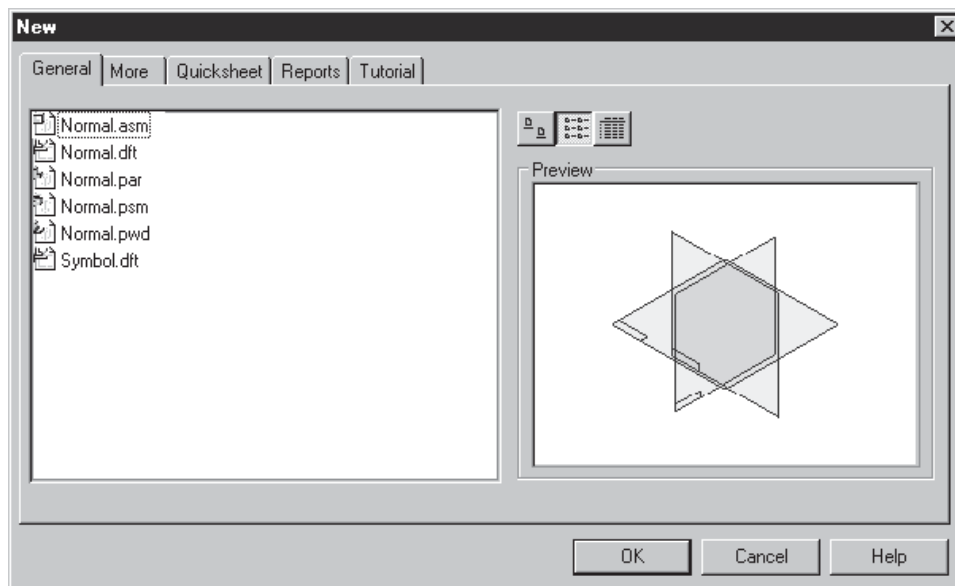


Figure 1-4 The New dialog box

General Tab

The **General** tab provides the default templates for starting the **Assembly** environment (**Normal.asm**), **Draft** environment (**Normal.dft** and **Symbol.dft**), **Part** environment (**Normal.par**), **Sheet Metal** environment (**Normal.psm**), and **Weldment** environment (**Normal.pwd**).

Double-click on **Normal.par** to open a new document in the **Part** environment of Solid Edge.



Note

*It is assumed that you have installed Solid Edge in **Metric** units. Therefore, you can use **Normal.par** from the **General** tab to open a new document in the **Part** environment.*

More Tab

The **More** tab provides the Metric and English templates for starting files in various environments of Solid Edge. The Metric templates are named as **Normmet.*** and the English templates are named as **Normeng.***.



Tip. The difference between the Metric and English templates is that in the Metric templates, the length is measured in millimeter (mm) and the mass is measured in kilogram (kg), whereas in the English templates, the length is measured in inches (in) and the mass is measured in pounds (lbm).

Quicksheet Tab

The **Quicksheet** tab provides the drawing template with empty (blank) drawing views of a part or an assembly. You can simply drag and drop any part or assembly document to populate the drawing views.

Reports Tab

The **Reports** tab provides the template for generating reports for the Solid Edge assemblies. You will learn more about reports in later chapters.

Tutorial Tab

The **Tutorial** tab provides the Metric template for starting various environments of Solid Edge.

Large Icon Button

The **Large Icon** button is used to display the templates in various tabs of the **New** dialog box in the form of large icons.

List Button

The **List** button is used to display the templates in various tabs of the **New** dialog box in the form of a list.

Detail Button

The **Detail** button is used to list the details of the templates in various tabs of the **New** dialog box. When you choose this button, the area on the left will be divided into four columns. The first column lists the names of the templates, the second column lists the sizes, the third column lists the types of the template files, and the last column lists the dates when the templates were last modified.

Preview Area

The **Preview** area shows the preview of the selected template.

A new Solid Edge document in the **Part** environment is shown in Figure 1-5. This figure also shows various components of the part document of Solid Edge.

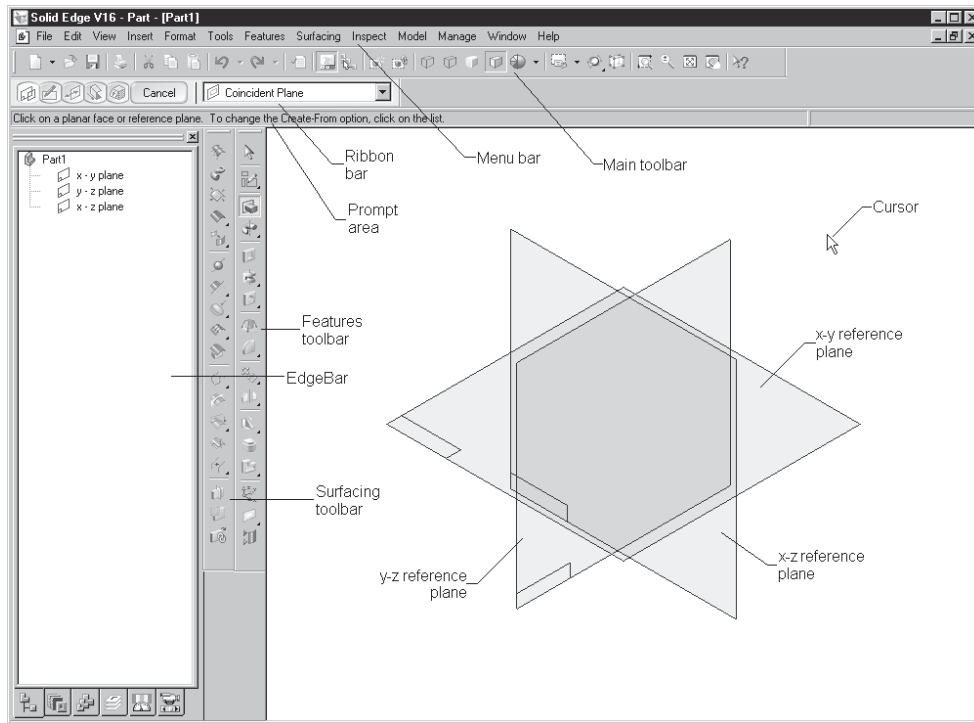


Figure 1-5 New document in the **Part** environment

INVOKING THE SKETCHING ENVIRONMENT

As mentioned earlier, whenever you start a new document in the **Part** environment of Solid Edge, the **Protrusion** tool will be active, and you will be prompted to select a planar face or a reference plane. If you select any of the three reference planes shown in Figure 1-5, it will be oriented parallel to the screen and the sketching environment will be invoked to draw the profile for the protrusion feature.

You can exit the **Protrusion** tool by pressing the ESC key or by choosing the **Select Tool** button from the **Features** toolbar. You can choose the **Sketch** button to draw an independent sketch, which can be used as a profile to create a single or multiple feature. When you choose this button, you will be prompted to select a planar face or a reference plane. As soon as you select a reference plane, it will be oriented parallel to the screen and the sketching environment will be invoked. Figure 1-6 shows the default screen appearance in the sketching environment of Solid Edge.



Tip. If the toolbar icons appear large, you can make them small. To do that, right-click on any toolbar to display the shortcut menu and then choose **Toolbars** to display the **Toolbars** dialog box. Clear the **Large buttons** check box and select **OK**. The toolbar icons will appear smaller in size now.

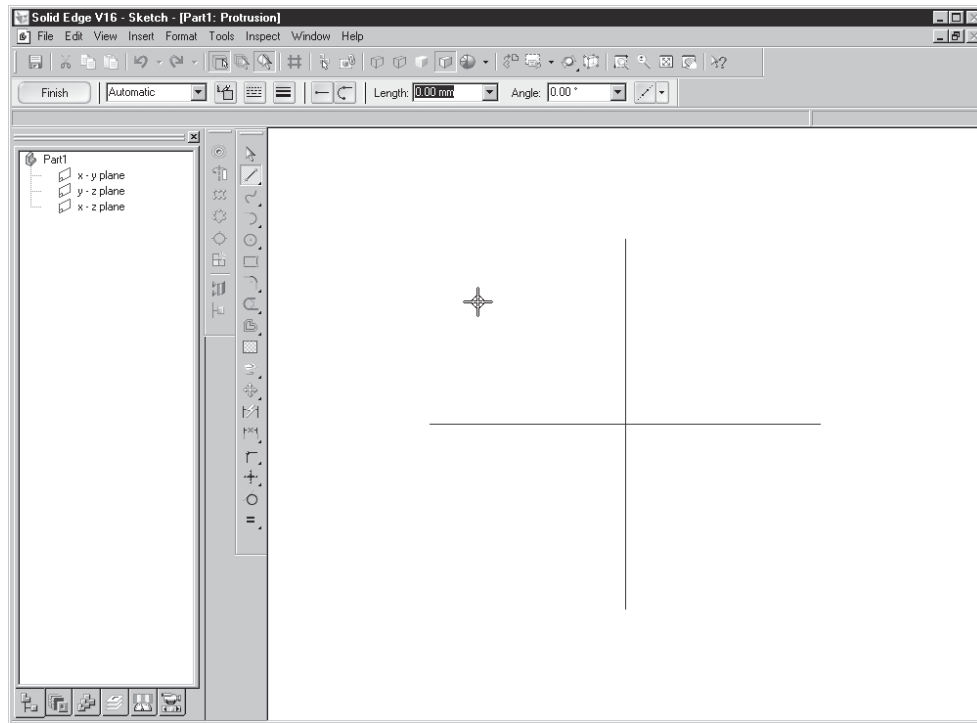


Figure 1-6 Screen appearance in the sketching environment of Solid Edge

THE DRAWING DISPLAY TOOLS

The drawing display tools are an integral part of any solid modeling tool. They enable you to zoom and pan the drawing so that you can view it clearly. The drawing display tools available in Solid Edge are discussed next.

