



Chapter 2

Drawing Sketches for Solid Models

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the need of the sketching environment.*
- *Understand various sketching tools.*
- *Understand various drawing display tools.*
- *Use various selection methods.*
- *Delete sketched entities.*

THE SKETCHING ENVIRONMENT

Most designs created in solid modeling consist of 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 created by using these entities. Generally, a profile-based feature is the base feature or the first feature. For example, the solid model shown in Figure 2-1 is created by using the profile shown in Figure 2-2.

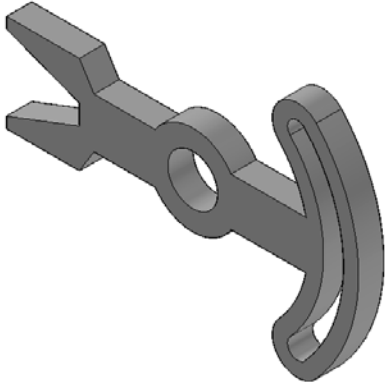


Figure 2-1 Solid model

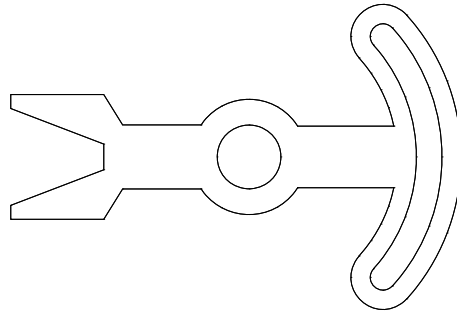


Figure 2-2 Profile of the solid model shown in Figure 2-1

In most designs, first you need to invoke the sketching environment and then create the profile of the model in it. After creating the profile, you need to 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 to start a new document in the **Part** environment. The first one is to start Solid Edge and then use the welcome screen to start a new file in the **Part** environment. The second one is to start a new part document by using the **New** dialog box. These methods are discussed next.

Starting the Part Environment in Solid Edge

To start the **Part** environment, first you need to start Solid Edge. To do so, choose the **Start** button at the lower left corner of the screen; the taskbar menu will be invoked. Next, choose **All Programs (Programs) > Solid Edge ST2 > Solid Edge** from the taskbar. Alternatively, you can double-click on the shortcut icon of **Solid Edge ST2** on the desktop of your computer.

The system will prepare to start Solid Edge. Once all files are loaded, the Solid Edge ST2 window will be displayed along with the welcome screen, as shown in Figure 2-3.



By default, only the environments of synchronous technology are available in the **Create** area of the welcome screen. To activate traditional environments, choose the **Application Button** from that is available at the top left corner of the welcome screen; a flyout will be displayed. Choose the **Solid Edge Options** button from the flyout; the **Solid Edge Options** dialog box will be displayed. Choose the **User Profile** button from the left pane

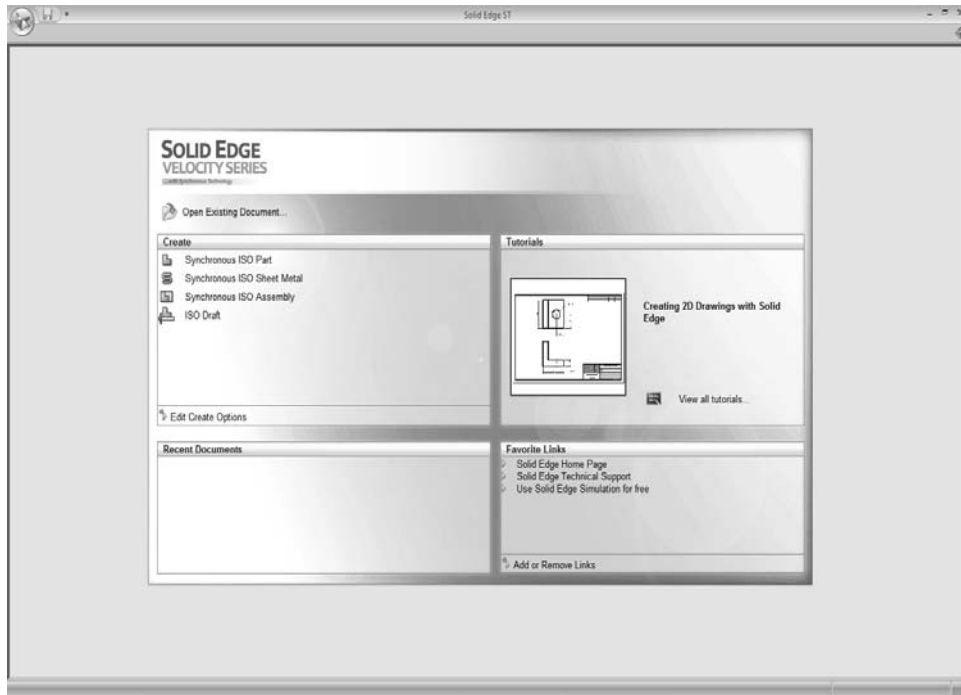


Figure 2-3 Welcome screen of Solid Edge ST2

of this dialog box, as shown in Figure 2-4; its corresponding options will be displayed in the right pane. Click on the **User type** drop-down list; a drop-down list will be displayed. Next, select the **Traditional and Synchronous** option from the drop-down list and choose **OK**; the Traditional as well as Synchronous environments will be displayed in the **Create** area of the welcome screen.

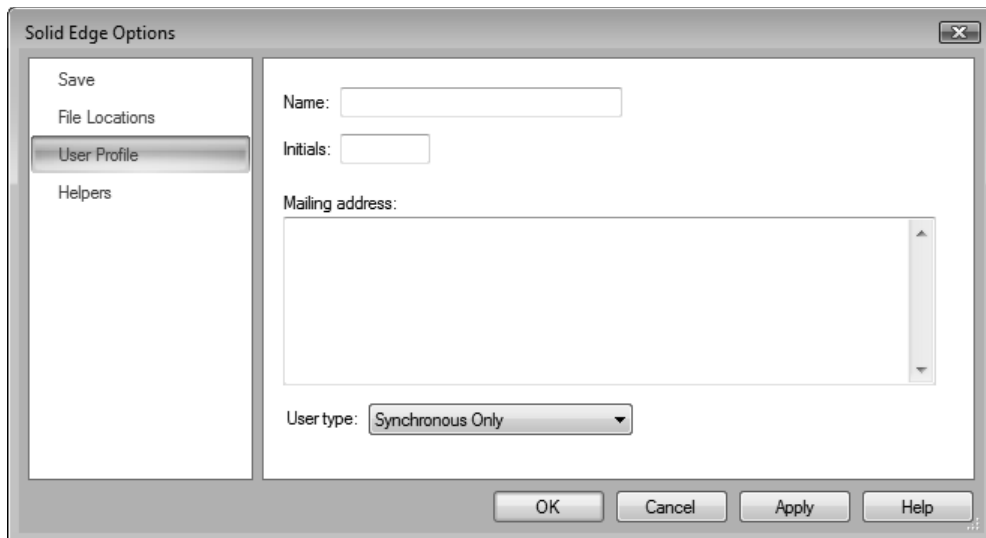


Figure 2-4 The Solid Edge Options dialog box

To start a new Part file in the default template, choose **Traditional ISO Part** from the **Create** area of the welcome screen.

Starting a New Part File by Using the New Dialog Box



You can start a new part file using the **New** dialog box. To invoke this dialog box, choose the **New** button in the **Quick Access toolbar** of the welcome screen. The **New** button is not available by default. Therefore, you need to add it to the **Quick Access toolbar**. To do so, click on the down arrow in the **Quick Access toolbar**; a flyout will be displayed. Choose the **New** option from the flyout; the **New** button will be added to the **Quick Access toolbar**. Now, choose the **New** button from the **Quick Access toolbar**; the **New** dialog box will be displayed, as shown in Figure 2-5. Alternatively, you can choose **Application Button > New** to invoke the **New** dialog box. The options in this dialog box are discussed next.

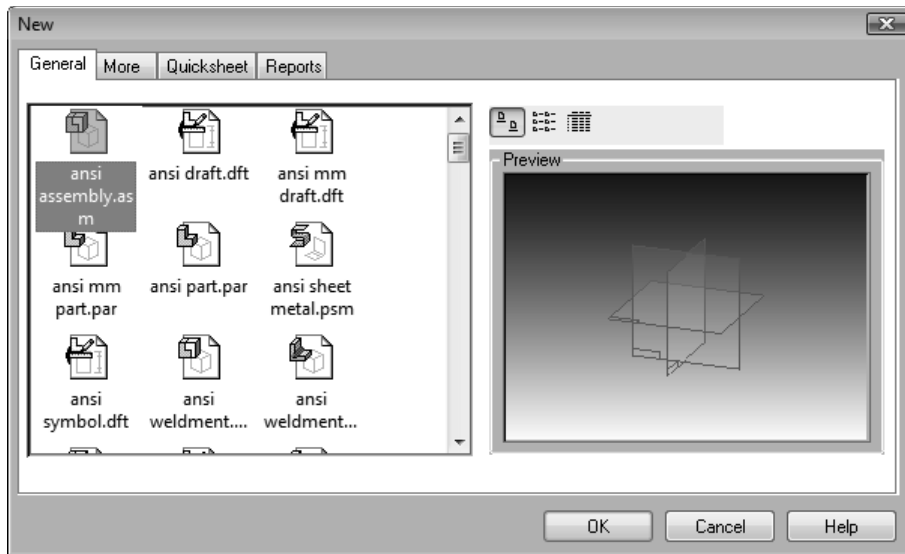


Figure 2-5 The New dialog box

General Tab

The **General** tab provides default templates for starting various environments such as **Assembly**, **Draft**, **Part**, **Sheet Metal**, and **Weldment**. Note that these templates are available in various standards such as **ansi mm**, **din**, **iso**, **jis**, **metric**, **sync din**, **sync iso**, **sync jis**, **sync uni**, **uni**, **ansi** and **sync ansi**. All these default templates are available at *C:/ Program files > Solid Edge ST2 > Template*.

To open a new document in the **Part** environment of Solid Edge ST, select the **iso part.par** template and then choose the **OK** button from the **New** dialog box. Alternatively, double-click on **iso part.par** to open a new document in the **Part** environment.

More Tab

The **More** tab provides some other predefined templates for starting files in various

environments of Solid Edge. All these predefined templates are available at *C:/ Program files > Solid Edge ST2 > Template > More*.

Quicksheet Tab

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

Reports Tab

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



Note

*You can customize the available tabs as per your requirement. To do so, create a folder named **Custom Template** in the **Program Files > Solid Edge ST2 > Template**. Now, save company-specific templates in this folder; the **Custom Template** tab will automatically be added to the **New** dialog box and the saved templates will also be listed in this tab.*

Large Icon

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

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

Detail

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 2-6. This figure also shows various components of the part document of Solid Edge.



Note

*Solid Edge provides you with an option to start this program directly in a particular environment. To do so, choose **Application Button**; a flyout will be displayed. Choose the **Solid Edge Options** button from the flyout; the **Solid Edge Options** dialog box will be displayed. In this dialog box, choose **Helpers** and then select the required environment from the drop-down list below the **Start using this template** radio button.*