Zooming to an Area

Menu: View > Zoom Area

Toolbar: Main > Zoom Area



The **Zoom Area** tool allows you to zoom on to a particular area by defining a box around it. When you choose this button, a plus sign of infinite length will be attached to the tip of the cursor and you will be prompted to click for the first corner or drag for the box. Specify a point on the screen to define the first corner of the zoom area. Next, move the cursor and specify another point to define the opposite corner of the zoom area. The drawing window defined inside the box will be zoomed and displayed on the screen.

Dynamic Zooming

Menu: View > Zoom
Toolbar: Main > Zoom



The **Zoom** tool enables you to dynamically zoom in or out of the drawing. You can also use this tool to increase the display area to double the current size. To zoom in, press and hold the left mouse button in the center of the screen and then drag the cursor down. Similarly, to zoom out, press and hold the left mouse button in the center of the screen and drag the cursor up.

To increase the drawing display area to double the current size, invoke this tool and click anywhere in the drawing window. Note that the drawing display area will be increased such that the point at which you clicked is brought to the center of the screen.

Fitting all Entities in the Current Display

Menu: View > Fit
Toolbar: Main > Fit



The **Fit** tool enables you to modify the drawing display area such that all entities in the drawing fit in the current display.

Panning Drawings

Menu: View > Pan
Toolbar: Main > Pan



The **Pan** tool allows you to dynamically pan drawings in the drawing window. When you invoke this tool, the arrow cursor will be replaced by a hand cursor and you will be prompted to click to select the origin or drag for dynamic pan. Press and hold the left mouse button down in the drawing window and then drag to pan the drawing. You can also pan the drawing by specifying two points in the drawing window. First, specify a point anywhere in the drawing window and then move the cursor. You will notice that a rubber-band line is displayed. One end of this line is fixed at the point you specified and the other end is attached to the hand cursor. Move the cursor and specify another point in the drawing window to pan the drawing.



Tip. You can also use the keyboard to modify the drawing display area. Various combinations of the keys are given below.

CTRL+ Top/Left arrow key = Zoom In
 CTRL+ Bottom/Right arrow key = Zoom Out
 SHIFT+ Left/Bottom arrow key = Rotate Left
 SHIFT+ Right/Top arrow key = Rotate Right
 CTRL + SHIFT+ Top arrow key = Pan Upward
 CTRL + SHIFT+ Left arrow key = Pan Toward Left
 CTRL + SHIFT+ Bottom arrow key = Pan Downward
 CTRL + SHIFT+ Right arrow key = Pan Toward Right

Restoring the Original Orientation of the Sketching Plane

Toolbar: Main > Sketch View



Sometimes, while using the drawing display tools, you may change the orientation of the sketching plane. The **Sketch View** tool enables you to restore the original orientation that was active when you invoked the sketching environment. Note that this tool is available only in the sketching environment.

SKETCHING TOOLS

All tools required to create a profile or a sketch in Solid Edge are available in the **Draw** toolbar and are discussed next.

Drawing Lines

Toolbar: Draw > Line



Lines are the most widely used sketching entities in any design. In Solid Edge, the **Line** tool enables you to draw straight lines and tangent or normal arcs originating from the endpoint of a selected line. When you invoke the **Line** tool, the **Line** ribbon bar will be displayed, as shown in Figure 1-7, and you will be prompted to click for the first point of the line. The methods of creating lines and arcs using this tool are discussed next.



Figure 1-7 Line ribbon bar

Drawing Straight Lines

To draw a straight line, specify a point in the drawing window by pressing the left mouse button. A rubber-band line is displayed with the start point fixed at the point you specified and the second point attached to the cursor. Also, you will be prompted to click for the second point of the line. Note that on moving the cursor in the drawing window, the length and angle of the line are modified accordingly in the **Line** ribbon bar. You can draw a line by specifying its length and angle in the **Line** ribbon bar. You can also specify its endpoint in the drawing window by pressing the left mouse button.

While drawing a line, you will notice that some symbols are displayed on the right of the cursor. For example, after specifying the start point of the line, if you move the cursor in the horizontal direction, a symbol similar to a horizontal line will be displayed. This symbol is called relationship handle and indicates the relationship that is applied to the entity being drawn. In the above-mentioned case of a horizontal line, the horizontal relationship handle is displayed on the right of the cursor. This relationship will ensure that the line you draw is horizontal. These relationships are automatically applied to the profile while drawing.



Note

Relationships are also applied between the sketched entities and the reference planes. You will learn more about relationships in later chapters.

The process of drawing lines does not end after you have defined the first line. You will notice that as soon as you define the endpoint of the first line, another rubber-band line starts. The start point of this line is the endpoint of the last line and the endpoint of the new line is attached to the cursor.

This process of drawing continuous lines continues until you right-click to terminate the continuous line. However, note that even after right-clicking, the **Line** tool will not be terminated and you will still be prompted to specify the first point of the line. You can terminate the **Line** tool by choosing the **Select Tool** button from the **Draw** toolbar or by pressing the ESC key. Figures 1-8 and 1-9 show continuous lines being drawn in Solid Edge.

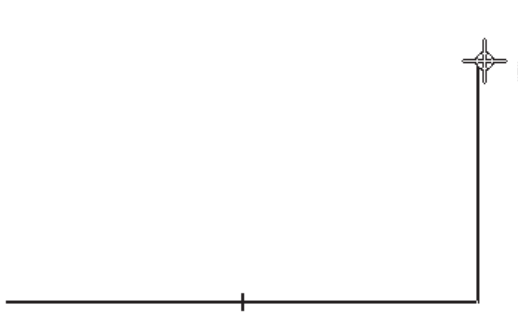


Figure 1-8 Vertical relationship handle displayed while drawing the vertical line

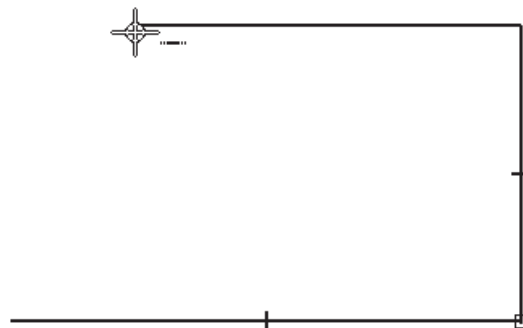


Figure 1-9 Horizontal relationship handle displayed while drawing the horizontal line

While drawing lines, you will notice that if the cursor is horizontally or vertically aligned with the endpoint or midpoint of a line or reference plane, some dashed lines are displayed. These dashed lines are called alignment indicators and are used to indicate the horizontal or vertical alignment of the current location of the cursor with a point. Figure 1-10 shows the alignment indicators originating from the endpoints of the existing lines.



Tip. If the alignment indicator is not displayed, move the cursor over the entity from which you want the alignment indicator to originate. The entity turns red in color and the alignment indicator is displayed.

Drawing Tangent and Normal Arcs

As mentioned earlier, you can also use the **Line** tool to draw a tangent or a normal arc. To switch to the arc mode when the **Line** tool is active, press the A key or choose the **Arc** button from the ribbon bar. You will notice that the **Length** and **Angle** edit boxes in the ribbon bar will be replaced by the **Radius** and **Sweep** edit boxes. These edit boxes can be used to define the radius and the included angle of the resulting arc.

Also, a small circle is displayed at the start point of the arc. This circle is divided into four regions using lines. These regions are called intent zones and are used to define the type of arc that will be created. To create an arc tangent to the line, move the cursor through a small distance in the zone that is tangent to the line; the tangent arc will be drawn. Similarly, if you

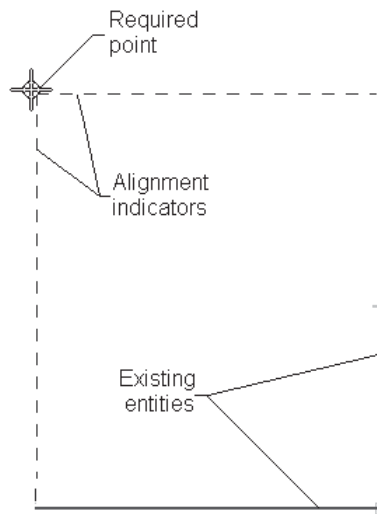


Figure 1-10 Using the alignment indicators to locate a point

move the cursor in the zone that is normal to the line, the normal arc will be drawn. After drawing the required arcs, you can switch back to the line mode by pressing the L key or by choosing the **Line** button from the ribbon bar. Figure 1-11 shows a tangent arc being drawn from within the **Line** tool.

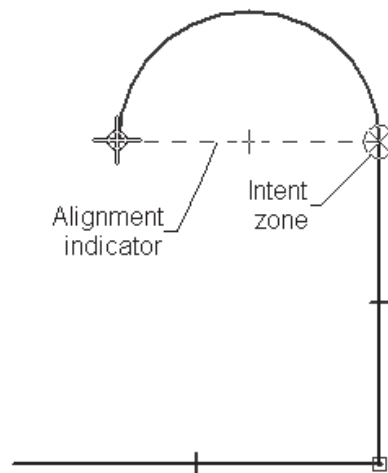


Figure 1-11 Drawing tangent arc from within the **Line** tool

Using the buttons available in the **Line** ribbon bar, you can specify the color of lines, type of lines, and width of lines. You can also draw a projection line of infinite length using the **Projection Line** button available on the right side of the **Line** ribbon bar. The projection lines are generally used in the drafting environment.



Tip. If you have selected an incorrect point as the start point of a line, right-click to cancel it. You will again be prompted to specify the first point of the line.

Drawing Circles

In Solid Edge, you can draw circles using three methods, which are discussed next.