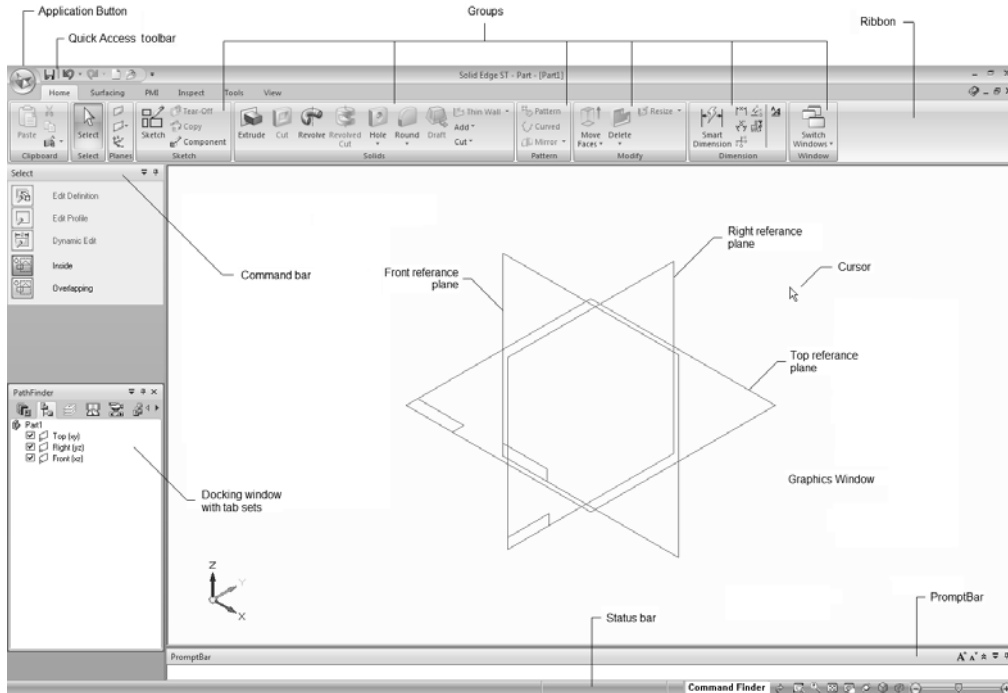


Figure 2-6 New document in the **Part** environment

INVOKING THE SKETCHING ENVIRONMENT

As mentioned earlier, whenever you start a new document in the **Part** environment of Solid Edge, three reference planes will be displayed, refer to Figure 2-6. You can invoke the sketching environment using any one of these reference planes. The sketching environment can be invoked using the **Sketch** tool. These sketches can be used multiple times to create features.



To invoke the sketching environment, choose **Home > Sketch > Sketch** from the **Ribbon**; you will be prompted to click on a planar face or a reference plane. Select a reference plane, the selected plane will be oriented parallel to the screen and the sketching environment will be invoked. Figure 2-7 shows the default screen in the sketching environment of Solid Edge.

SKETCHING TOOLS

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

Drawing Lines

Ribbon: Draw > Line



Lines are the most widely used sketched entities in any design. In Solid Edge, the **Line** tool is used to draw straight lines, as well as tangent or normal arcs originating

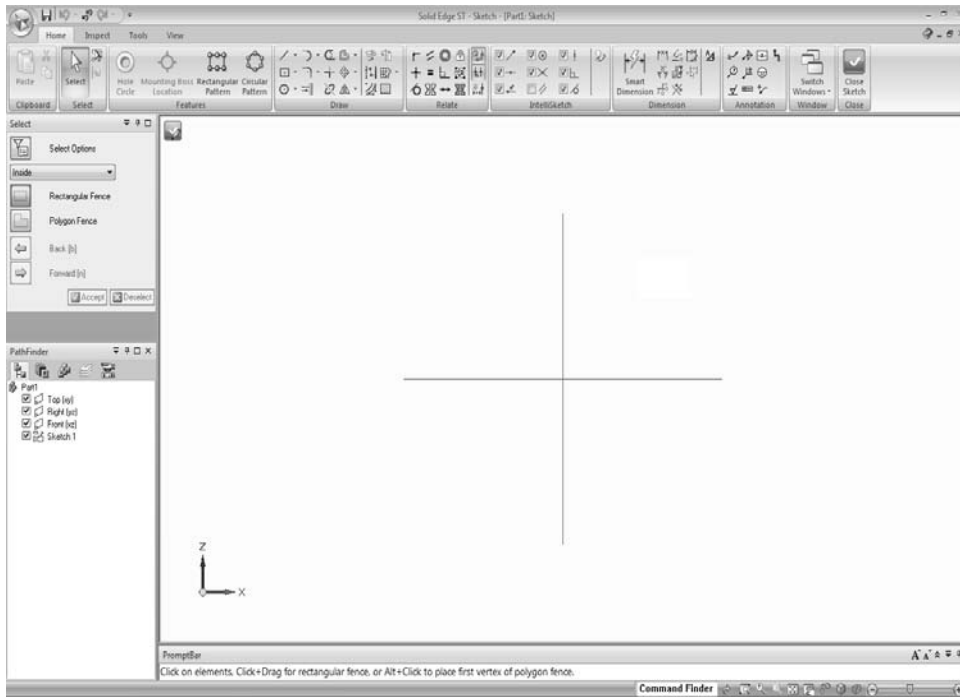
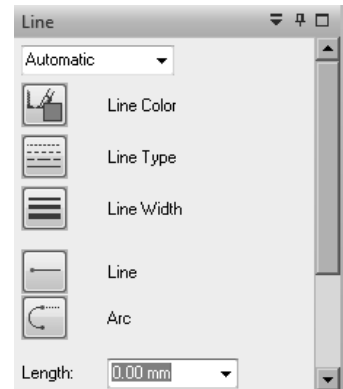


Figure 2-7 The default screen in the sketching environment of Solid Edge

from the endpoint of a selected line. The **Line** button is chosen by default in the **Sketch** environment. As a result, the properties of the line are displayed in the command bar, as shown in Figure 2-8. By default, the command bar is available on the left of the screen. The methods to create lines and arcs by using this tool are discussed next.

Drawing Straight Lines

To draw a straight line, choose the **Line** button from the **Ribbon**; you will be prompted to select the first point of the line. Specify the point (click) in the drawing window by pressing the left mouse button; a rubber-band line will be displayed between the cursor and the specified point. Also, you will be prompted to select the second point of the line. Note that on moving the cursor in the drawing window, the length and angle of the line also get modified accordingly in the **Line** command bar. Next, specify the endpoint of the line in the drawing window by pressing the left mouse button. Alternatively, you can draw a line by specifying its length and angle in the **Line** command bar.