Drawing a Circle by Specifying the Center Point and Radius

Toolbar: Draw > Circle by Center



This is the most widely used method of drawing circles. In this method, you need to specify the center point of a circle and a point on it. The point on the circle defines the radius of the circle. To draw the a circle using this method, choose the **Circle by Center** button from the **Draw** toolbar; the **Circle** ribbon bar will be displayed and you will be prompted to specify the center point of the circle. Specify the center point of the circle in the drawing window. Next, you will be prompted to specify a point on the circle. Specify a point on the circle to define the radius. Alternatively, you can also enter the value of the diameter or radius in the ribbon bar. Figure 1-12 shows a circle drawn using this method.

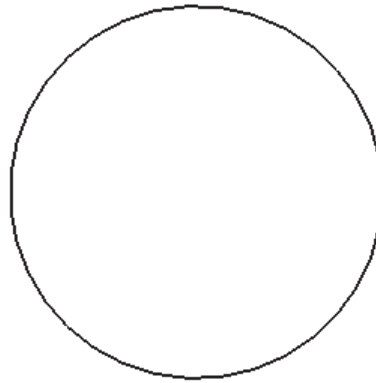


Figure 1-12 Circle drawn using the **Circle by Center** method

Drawing a Circle by Specifying Three Points

Toolbar: Draw > Circle by Center > Circle by 3 Points



This method is used to draw a circle using the three points that you need to define on it. To invoke this method, press and hold the left mouse button down on the **Circle by Center** button in the **Draw** toolbar to display a flyout. From the flyout, choose the **Circle by 3 Points** button. You will be prompted to specify the first and second points of the circle. On specifying these two points, small reference circles will be displayed on these two points, as shown in Figure 1-13. Now, specify the third point, which is a point on the circle. This completes the circle.

Drawing a Tangent Circle

Toolbar: Draw > Circle by Center > Tangent Circle



This method is used to draw a circle that is tangent to one or two existing entities. To invoke this method, choose the **Tangent Circle** button from the **Circle by Center** flyout. You will be prompted to specify the first point on the circle. The circle will be

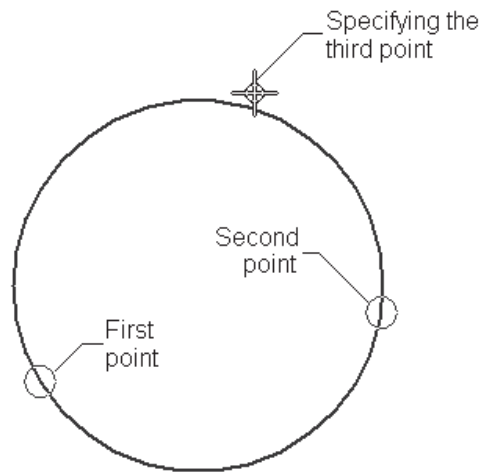


Figure 1-13 Drawing circle using the *Circle by 3 Points* method

drawn using two or three points, depending on how you specify the first point of the circle. If you specify the first point on an entity, you will be prompted to specify the second point and the circle will be drawn using these two points. However, if you do not specify the first point on any existing entity, then you need to define the circle using three points.

When you move the cursor close to an existing entity to specify the second or third point, the tangent relationship handle will be displayed. Now, if you specify the point, the resulting circle will be tangent to the selected entity. Also, small reference circles will be displayed at the points where the circle is tangent to the selected entities. Figure 1-14 shows a circle tangent to two lines.

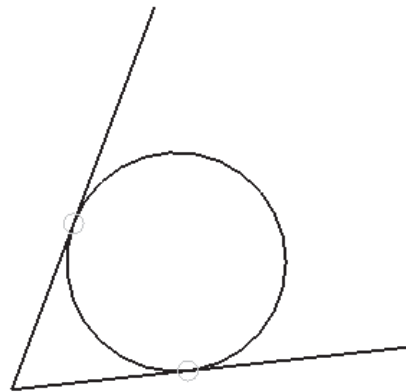


Figure 1-14 Drawing circle tangent to two lines

Drawing Ellipses

In Solid Edge, you can draw ellipses using the following two methods:

Drawing Ellipse by Specifying Three Points

Toolbar: Draw > Circle by Center > Ellipse by 3 Points



This method is used to draw an ellipse by specifying three points. The first two points are the first and second endpoints of the primary axis of the ellipse and the third point is a point on the ellipse. To draw an ellipse using this method, choose the

Ellipse by 3 Points button from the **Circle by Center** flyout in the **Draw** toolbar. You will be prompted to specify the first and second endpoints of the primary axis of the ellipse. After you specify these two points, a reference ellipse will be displayed on the screen and you will be prompted to specify a point on the ellipse. The primary axis will act as the major or the minor axis, depending on where you specify the point. Figure 1-15 shows a profile in which the cursor is moved to define the point on the ellipse after defining the primary axis. Note that you can also enter values in the **Ellipse** ribbon bar, which is displayed on invoking this tool.

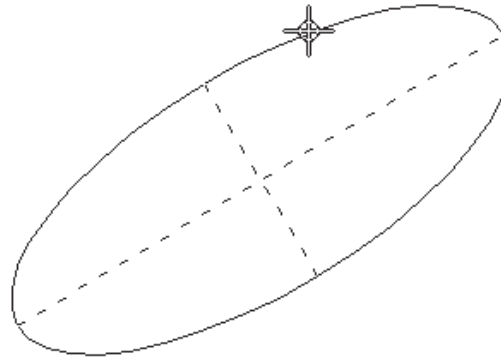


Figure 1-15 Drawing ellipse by specifying three points

Drawing the Center Point Ellipse

Toolbar: Draw > Circle by Center > Ellipse by Center



In this method, you need to define the center point of the ellipse first. After you define the center point of the ellipse, you will be prompted to specify the endpoint of the primary axis. Next, you will be prompted to specify the endpoint of the secondary axis. Alternatively, you can enter the values in the ribbon bar.

Placing Sketched Points

Toolbar: Draw > Line > Point



Points generally help as a reference in drawing the other sketched entities. To place a point, choose the **Point** button from the **Line** flyout in the **Draw** toolbar. You will be prompted to click for the point. You can place the point by defining its location in the drawing window or by entering its coordinates in the **Point** ribbon bar.

Drawing Arcs

In Solid Edge, you can draw arcs using the following three methods:

Drawing a Tangent or a Normal Arc

Toolbar: Draw > Tangent Arc



This method of drawing arcs is similar to drawing tangent and normal arcs from within the **Line** tool. On invoking this tool, you will be prompted to specify the start point of the arc. Move the cursor close to the endpoint of the entity where you want the tangent arc to start. You will notice that the endpoint relationship handle is displayed on the right of the cursor. This handle has a small inclined line with a point at the upper end, which suggests that if you select the point now, the endpoint of the entity will be snapped. Select the endpoint and then move the cursor; the intent zones will be displayed. Move the

cursor through a small distance in the required intent zone and then specify the endpoint of the arc. Alternatively, you can enter the radius and included angle of the arc in the **Arc** ribbon bar, which is displayed when you invoke this tool.

Drawing a Three-Point Arc

Toolbar: Draw > Tangent Arc > Arc by 3 Points



This method is used to draw an arc by specifying its start and endpoint, and a point on it. You can specify the radius of this arc in the ribbon bar. However, in this case, you will only be allowed to specify only the start point and the endpoint of the arc. The third point only specifies the direction in which the arc will be drawn. Figure 1-16 shows a three-point arc being drawn.

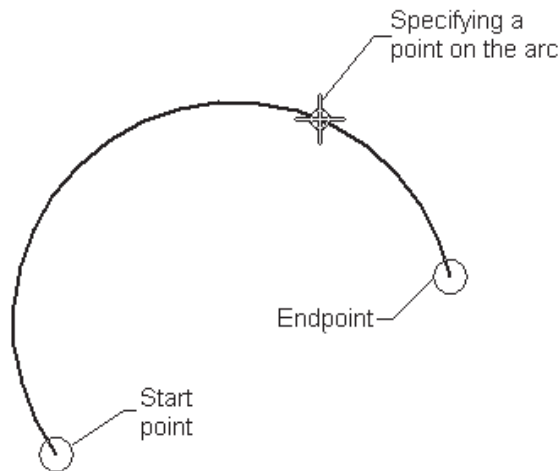


Figure 1-16 Drawing an arc using the Arc by 3 Points method

Drawing a Center Point Arc

Toolbar: Draw > Tangent Arc > Arc by Center



This method is used to draw an arc by specifying its center point, start point, and endpoint. On invoking this tool, you will be prompted to specify the center point of the arc. Next, you will be prompted to specify its start point and endpoint. Note that when you specify the start point of the arc after specifying the center point, the radius will be automatically defined. Therefore, the endpoint is used only to define the arc length. Figure 1-17 shows an arc being drawn using this method.

Drawing Rectangles

Toolbar: Draw > Rectangle



In Solid Edge, the rectangles are drawn by specifying three points. The first two points define the width of the rectangle and the third point defines the height. When you invoke this tool, you will be prompted to specify the first corner. Specify a point

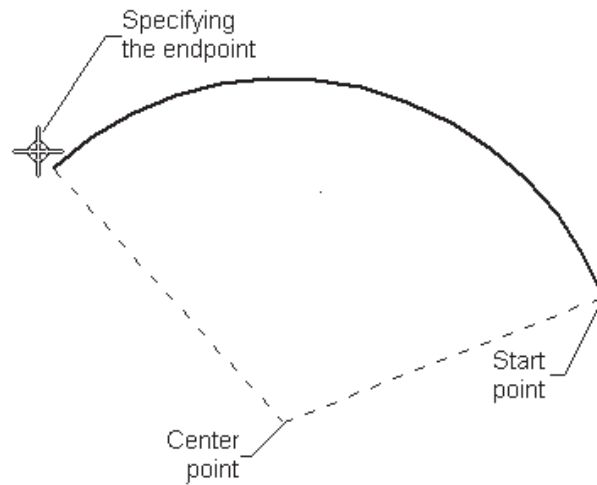


Figure 1-17 Drawing an arc using the **Arc by Center** method

in the drawing window to define the start point of the rectangle. Next, you will be prompted to specify the second point. This point will define the width of the rectangle. You can also define this point at an angle. As a result, the rectangle will be drawn at an angle. Finally, you will be prompted to specify a point to create the rectangle, which will define the height of the rectangle. Alternatively, you can specify the width, height, and angle of the rectangle in the **Rectangle** ribbon bar. Figure 1-18 shows a rectangle drawn at an angle.

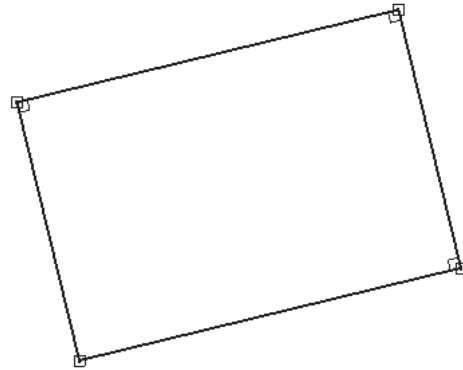


Figure 1-18 Rectangle drawn at an angle

This tool also enables you to draw a square.

To draw a square, specify the first two points to define the width of the rectangle. Next press and hold the SHIFT key down and then move the mouse a little; the square will be drawn.



Tip. You can also draw a rectangle by pressing and holding the left mouse button at a point and dragging the cursor across to define the opposite corner of the rectangle. When you release the left mouse button, the rectangle will be drawn.

Drawing Curves

Toolbar: Draw > Curve



The **Curve** tool allows you to draw curves using two methods: specifying points in the drawing window and dragging the cursor in the drawing window. These methods are discussed next.

Drawing a Curve by Dragging the Cursor

In this method, you need to press and hold the left mouse button down and drag the cursor to create the curve. A reference curve will be displayed in the drawing window as you drag the cursor. Once you release the left mouse button, a curve will be drawn that has exactly the same shape as the reference curve. Figure 1-19 shows a curve drawn using this method.

Drawing a Curve by Specifying Points in the Drawing Window

In this method, you need to continuously specify points on the curve to draw it. After specifying the first point, you do not need to drag the cursor. You can simply move the cursor and specify the second point. Continue this process until you have specified all points required to draw the curve. Figure 1-20 shows a curve drawn using this method.



Figure 1-19 Curve drawn by dragging the cursor in the drawing window

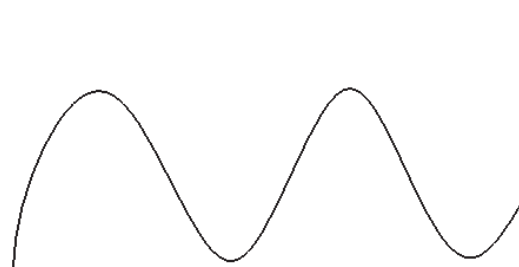


Figure 1-20 Curve drawn by specifying the points in the drawing window

Converting Sketched Entities into Curves

Toolbar: Draw > Curve > Convert to Curve



In Solid Edge, you can convert the sketched entities such as lines, arcs, circles, and ellipses into bezier spline curves using the **Convert to Curve** tool. On invoking this tool, you will be prompted to select an element to be converted into curve. As soon as you select the element, it is converted into a bezier spline curve. Note that you may not be able to view the changes in the sketched entity unless you select it. When you select the sketched entity, you will notice that the number of handles in it are increased and the control polygon is displayed on that entity. If you drag the converted entity using any of its handles, it will become curved.

Filleting Sketched Entities

Toolbar: Draw > Fillet



Filleting is defined as the process of rounding the sharp corners of a profile to reduce the stress concentration. You can create a fillet by removing the sharp corner and replacing it by the round. In Solid Edge, you can create a fillet between any two sketched entities. On invoking the **Fillet** tool, the **Fillet** ribbon bar will be displayed. Define the radius of the fillet in the **Radius** edit box in the ribbon bar and press ENTER. Now, one

by one select the two entities that you want to fillet; the fillet will be created. You can also directly select the sharp corner to be filleted. The two entities comprising the corner are will be in red when you move the cursor over the corner. Select the corner at this stage to create the fillet. Figure 1-21 shows a profile before and after filleting.

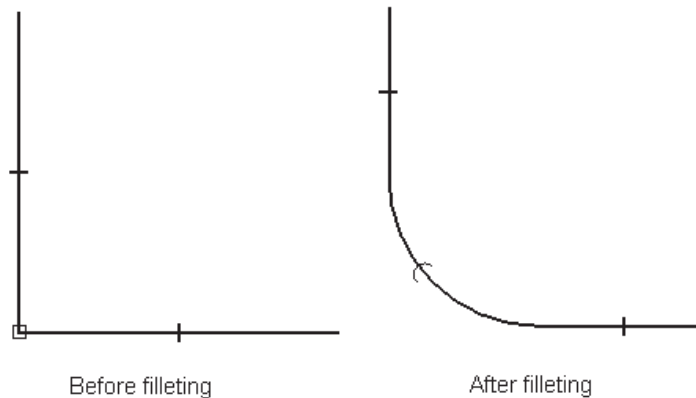


Figure 1-21 Sketch before and after creating the fillet

You are also given an option of retaining the sharp corner even after creating the fillet. If you want to retain the sharp corner also after filleting, choose the **No Trim** button from the **Fillet** ribbon bar and then select the corner to be filleted. The fillet will be created and the sharp corner will also be retained. Figure 1-22 shows a profile in which the fillet is created and the sharp corner is also retained.



Note

Ideally, the profiles that have the fillet created with the sharp corners retained may not give the desired result when used to create features. Therefore, they should be avoided.

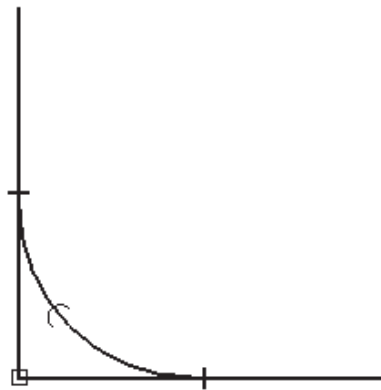


Figure 1-22 Retaining the sharp corner after creating the fillet

Chamfering Sketched Entities

Toolbar: Draw > Fillet > Chamfer



Chamfering is defined as the process of beveling the sharp corners of a profile to reduce the stress concentration. You can create the chamfer only between two linear entities. The chamfer can be created by defining the distance of the corner being chamfered from the two edges of the profile or by defining the angle of the chamfer and the distance along one of the edges. To create the chamfer, invoke the **Chamfer** tool; the **Chamfer** ribbon bar will be displayed. You can specify the angle, setback A, and setback B values using this ribbon bar. The setback A and the setback B values define the chamfer distances along the first and the second edge, respectively. Note that you can specify any two of the three

values. The third value is automatically updated on the basis of the two values you define.

After setting any two values in the **Chamfer** ribbon bar, select the first and the second line to be chamfered. The preview of the resulting chamfer will be displayed; click to create the chamfer. Note that by default, the first line is taken as the setback A element and the second line is taken as the setback B element. If you want to reverse the order, move the cursor over the first line; the preview will change automatically. Now, the second line is taken as the setback A element and the first line is taken as the setback B element. Figure 1-23 shows the preview of the chamfer.

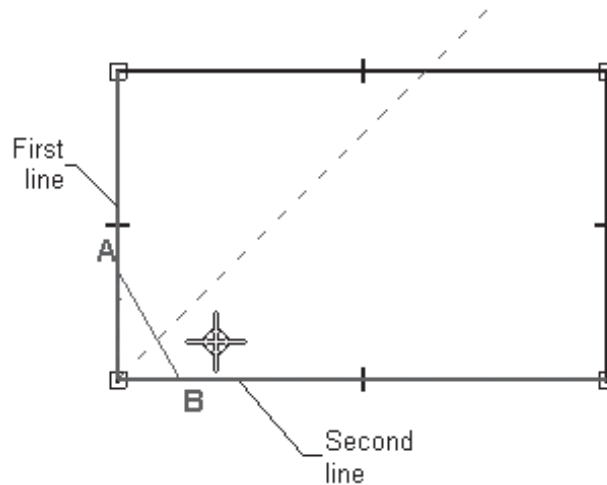


Figure 1-23 Selecting lines to create chamfer



Tip. In *Solid Edge*, you can create fillets or chamfers by simply dragging the cursor across the entities that you need to fillet or chamfer. For example, if you want to chamfer two lines, invoke the **Chamfer** tool and drag the cursor across them. The corner of these two lines will be chamfered and the angle and distances of the chamfer will depend on how far from the corner you dragged the mouse.

SELECTING THE SKETCHED ENTITIES

When you choose the **Select Tool** button from the **Draw** toolbar, the select mode will be invoked. In this mode, you can select the sketched entities available in the drawing window by clicking on them. The selected entities will be highlighted in yellow. You can select all entities in the drawing window by pressing the CTRL+A key.

In addition to these methods of selection, you can also select entities by dragging a rectangular box or drawing a polygonal fence around them. You can choose the button of the rectangular box or polygonal fence from the ribbon bar. To drag the box or the fence, press and hold the left mouse button down and drag the cursor in the drawing window. The entities that will be selected will depend on the option selected from the **Fence Filter** drop-down list available in the ribbon bar that is displayed in the select mode. These options are discussed next.

Inside

This is the default selection mode and it ensures that only those entities are selected that lie completely inside the box that is created when you hold down the left mouse button and drag the cursor.