*Figure 2-8 The **Line** command bar*

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 the relationship handle and it indicates the relationship that will be applied to the entity being drawn. In the above-mentioned case, 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 a line.



Note

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

The process of drawing lines does not end after defining 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 becomes the endpoint of the first line and the endpoint of the new line is attached to the cursor.

The process of drawing consecutive lines continues until you right-click to terminate it. However, note that even after right-clicking, the **Line** tool will not be deactivated and you will still be prompted to specify the first point of the line. You can terminate the **Line** tool by choosing **Select > Select** from the **Ribbon** or by pressing the ESC key. Figures 2-9 and 2-10 show the continuous lines being drawn.

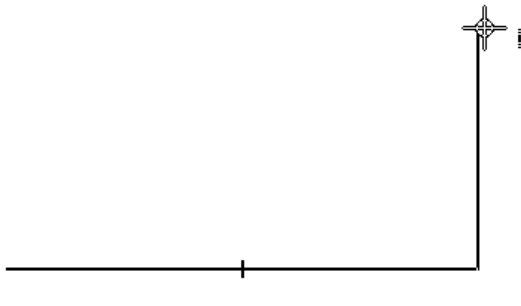


Figure 2-9 Vertical relationship handle displayed while drawing the vertical line

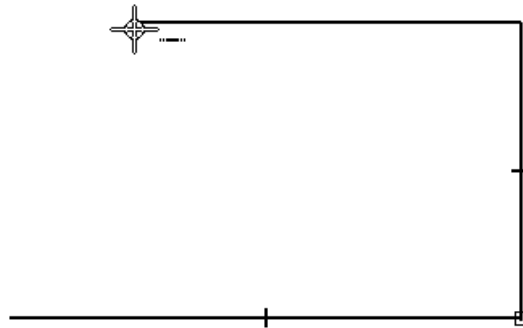


Figure 2-10 Horizontal relationship handle displayed while drawing the horizontal line

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, 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 2-11 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 will turn orange in color and the alignment indicator will be displayed.

Drawing Tangent and Normal Arcs

As mentioned earlier, you can draw a tangent or a normal arc by using the **Line** tool. To draw an arc when the **Line** tool is active, press the A key or choose the **Arc** button from the

command bar. You will notice that the **Length** and **Angle** edit boxes in the command bar are 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 will be displayed at the start point of the arc. This circle is divided into four regions. These regions are called intent zones, and are used to define the type of the arc to be created. To create an arc tangent to a line, move the cursor through a small distance in the zone that is tangent to the line; the tangent arc will be displayed. Next, click to specify the endpoint of the arc. Similarly, if you move the cursor in the zone that is normal to the line, the normal arc will be displayed. Next, click to specify the endpoint of the arc. After drawing the required arc, the system will automatically switch back to the line mode. You can activate the line mode or the arc mode by pressing the L or the A key, respectively. Figure 2-12 shows a tangent arc being drawn from within the **Line** tool.

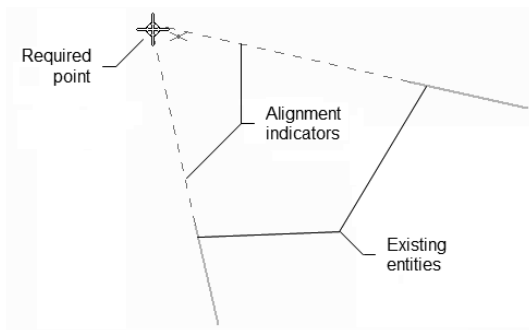


Figure 2-11 The alignment indicators originating from the endpoints of the existing lines

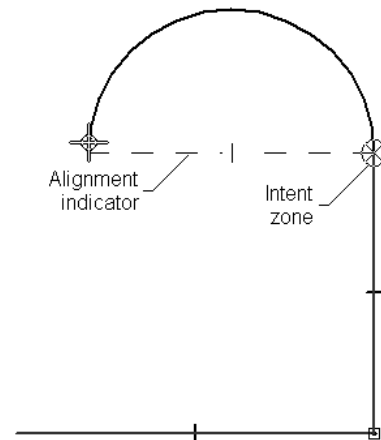


Figure 2-12 A tangent arc drawn from the **Line** tool



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.

The buttons in the **Line** command bar are used to specify the color, type, and width of lines. You can also draw a projection line of infinite length by choosing the **Projection Line** button from the **Line** command bar. Projection lines are generally used in the **Draft** environment.

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 the Radius

Ribbon: Draw > Circle by Center Point



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 a circle using this method, choose the **Circle by Center Point** button from the **Draw** group of the **Ribbon**; the **Circle by Center Point** command 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 enter the value of the diameter or radius in the command bar. Figure 2-13 shows a circle drawn using this method.

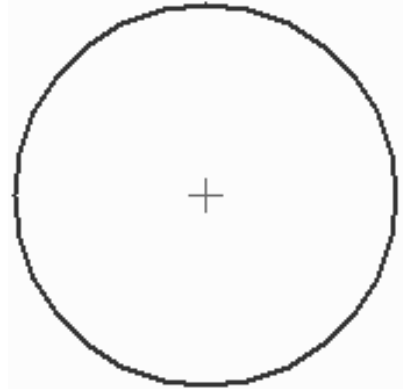


Figure 2-13 Circle drawn using the Circle by Center Point method

Drawing a Circle by Specifying Three Points

Ribbon: Draw > Circle by Center Point > Circle by 3 Points



This method is used to draw a circle by using the three points that you need to define on it. To use this method, click on the down arrow on the right of the **Circle by Center Point** button in the **Draw** group of the **Ribbon**; a flyout will be displayed. From the flyout, choose the **Circle by 3 Points** option; you will be prompted to specify the first point and then the second point on the circle. Specify these two points; small reference circles will be displayed on these two points, as shown in Figure 2-14. Also, you will be prompted to specify the third point. Specify the third point on the circle to create it.