Outside

This selection mode ensures that only those entities are selected that lie completely outside the box that is created when you hold down the left mouse button and drag the cursor.

Overlapping

This selection mode ensures that all entities that lie partially inside the box or even touch the box are selected.

Inside and Overlapping

This mode is a combination of the inside and the overlapping modes. This selection mode ensures that all entities that lie partially or fully inside the box are selected.

Outside and Overlapping

This mode is a combination of the outside and the overlapping modes. This selection mode ensures that all entities that partially inside the box or are completely outside the box are selected.

DELETING THE SKETCHED ENTITIES

To delete the sketched entities, select them using any of the object selection methods discussed above. The selected entities turn yellow in color. Next, press the DELETE key on the keyboard. All selected entities will be deleted.

TUTORIALS

As mentioned in Introduction, Solid Edge is parametric in nature. Therefore, you can draw a profile of any dimensions and then modify its size by changing the values of the dimensions. However, in this chapter, you will use the ribbon bars to draw the profile to the exact dimensions. This will help you in improving your sketching skills.

Tutorial 1

In this tutorial, you will draw the profile for the model shown in Figure 1-24. The profile to be drawn is shown in Figure 1-25. Do not dimension the profile because the dimensions are just for your reference.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

- Start Solid Edge in the **Part** environment.
- Exit the **Protrusion** tool and invoke the **Sketch** tool.
- Select the x-z plane as the sketching plane and invoke the sketching environment.

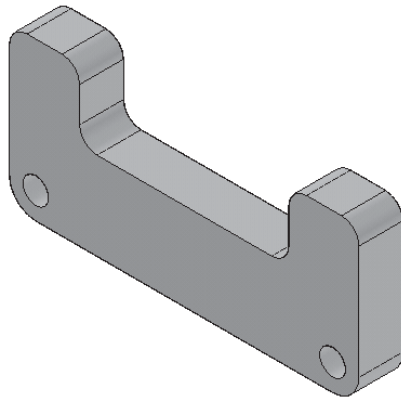


Figure 1-24 Model for Tutorial 1

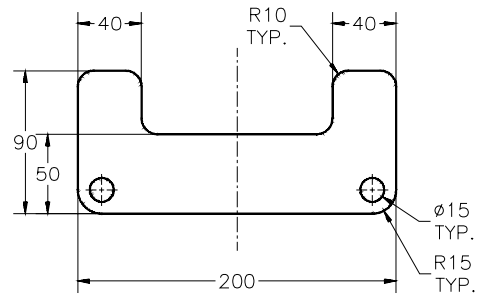


Figure 1-25 Profile for Tutorial 1

- d. Draw the outer loop of the profile using the **Line** tool.
- e. Fillet the sharp corners of the outer loop using the **Fillet** tool.
- f. Draw the circles using the centers of the fillets to complete the profile.
- g. Save the file and close it.

Starting Solid Edge and Selecting the Sketching Plane

The profile of the model will be created in the sketching environment of Solid Edge. You can invoke the sketching environment in the **Part** environment, whenever required. Therefore, you need to start a new part file first.

1. Choose the **Start** button available at the lower left corner of the screen to display a menu with additional options.
2. Choose **All Programs** (or **Programs**) > **Solid Edge V16** > **Part** from the start menu to start a new part file.

After Solid Edge is started and a new part file loaded, the **Protrusion** button is chosen in the **Features** toolbar and the **Protrusion** ribbon bar is displayed. Also, you are prompted to select a planar face or a reference plane to define the sketching plane for drawing a profile.

If the base feature of a model consists of multiple closed loops, it is recommended that you draw the profile of the base feature as an independent sketch using the **Sketch** tool and not as the profile of the **Protrusion** tool. This is because the independent sketch can be used to create multiple features. For example, you can include the circles in the sketch and then use them separately to create the cutouts in the base feature. Therefore, you need to exit the **Protrusion** tool and invoke the **Sketch** tool to draw an independent sketch.

3. Choose the **Sketch** button from the **Features** toolbar; the **Sketch** ribbon bar is displayed and you are prompted to select a planar face or a reference plane.



4. Select the x-z plane to draw the profile; the sketching environment is invoked and the sketch plane orients itself parallel to the screen. Also, the **Line** tool is automatically invoked.

Drawing Lines of the Outer Loop

You can draw the outer loop using the **Line** tool. The sharp corners will be rounded using the **Fillet** tool. In this chapter, you will use the ribbon bar to enter the exact values of the sketched entities.

Because the **Line** tool is active, its ribbon bar is displayed on top of the **EdgeBar** and you are prompted to specify the start point of the line. You can start drawing the line from the origin, which is the point where the x-y, y-z, and x-z planes intersect and hence, its coordinates are 0,0,0. In the current view, the origin is the intersection point of the two planes displayed as horizontal and vertical lines.


1. Move the cursor close to the origin. One of the two planes, which are displayed as blue horizontal or vertical lines, is highlighted and the midpoint relationship handle is displayed.
2. Click to specify the start point of the line.

The point you specify is selected as the start point of the line and the endpoint is attached to the cursor. As you move the cursor on the screen, the line stretches and its length and angle values in the ribbon bar are dynamically modified.

Next, you need to specify the other points to define the first line and the other lines. This will be done using the **Length** and **Angle** edit boxes in the **Line** ribbon bar.

3. Type **200** as the value in the **Length** edit box of the **Line** ribbon bar and press ENTER. Now, type **0** as the value in the **Angle** edit box and press ENTER.

You will notice that the line is drawn, but it is not completely displayed in the current display. To include it in the current display, you need to modify the drawing display area using the **Fit** tool.

4. Choose the **Fit** button from the **Main** toolbar; the current drawing display area is modified and the line is displayed completely in the current view. Also, the **Line** tool is still active and you are prompted to specify the second point of the line. 

5. Type **90** as the value in the **Length** edit box and press ENTER. Again, type **90** as the value in the **Angle** edit box and press ENTER. A vertical line of 90 length is drawn.
6. Type **40** as the value in the **Length** edit box and press ENTER. Type **180** as the value in the **Angle** edit box and press ENTER. A horizontal line of 40 length is drawn toward the left of the last line.
7. Type **40** as the value in the **Length** edit box and press ENTER. Type **-90** as the value in

- the **Angle** edit box and press ENTER. A vertical line of 40 length is drawn downward.
8. Type **120** as the value in the **Length** edit box and press ENTER. Type **180** as the value in the **Angle** edit box and press ENTER. A horizontal line of 180 length is drawn.
 9. Move the cursor vertically upward. A rubber-band line is displayed with its starting point at the endpoint of the previous line and the endpoint attached to the cursor.
 10. Move the cursor once toward the vertical line of 40 length drawn earlier and then move it back in the vertical direction from the start point of this line. When the line is vertical, the vertical relationship handle is displayed.
 11. Move the cursor vertically upward until the horizontal alignment indicator is displayed from the top endpoint of the vertical line of 40 length. Note that at this point, the length in the **Length** edit box of the ribbon bar is **40** and the angle is **90**. Click to specify the endpoint of this line.
 12. Move the cursor horizontally toward the left and make sure that the horizontal relationship handle is displayed. Click to specify the endpoint of the line when the vertical alignment indicator is displayed from the vertical plane. If the alignment indicator is not displayed, move the cursor once on the vertical plane and then move it back.
 13. Move the cursor vertically downward to the origin. If the first line is not highlighted in red, move the cursor over it once and then move it back to the origin. The endpoint relationship handle is displayed. This relationship ensures that this line ends at the start point of the first line.
 14. Click to specify the endpoint of the line when the endpoint relationship handle is displayed. Choose the **Fit** button to fit the sketch in the drawing window.
 15. Press the ESC key from the keyboard to exit the **Line** tool. The sketch after drawing the lines is shown in Figure 1-26.

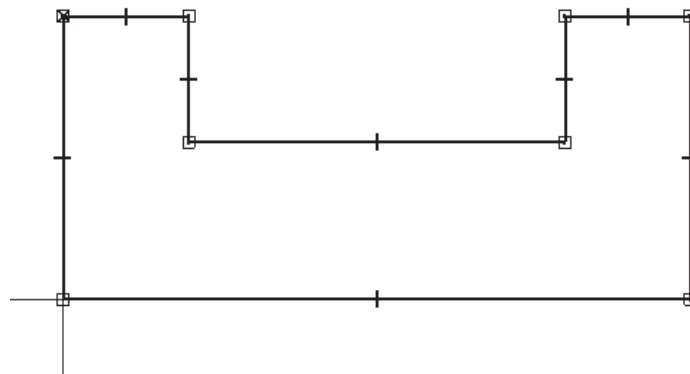



Figure 1-26 Sketch after drawing lines

Filleting the Sharp Corners

Next, you need to fillet the sharp corners so that there are no sharp edges in the final model. You can fillet the corners using the **Fillet** tool.

1. Choose the **Fillet** button from the **Draw** toolbar. Note that if you had invoked the **Chamfer** tool earlier, it will now be displayed as the default tool in the **Draw** toolbar. In this case, press and hold the left mouse button down on the **Chamfer** button to display a flyout. Choose the **Fillet** button from this flyout to invoke the **Fillet** tool. 

The **Fillet** ribbon bar is displayed. To fillet any sharp corner, you first need to specify the fillet radius. You can fillet the bottom left and bottom right corners first and then the remaining corners. This is because the fillet radii of the bottom left and bottom right corners are the same and those of the remaining corners are the same.

2. Type **15** as the value in the **Radius** edit box in the **Fillet** ribbon bar and press ENTER. Now, move the cursor over the bottom left corner of the sketch; the two lines comprising this corner are highlighted in red.
3. Now, click to select this corner; the fillet is created at the bottom left corner.
4. Similarly, move the cursor over the bottom right corner and click to select it when the two lines that form this corner are highlighted in red.

Next, you need to modify the fillet radius value and fillet the remaining corners.

5. Type **10** as the value in the **Radius** edit box in the ribbon bar and press ENTER.
6. Select the remaining corners of the sketch one by one and fillet them with a radius of 10. The sketch, after creating fillets, is shown in Figure 1-27.

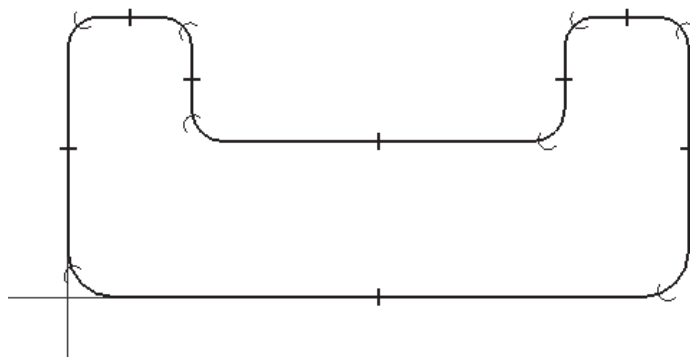



Figure 1-27 Sketch after creating fillets

Drawing Circles

Finally, you need to draw circles to complete the profile. The circles will be drawn using the **Circle by Center** tool. You will use the center points of the fillets as the center points of the circles.

1. Choose the **Circle by Center** button from the **Draw** toolbar. The **Circle** ribbon bar is displayed and you are prompted to select the center point of the circle. 
2. Type **15** as the value in the **Diameter** edit box of the **Circle** ribbon bar and press ENTER. A circle of the specified diameter is attached to the cursor and you are prompted to specify the center point of the circle. The circle attached to the cursor moves as you move the cursor on the screen.
3. Move the cursor over the fillet on the bottom left corner once.

The fillet is highlighted in red and the center point of the circle is displayed. The center point is represented by a plus sign (+).
4. Move the cursor over the center point of the fillet represented by the plus sign. The fillet is highlighted in red and the concentric relationship handle is displayed on the right of the cursor.
5. Click to specify this point as the center point of the circle. A circle is drawn at this point and you are again prompted to specify the center point of the circle.
6. Move the cursor over the bottom right fillet so that its center point is also displayed.
7. Move the cursor over the center point of the bottom right fillet and click when the concentric relationship handle is displayed. The final profile for Tutorial 1 is shown in Figure 1-28.

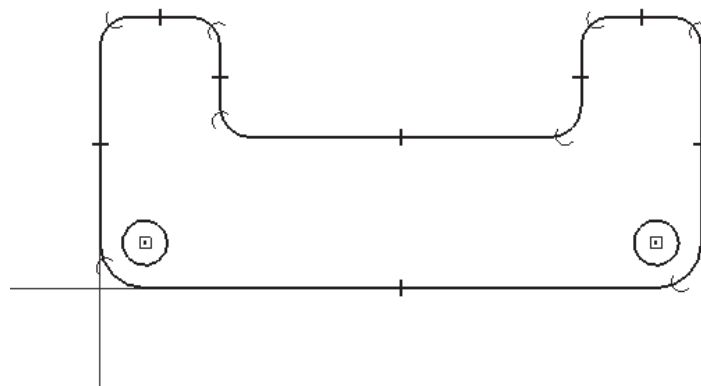


Figure 1-28 Final profile for Tutorial 1

Saving the File

It is recommended that you exit the sketching environment before saving the file. This is because you cannot close a file in the sketching environment.

You can exit the sketching environment by choosing the **Finish** button from the ribbon bar that is displayed in the select mode.

1. Press the ESC key from the keyboard to exit the current tool.
2. Choose the **Finish** button from the ribbon bar to close the sketching environment; the **Sketch** ribbon bar is displayed. Also, the current view is automatically changed to the isometric view. Choose the **Fit** button to fit the sketch in the drawing window.
3. Enter the name of the sketch as **Base Sketch** in the **Name** edit box in the ribbon bar. The sketch is displayed by this name in the **EdgeBar**.
4. Choose the **Save** button from the **Main** toolbar; the **Part1 Properties** dialog box is displayed. This dialog box can be used to specify the properties of the part file.
5. Choose **OK** from the **Part1 Properties** dialog box; the **Save As** dialog box is displayed.

It is recommended that you create separate folder for every chapter in the textbook.

6. Browse to the *My Documents* folder and then create a folder with the name *Solid Edge* in the *My Documents* folder. Make the *Solid Edge* folder current and then create a folder with the name *c01* in this folder.
7. Make the *c01* folder current and save the file with the name *c01tut1.par*. The location of this file is given below.

`\My Documents\Solid Edge\c01\c01tut1.par`

8. Choose **File > Close** from the menu bar to close the file.

Tutorial 2

In this tutorial, you will draw the profile of the model shown in Figure 1-29. The profile to be drawn is shown in Figure 1-30. Do not dimension the profile because the dimensions are just for your reference.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

- a. Start a new part file.
- b. Exit the **Protrusion** tool and invoke the **Sketch** tool.
- c. Select the y-z plane as the sketching plane and invoke the sketching environment.

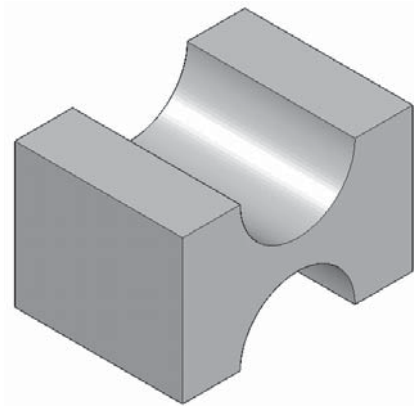


Figure 1-29 Model for Tutorial 2

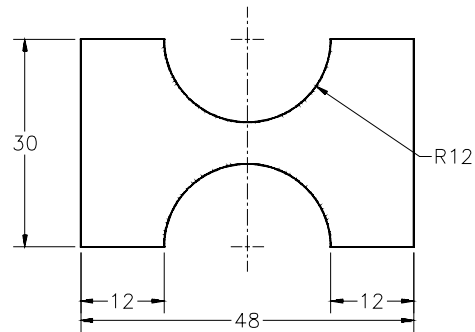


Figure 1-30 Profile for Tutorial 2

- d. Draw the profile of the model using the **Line** tool.
- e. Save the file and close it.

Starting a New Part File and Selecting the Sketching Plane

You can start a new part file by choosing the **New** button from the **Main** toolbar, which remains on the screen after you close all files.

1. Choose the **New** button from the **Main** toolbar; the **New** dialog box is displayed.
2. Select **Normal.par**, as shown in Figure 1-31, and choose **OK** to start a new part file.

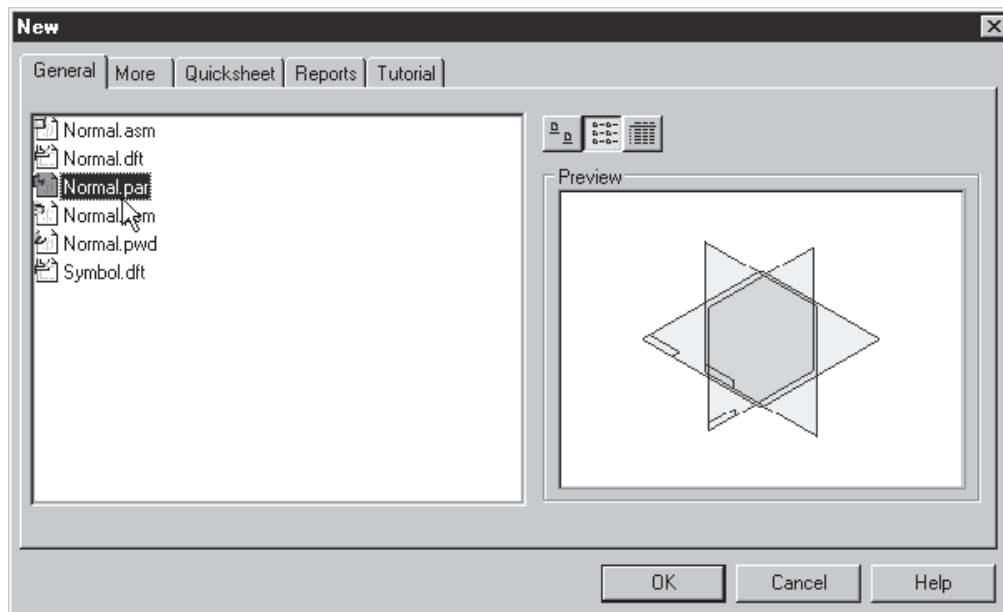



Figure 1-31 The **New** dialog box to start a new file in Solid Edge

In the new part file, the **Protrusion** button is chosen in the **Features** toolbar and the **Protrusion** ribbon bar is displayed. You are prompted to select a planar face or a reference plane.

3. Choose the **Sketch** button from the **Features** toolbar; the **Sketch** ribbon bar is displayed and you are prompted to select a planar face or a reference plane. 
4. Select the y-z plane to draw the profile; the sketching environment is invoked and the sketch plane orients itself parallel to the screen. Also, the **Line** tool is automatically invoked.

Drawing the Profile

Because the **Line** tool is active, its ribbon bar is displayed on top of the **EdgeBar** and you are prompted to specify the start point of the line. You can start drawing the line from the origin.

1. Move the cursor close to the origin. One of the two planes, which are displayed as blue horizontal or vertical lines, is highlighted and the midpoint relationship handle is displayed.
2. Click to specify the start point of the line.

The point you specify is selected as the start point of the line and the endpoint is attached to the cursor. When you move the cursor on the screen, the line stretches and its length and angle values in the ribbon bar are dynamically modified.