Drawing a Tangent Circle

Ribbon: Draw > Circle by Center Point > Tangent Circle



This method is used to draw a circle that is tangent to one or two existing entities. To draw a circle using this method, choose the **Tangent Circle** button from the **Circle by Center Point** flyout in the **Draw** group of the **Ribbon**; you will be prompted to specify the first point on the circle. The circle will be drawn using two or three points, depending upon how you specify the first point of the circle. If you specify the first point on an existing entity, then 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 entities. Also, small reference circles will be displayed at the points where the circle is tangent to the selected entities. Figure 2-15 shows a circle tangent to two lines.

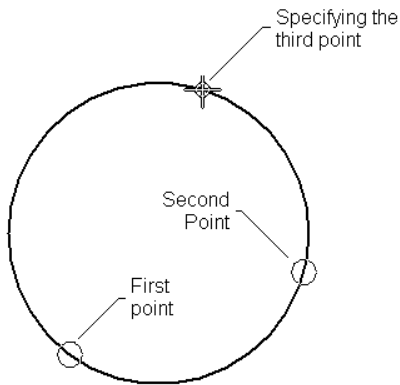


Figure 2-14 Circle drawn using the **Circle by 3 Points** method

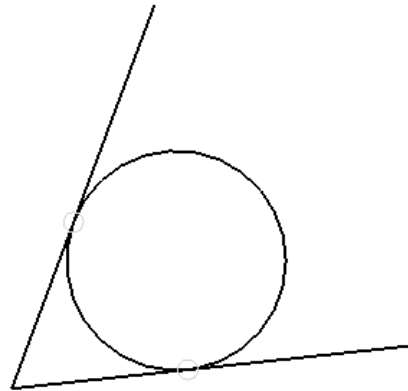


Figure 2-15 A circle drawn tangent to two lines

Drawing Ellipses

In Solid Edge, you can draw ellipses using two methods that are discussed next.

Drawing an Ellipse by Specifying Three Points

Ribbon: Draw > Circle by Center Point > 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 by using this method, choose the

Ellipse by 3 Points button from the **Circle by Center Point** flyout in the **Draw** group of the **Ribbon**; you will be prompted to specify the first and second endpoints of the primary axis of an ellipse. Specify the 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 axis or the minor axis, depending on the location of point specification. Figure 2-16 shows a profile in which the cursor is moved to define the third point on the ellipse to create it. Note that to draw an ellipse, you can also enter values in the **Ellipse** command bar, which is displayed on invoking this tool.

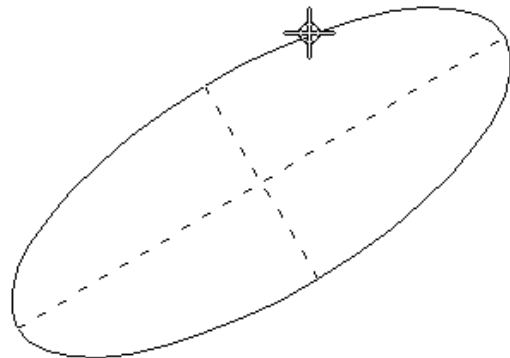


Figure 2-16 An ellipse drawn by specifying three points

Drawing a Center Point Ellipse

Ribbon: Draw > Circle by Center Point > Ellipse by Center Point



This method is used to draw a center point ellipse. In this method, first you need to define the center point of an ellipse. On doing so, you will be prompted to specify the endpoint of the primary axis. Specify the endpoint of the primary axis; you will be prompted to specify the endpoint of the secondary axis. Specify the endpoint of the secondary axis; the ellipse will be created. You can also draw an ellipse by entering the required dimensions in the command bar and then specifying its center point.

Placing Sketched Points

Ribbon: Draw > Line > Point



Points are generally used as references for drawing other sketched entities. To place a point, choose the **Point** button from the **Line** flyout in the **Draw** group of the **Ribbon**; you will be prompted to click for the point. Next, place the point by defining its location in the drawing window or by entering its coordinates in the **Point** command bar.

Drawing Arcs

In Solid Edge, you can draw arcs using three methods that are discussed next.

Drawing a Tangent or Normal Arc

Ribbon: Draw > Tangent Arc



This method of drawing arcs is similar to drawing tangent and normal arcs using 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 the included angle of the arc in the **Tangent Arc** command bar, which is displayed when you invoke this tool.

Drawing a Three Points Arc

Ribbon: Draw > Tangent Arc > Arc by 3 Points



The **Arc by 3 Points** tool is used to draw an arc by specifying its start point, endpoint, and the third point on its periphery. The third point is used to specify the direction in which the arc will be drawn. You can specify the radius of this arc in the command bar. On specifying the radius, you need to specify only the start point and the endpoint of the arc. Figure 2-17 shows a three-points arc drawn.

Drawing a Center Point Arc

Ribbon: Draw > Tangent Arc > Arc by Center Point



This method is used to draw an arc by specifying its center point, start point, and endpoint. Invoke the **Arc by Center Point** tool; you will be prompted to specify the center point of the arc. Specify the center point of the arc; you will be prompted to specify its start point and endpoint. Note that when you specify the start point of the arc, the radius will be automatically defined. And on specifying the endpoint of the arc, the length of the arc will be defined. Figure 2-18 shows an arc drawn by using this method.