3. Type **12** as the value in the **Length** edit box of the **Line** ribbon bar and press ENTER. Type **0** as the value in the **Angle** edit box and press ENTER.

The first line is drawn and another rubber-band line is displayed with the start point at the endpoint of the previous line and the endpoint attached to the cursor. But because the next entity is an arc, you need to invoke the arc mode.

4. Press the A key from the keyboard to invoke the arc mode. Alternatively, you can also choose the **Arc** button from the ribbon bar to invoke the arc mode.

A rubber-band arc is displayed with the start point fixed at the endpoint of the last line and the endpoint attached to the cursor. Also, the intent zones are displayed at the start point of the arc.

5. Move the cursor to the start point of the arc and then move it vertically upward through a small distance. Now, move the cursor toward the right. You will notice that a normal arc starts from the endpoint of the last line.
6. Enter **12** as the radius and **180** as the angle of the arc in the **Radius** and **Sweep** edit boxes, respectively, in the ribbon bar.

The preview of the resulting arc is displayed, but the arc is still not drawn. To draw the arc, you need to specify a point on the screen with the values mentioned in the ribbon bar.

7. Move the cursor close to the horizontal plane and click when the plane is highlighted in red. The arc is drawn and the line mode is invoked again.
8. Enter **12** as the length and **0** as the angle in the **Length** and **Angle** edit boxes, respectively. Choose the **Fit** button from the **Main** toolbar to fit the sketch in the drawing window.
9. Enter **30** as the length and **90** as the angle in the **Length** and **Angle** edit boxes, respectively.
10. Move the cursor horizontally toward the left. Make sure the horizontal relationship handle is displayed. Click to specify the endpoint of the line when the vertical alignment indicator is displayed from the endpoint of the arc.

Next, you need to draw the arc. Therefore, you need to invoke the arc mode.

11. Press the A key from the keyboard to invoke the arc mode. A rubber-band arc is displayed with its start point fixed at the endpoint of the last line.
12. Move the cursor to the start point of the arc and then move it vertically downward through a small distance. When the normal arc appears, move the cursor toward the left.
13. Move the cursor over the lower arc once and then move it toward the left, in line with the upper right horizontal line from where this arc starts.

The horizontal alignment indicator is displayed originating from the upper left horizontal line. At the point where the cursor is vertically in line with the start point of the lower arc, the vertical alignment indicator appears from the start point of the lower arc, as shown in Figure 1-32.

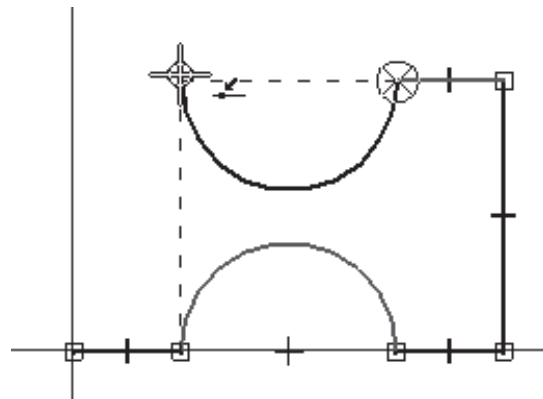


Figure 1-32 Horizontal and vertical alignment indicators displayed to define the endpoint of the arc

14. Click to define the endpoint of the arc when the horizontal and vertical alignment indicators are displayed. The arc is drawn and the line mode is invoked again.
15. Move the cursor horizontally toward the left and click to define the endpoint of the line when the vertical reference plane is highlighted in red.
16. Move the cursor to the first line and then move it to the start point of this line. The endpoint relationship handle is displayed.

17. Click to define the endpoint of this line when the endpoint relationship handle is displayed. The final profile of the model is shown in Figure 1-33.

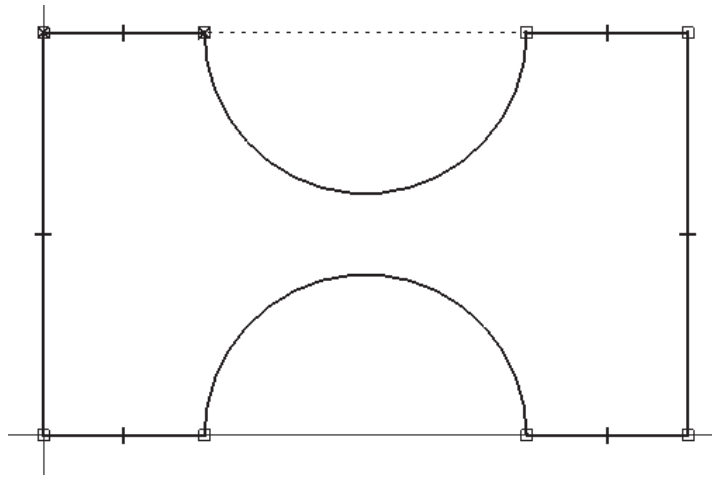


Figure 1-33 Profile for Tutorial 2

Saving the File

1. Press the ESC key button to exit the current tool.
2. Choose the **Finish** button from the ribbon bar; the sketching environment is closed and the **Sketch** ribbon bar is displayed. Also, the current view is automatically changed to the isometric view.
3. Enter the name of the sketch as **Base Sketch** in the **Name** edit box in the ribbon bar. The sketch will be displayed by this name in the **EdgeBar**.
4. Choose the **Save** button from the **Main** toolbar. The **Part2 Properties** dialog box is displayed. This dialog box can be used to specify the properties of the part file.

Note that if you had started a new session of Solid Edge before starting this tutorial, the name of the dialog box will be **Part1 Properties**.

5. Choose **OK** from the **Part2 Properties** dialog box; the **Save As** dialog box is displayed.
6. Browse to the *My Documents\Solid Edge\c01* folder, if it is not the current folder. Save the file with the name *c01tut2.par*. The location of this file is given below.

\My Documents\Solid Edge\c01\c01tut2.par

8. Choose **File > Close** from the menu bar to close the file.

Tutorial 3

In this tutorial, you will draw the profile for the base feature of the model shown in Figure 1-34. The profile to be drawn is shown in Figure 1-35. Do not dimension the profile because the dimensions are just for your reference. (Expected time: 30 min)

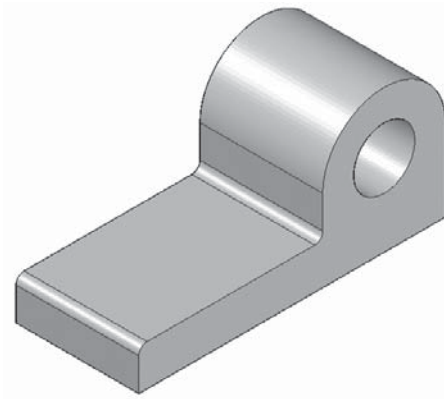


Figure 1-34 Model for Tutorial 3

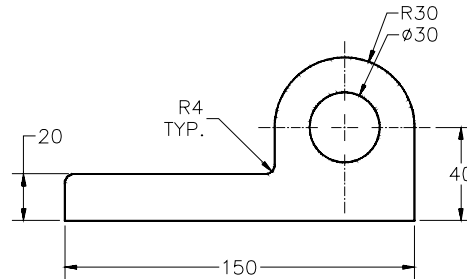


Figure 1-35 Profile for Tutorial 3

The following steps are required to complete this tutorial:


- Start a new part file.
- Exit the **Protrusion** tool and invoke the **Sketch** tool.
- Select the y-z plane as the sketching plane and invoke the sketching environment.
- Draw the profile of the model using the **Line** tool.
- Fillet the two corners of the outer loop and then draw the inner circle.
- Save the file and close it.

Starting a New Part File and Selecting the Sketching Plane

As mentioned earlier, you can start a new part file by choosing the **New** button from the **Main** toolbar, which remains on the screen after you close all files.

- Choose the **New** button from the **Main** toolbar; the **New** dialog box is displayed.
- Select **Normal.par** and choose **OK** to start a new part file.

The **Protrusion** button is chosen by default in the new part file and the **Protrusion** ribbon bar is displayed. You are prompted to select a planar face or a reference plane.

- Choose the **Sketch** button from the **Features** toolbar. The **Sketch** ribbon bar is displayed and you are prompted to select a planar face or a reference plane. 
- Select the y-z plane to draw the profile; the sketching environment is invoked and the sketch plane orients itself parallel to the screen. Also, the **Line** tool is automatically invoked.

Drawing the Profile

Because the **Line** tool is active, its ribbon bar is displayed on top of the **EdgeBar** and you are prompted to specify the start point of the line. You can start drawing the line from the origin.

1. Move the cursor close to the origin; one of the two planes, which are displayed as blue horizontal or vertical lines, is highlighted and the midpoint relationship handle is displayed.
2. Click to specify the start point of the line.

The point you specify is selected as the start point of the line and the endpoint is attached to the cursor.

3. Type **150** as the value in the **Length** edit box of the **Line** ribbon bar and press ENTER. Type **0** as the value in the **Angle** edit box and press ENTER.

The line of 150 length is drawn, but is not completely visible on the screen. To display the complete line on the screen, you need to modify the drawing display area using the **Fit** tool.

4. Choose the **Fit** button from the **Main** toolbar; the line is now completely displayed in the current view.



5. Enter **40** as the length and **90** as the angle of the line in the **Length** and **Angle** edit boxes, respectively, of the ribbon bar.

Next, you need to draw a tangent arc from this point.

6. Press the A key from the keyboard to invoke the arc mode. Move the cursor back to the start point of the arc and then move it vertically upward through a small distance.
7. Move the cursor toward the left when the tangent arc is displayed. Enter **30** as the radius and **180** as the angle in the **Radius** and **Sweep** edit boxes, respectively.
8. Specify a point in the drawing window to place the arc. The arc is drawn and the line mode is invoked again.
9. Enter **20** as the length and **-90** as the angle of the line in the **Length** and **Angle** edit boxes, respectively.
10. Move the cursor horizontally toward the left and make sure the horizontal relationship handle is displayed. Click to define the endpoint of the line when the vertical plane is highlighted in red.

11. Move the cursor to the first line to highlight it and then move it to the start point of the first line. The first line is highlighted in red and the endpoint relationship handle is displayed.
12. Click to specify the endpoint of the line when the endpoint relationship handle is displayed. The profile, after drawing the outer loop, is displayed in Figure 1-36.

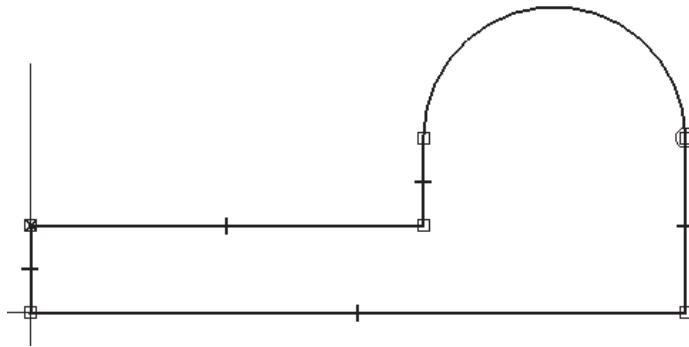


Figure 1-36 Outer loop of the profile for Tutorial 3

Filleting the Sharp Corners

Next, you need to fillet the sharp corners so that there are no sharp edges in the final model. You can fillet the corners using the **Fillet** tool.

1. Choose the **Fillet** button from the **Draw** toolbar. The **Fillet** ribbon bar is displayed.
2. Type **4** as the value in the **Radius** edit box in the **Fillet** ribbon bar and press ENTER. Now, move the cursor over the corner where the outer left vertical line and the upper horizontal line intersect. The two lines comprising this corner are highlighted in red.
3. Now, click to select this corner; the fillet is created at this corner.
4. Similarly, move the cursor over the corner where the upper horizontal line intersects the vertical line originating from the left endpoint of the arc. Click to select it when the two lines that form this corner are highlighted in red.

Drawing the Circle

Next, you need to draw a circle to complete the profile. The circle will be drawn using the **Circle by Center** tool.

1. Choose the **Circle by Center** button from the **Draw** toolbar; the **Circle** ribbon bar is displayed.
2. Enter **30** as the value of the diameter of the circle in the **Diameter** edit box.
3. Move the cursor over the arc of 30 radius. The arc is highlighted in red and its center point is displayed, which is represented by a plus sign (+).
4. Move the cursor over the center point of the arc and click to define the center point of the circle when the concentric relationship handle is displayed.

This completes the profile. The final profile for Tutorial 3 is shown in Figure 1-37.

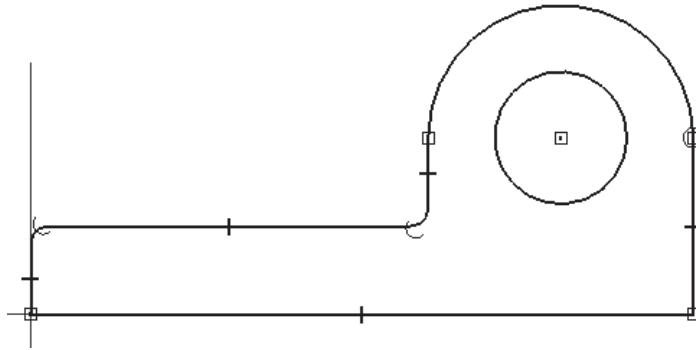


Figure 1-37 Completed profile for Tutorial 3

Saving the File

1. Press the ESC key from the keyboard to exit the current tool.
2. Choose the **Finish** button from the ribbon bar; the sketching environment is closed and the **Sketch** ribbon bar is displayed. Also, the current view is automatically changed to the isometric view.
3. Enter the name of the sketch as **Base Sketch** in the **Name** edit box in the ribbon bar. The sketch is displayed by this name in the **EdgeBar**.
4. Choose the **Save** button from the **Main** toolbar; the **Part3 Properties** dialog box is displayed. This dialog box can be used to specify the properties of the part file.
5. Choose **OK** from the **Part3 Properties** dialog box; the **Save As** dialog box is displayed.

6. Browse to the *My Documents\Solid Edge\c01* folder, if it is not current. Save the file with the name *c01tut3.par*. The location of this file is given below.

\My Documents\Solid Edge\c01\c01tut3.par

7. Choose **File > Close** from the menu bar to close the file.

Self-Evaluation Test

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

1. Most of the designs created in a solid modeling tool consist of the profile-based features, placed features, and reference features. (T/F)
2. If the base feature of a model consists of multiple closed loops, it is recommended that you draw the profile of the base feature as an independent sketch using the **Sketch** tool. (T/F)
3. You can use the ribbon bars to specify the exact values of the sketched entities. (T/F)
4. The **Sketch** button is chosen by default when you start a new part file. (T/F)
5. You can restore the original orientation of the sketching plane using the _____ tool in the **Main** toolbar.
6. You can invoke the arc mode within the **Line** tool by pressing the _____ key from the keyboard.
7. You can bevel the corners in the sketch using the _____ tool.
8. You can retain the sharp corners even after filleting them by choosing the _____ button from the **Fillet** ribbon bar.
9. Pressing the _____ key after defining the first edge of the rectangle results in a square.
10. You can exit the sketching environment by choosing the _____ button from the ribbon bar that is displayed when you choose the **Select Tool** button.

Review Questions

Answer the following questions.

- Which one of the following options is selected from the **New** dialog box to start a new part file?
(a) **Normal.asm** (b) **Normal.dft**
(c) **Normal.par** (d) **Normal.psm**
- Which one of the following tools is used to round the sharp corners in a sketch?
(a) **Fillet** (b) **Chamfer**
(c) **Round** (d) None
- Which edit box in the arc mode replaces the **Angle** edit box in the **Line** ribbon bar?
(a) **Arc** (b) **Sweep**
(c) **Value** (d) None
- In Solid Edge, how many methods can be used to draw arcs?
(a) 4 (b) 3
(c) 6 (d) 5
- By using which one of the following tools can you convert an existing sketched entity into a bezier spline curve?
(a) **Convert to Sketch** (b) **Convert to Arc**
(c) **Convert** (d) **Convert the Curve**
- The part file in Solid Edge is saved with .prt extension. (T/F)
- You can select the entities by dragging a box around them. (T/F)
- If **Overlapping** is the current selection mode, all entities that lie inside the box or even intersect the box are selected. (T/F)
- In Solid Edge, you can create fillets or chamfers by simply dragging the cursor across the entities that you want to fillet or chamfer. (T/F)
- You can also draw a rectangle by pressing and holding the left mouse button at a point and dragging the cursor across to define the opposite corner of the rectangle. (T/F)

Exercises

Exercise 1

Draw the profile of the base feature of the model shown in Figure 1-38. The profile to be drawn is shown in Figure 1-39. Do not dimension the profile because the dimensions are just for your reference. **(Expected time: 30 min)**

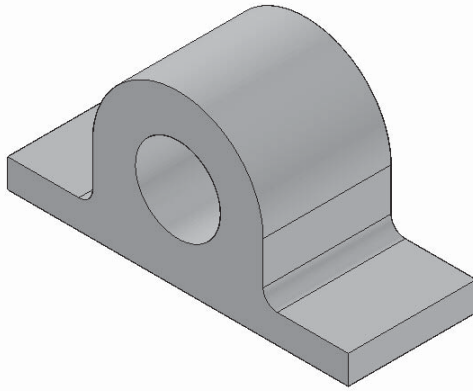


Figure 1-38 Model for Exercise 1

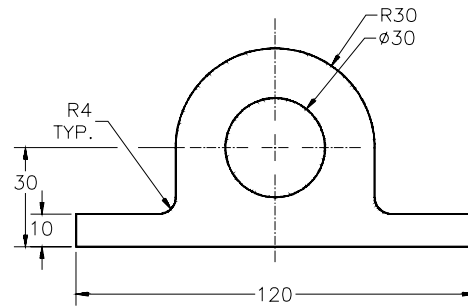


Figure 1-39 Profile for Exercise 1

Exercise 2

Draw the profile of the base feature of the model shown in Figure 1-40. The profile to be drawn is shown in Figure 1-41. Do not dimension the profile because the dimensions are just for your reference. **(Expected time: 30 min)**

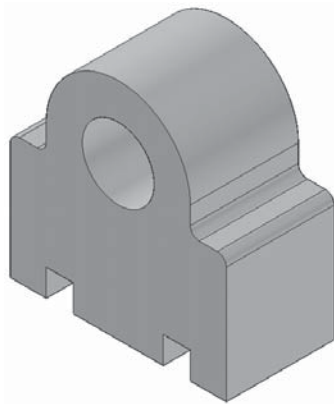


Figure 1-40 Model for Exercise 2

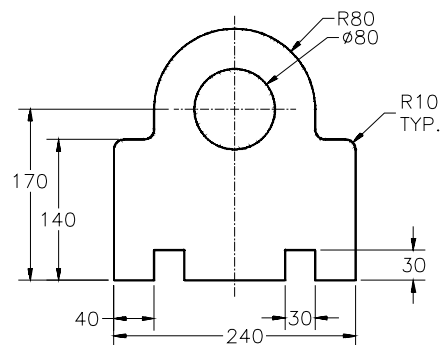


Figure 1-41 Profile for Exercise 2

Answers to Self Evaluation Test

1. T, 2. T, 3. T, 4. F, 5. Sketch View, 6. A, 7. Chamfer, 8. No Trim, 9. SHIFT, 10. Finish