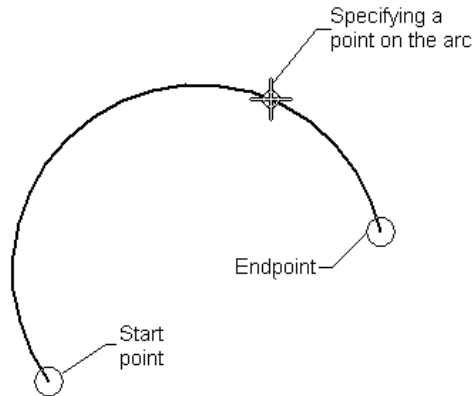


Figure 2-17 An arc drawn by using the **Arc by 3 Points** method

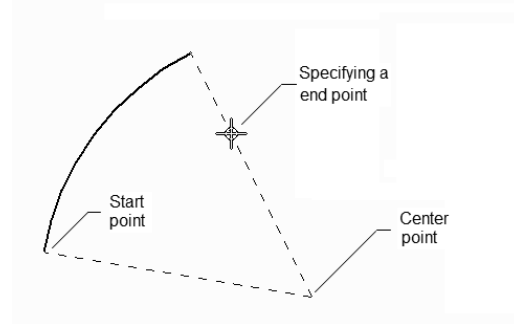


Figure 2-18 An arc drawn by using the **Arc by Center Point** method

Drawing a Rectangle by Specifying its Center

Ribbon: Draw > Rectangle by Center



In Solid Edge ST2, you can draw a rectangle by specifying its center point and any one of its vertices. To draw a rectangle by using this method, choose the **Rectangle by Center** tool from the **Draw** group of the **Ribbon**; you will be prompted to specify the center point of the rectangle. Click in the drawing window to specify the center point of the rectangle and move the cursor; the dynamic preview of the rectangle will be displayed in the graphics window, as shown in Figure 2-19 and you will be prompted to click to create the rectangle. Specify a point, the point you specify will define the height and width of the rectangle. Alternatively, you can specify the width, height, and angle of the rectangle in the **Rectangle** command bar.

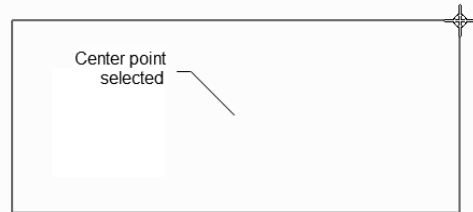


Figure 2-19 Dynamic preview of the rectangle displayed on specifying its center



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 a Rectangle by Specifying Two Points

Ribbon: Draw > Rectangle by Center> Rectangle by 2 Points



You can also draw a rectangle by specifying two diagonally opposite corners. To draw a rectangle by using this method, choose the **Rectangle by 2 Points** tool from the **Rectangle by Center** flyout in the **Draw** group of the **Ribbon**; you will be prompted to specify the first corner of the rectangle. Click in the drawing window to specify it; the dynamic preview of the rectangle will be displayed in the graphics window, as shown in Figure 2-20, and you will be prompted to click in the drawing window to specify the second corner. Click in the drawing window to specify the diagonally opposite corner; a rectangle will be created.

Drawing a Rectangle by Specifying Three Points

Ribbon: Draw > Rectangle by Center> Rectangle by 3 Points



You can also draw rectangles by specifying three points. The first two points define the width and orientation of the rectangle and the third point defines its height. Invoke the **Rectangle by 3 Points** tool; you will be prompted to specify the first corner. Specify a point in the drawing window to define the start point of the rectangle; 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. On doing so, the rectangle will be drawn at an angle. After specifying the width of the rectangle, you will be prompted to specify a point that will define the height of the rectangle. Specify the point; the rectangle will be created. Alternatively, you can specify the width, height, and angle of the rectangle in the **Rectangle** command bar. Figure 2-21 shows a rectangle drawn at an angle.

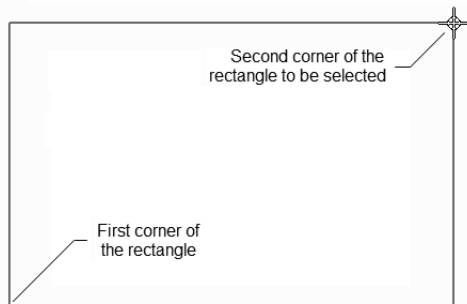


Figure 2-20 Drawing a rectangle by specifying two diagonally opposite corners

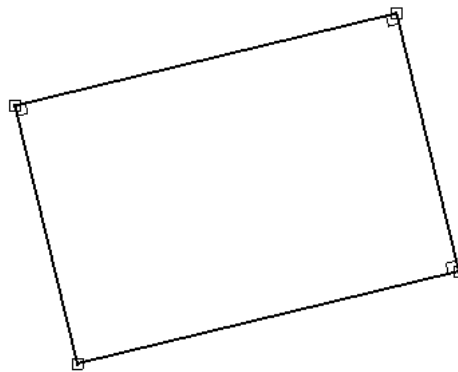


Figure 2-21 Rectangle drawn at an angle

Drawing Polygons

Ribbon: Draw > Rectangle by Center> Polygon by Center



The polygons drawn in Solid Edge are regular polygons. A regular polygon is a geometric figure with many sides, and in it the length of all sides and the angle between them are the same. In Solid Edge, you can draw a polygon with the number of sides

ranging from 3 to 200. To create a polygon, choose **Draw > Rectangle by Center > Polygon by Center** from the **Draw** group of the **Ribbon**; you will be prompted to click to specify the center point of the polygon. Click in the graphics window to specify the center point of the polygon; the dynamic preview of the polygon will be displayed. In the preview, you will notice that an imaginary circle is drawn such that all its vertices touch the circle. This imaginary circle will be used as the construction geometry for creating the polygon.

Note that if the **By Midpoint** button is chosen in the **Polygon by Center** command bar, you can specify the midpoint of an edge of the polygon, as shown in Figure 2-22. If you choose the **By Vertex** button from the **Polygon by Center** command bar, you can specify the vertex of the polygon, as shown in Figure 2-23.

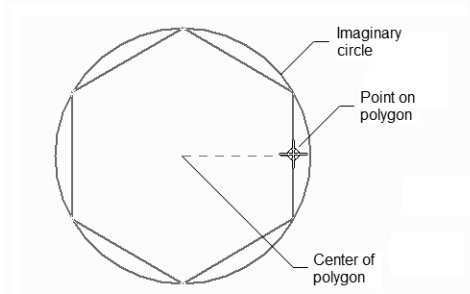


Figure 2-22 Drawing a polygon by using the **By Midpoint** button

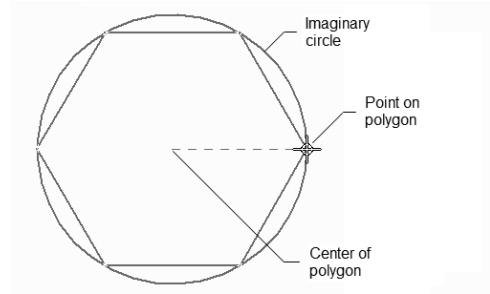


Figure 2-23 Drawing a polygon by using the **By Vertex** button

Next, in the **Polygon by Center** command bar, specify the number of sides of the polygon in the **Sides** spinner. In the **Distance** spinner, specify the distance between the two specified points of the polygon. In the **Angle** spinner, specify the orientation of the polygon with respect to the horizontal axis.

Drawing Curves

Ribbon: Draw > Line > 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 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 approximately the same shape as the reference curve. Figure 2-24 shows a curve drawn by using this method.

Drawing a Curve by Specifying Points in the Drawing Window

In this method, you need to continuously specify points in the drawing area to draw the curve passing through them. 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 specifying points until

you have specified all the points required for drawing the curve. Figure 2-25 shows a curve drawn by using this method.



Figure 2-24 Curve drawn by dragging the cursor in the drawing window

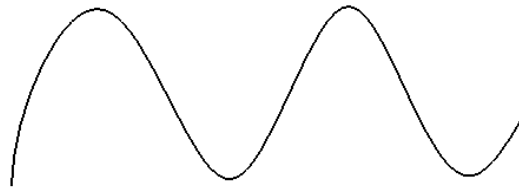


Figure 2-25 Curve drawn by specifying points in the drawing window

Converting Sketched Entities into Curves

Ribbon: Draw > Convert to Curve



In Solid Edge, you can convert the sketched entities such as lines, arcs, circles, and ellipses into bezier spline curves by using the **Convert to Curve** tool. On invoking this tool, you will be prompted to select an element to be converted into a curve. As soon as you select the element, it will be 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 has increased and the control polygon is displayed on that entity. If you drag the converted entity using any of its handles, it will become a curve.

Filleting Sketched Entities

Ribbon: Draw > Fillet



Filleting is a process of rounding sharp corners of a profile. You can create a fillet by removing sharp corner and then replacing it with round corners. In Solid Edge, you can create a fillet between any two sketched entities. On invoking the **Fillet** tool, the **Fillet** command bar will be displayed. Enter the radius of the fillet in the **Radius** edit box of the command bar and press ENTER. Now, 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 corners will be highlighted in orange when you move the cursor over the corner. Select the corners at this stage to create the fillet. Figure 2-26 shows a profile before and after filleting.

You can retain the sharp corner even after creating the fillet. To do so, choose the **No Trim** button from the **Fillet** command bar and then select the corner to be filleted; the fillet will be

created and the sharp corner will be retained. Figure 2-27 shows a profile in which the fillet is created with the sharp corner retained.

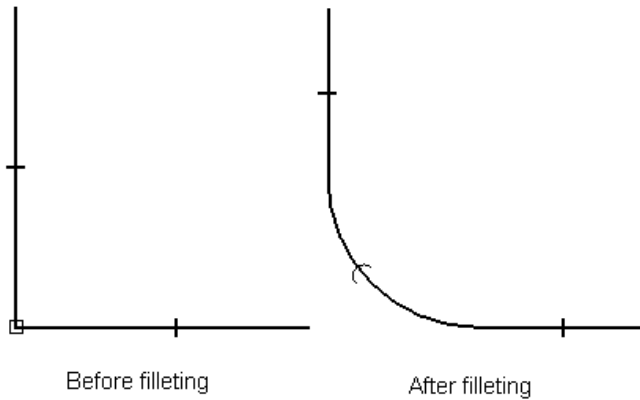


Figure 2-26 Sketch before and after creating the fillet

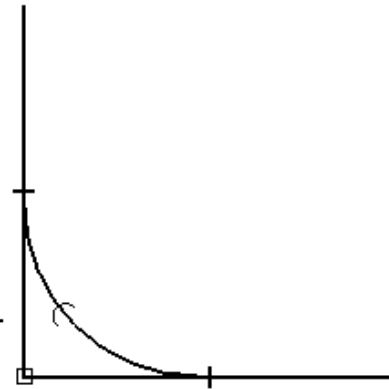


Figure 2-27 Sharp corner retained after creating the fillet



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.

Chamfering Sketched Entities

Ribbon: Draw > Fillet > Chamfer



Chamfering is a process of beveling the sharp corners of a profile to reduce the stress concentration. You can create a chamfer only between two linear entities. A 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 a chamfer, invoke the **Chamfer** tool; the **Chamfer** command bar will be displayed. In this command bar, the **Angle**, **Setback A**, and **Setback B** spinners are available. The **Setback A** and **Setback B** values define the chamfer distance along the first and second edges, respectively and the **Angle** spinner defines the inclination angle of the chamfer. Note that you can specify any two of the three values. The third value is automatically updated on the basis of the two values that you defined.

After setting any two values in the **Chamfer** command bar, select the first line and the second line to be chamfered; the preview of the resulting chamfer will be displayed. Next, click to create the chamfer. Note that the first line is taken as the setback A element and the second line



Tip. In Solid Edge, you can create fillets or chamfers by 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. The angle and distances of the chamfer will depend on how far you dragged the mouse from the corner.

is taken as the setback B element, by default. If you want to reverse the order, move the cursor over the first line. You will notice that now the second line is taken as the setback A element, and the first line is taken as the setback B element. Consequently, the preview will also change automatically. By default, the setbacks A and B are displayed in orange. Figure 2-28 shows the preview of the chamfer.

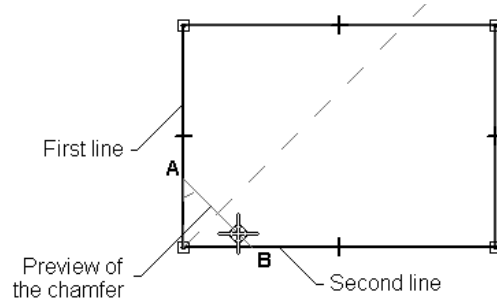


Figure 2-28 Preview of the chamfer

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 on to an Area

Ribbon: View > Orient > Zoom Area
Status Bar: Zoom Area



The **Zoom Area** tool allows you to zoom on to a particular area by defining a box around it. You can invoke this tool from the **Orient** group in the **View** tab of the **Ribbon** or from the Status bar. On invoking this tool, a plus sign (+) of infinite length will be attached to the tip of the cursor and you will be prompted to click to define the first corner or drag to specify 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 area defined inside the box will be zoomed and displayed on the screen.



Note

*If the triad is not visible in the sketching environment of Solid Edge, choose the **Application Button**; a flyout will be displayed. Choose the **Solid Edge Options** button from the flyout; the **Solid Edge Options** dialog box will be displayed. Choose the **View** button from the left pane of this dialog box and then select the **Show orientation triad** check box. Next, choose the **Apply** button; the triad will be displayed in the Graphics window.*

Dynamically Zooming a Drawing

Ribbon: View > Orient > Zoom
Status Bar: 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 into the drawing, press and hold the left mouse button in the center of the screen and then drag the cursor down. To zoom out of it, 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. The drawing display area will be increased such that the point at which you clicked will be moved to the center of the screen. Alternatively, you can zoom in and zoom out of the drawing area by scrolling the mouse, when no other tool is active.

Fitting all Entities into the Current Display

Ribbon: View > Orient > Fit
Status Bar: 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

Ribbon: View > Orient > Pan
Status Bar: Pan



The **Pan** tool allows you to dynamically pan the drawing 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 the cursor for the dynamic pan. Press and hold the left mouse button in the drawing window, and then drag the cursor 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 will be fixed at the point you specified and the other end will be attached to the hand cursor. Move the cursor and specify another point in the drawing window to pan the drawing. Alternatively, you can press the SHIFT key and drag the mouse by pressing the middle mouse button for panning.

Restoring the Original Orientation of the Sketching Plane

Ribbon: View > Views > Sketch View
Status Bar: Sketch View



Sometimes while using the drawing display tools, the orientation of the sketching plane may change. 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.



Tip. You can also use the keyboard to modify the drawing display area. To do so, the following combination of keys can be used:

CTRL+ Top/Left arrow key = Zoom in
 CTRL+ Bottom/Right arrow key = Zoom out
 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

SELECTING SKETCHED ENTITIES

You can select the sketched entities available in the drawing window by invoking the **Select** mode. To do so, choose **Home > Select > Select** from the **Ribbon**; the **Select** mode will be invoked. Note that you can exit from any active tool and activate the **Select** mode by pressing the ESC key. You can then click on the required entities to select them. You can also select all entities in the graphics window by pressing the CTRL+A keys or by choosing **Select All (CTRL + A)** from the **Select** group of the **Ribbon**. The selected entities will be highlighted in green.

By default, the **Rectangular Fence** button is chosen in the **Select** command bar. As a result, you can select entities by dragging a rectangular box around them. If you choose the **Polygon Fence** button from the **Select** command bar, you can select the entities by drawing a polygonal boundary (fence) around them. You can draw a polygon fence by specifying multiple points in the graphics window. The entities to be selected depend on the option selected from the **Fence Filter** drop-down list in the **Select** command bar. The options in this drop-down list are discussed next.

Inside

This is the first option of the **Fence Filter** drop-down list. This option ensures that only those entities are selected that lie completely inside the boundary that is created by using the **Rectangular Fence** or the **Polygon Fence** button.

Outside

This selection option ensures that only those entities are selected that lie completely outside the boundary that is created by using the **Rectangular Fence** or the **Polygon Fence** button.

Overlapping

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

Inside and Overlapping

This option is a combination of the **Inside** and the **Overlapping** options. This selection option ensures that all entities that lie partially or fully inside the boundary are selected.

Outside and Overlapping

This option is a combination of the **Outside** and the **Overlapping** options. This selection option ensures that all entities that are partially inside the boundary or completely outside the boundary are selected.

DELETING SKETCHED ENTITIES

To delete the sketched entities, select them using any one of the object selection methods discussed above. The selected entities turn green in color. Next, press the DELETE key; all the selected entities will be deleted.

TUTORIALS

As mentioned in Chapter 1, 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 command bar to draw the profile with exact dimensions. This will help you in improving your sketching skills.

Tutorial 1

In this tutorial, you will draw the profile of the model shown in Figure 2-29. The profile to be drawn is shown in Figure 2-30. Do not dimension the profile because the dimensions are given for your reference only. **(Expected time: 30 min)**

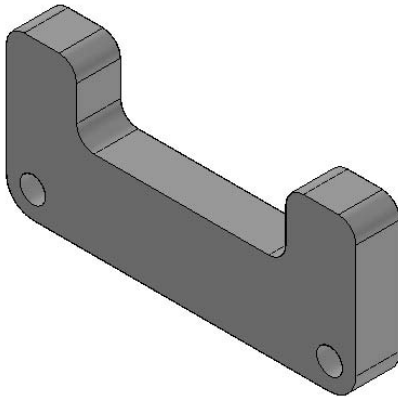


Figure 2-29 Model for Tutorial 1

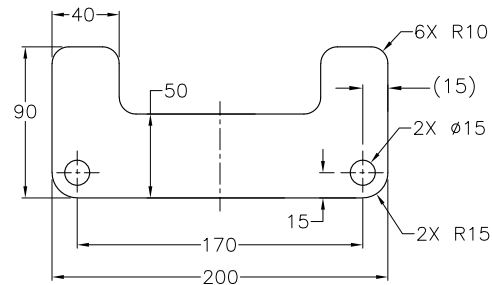


Figure 2-30 Profile for Tutorial 1

The following steps are required to complete this tutorial:



- Start Solid Edge and then start a new file in the **Part** environment.
- Invoke the sketching environment.
- Draw the outer loop of the profile by using the **Line** tool.
- Fillet the sharp corners of the outer loop by using the **Fillet** tool.
- Draw the circles by using the centers of the fillets to complete the profile.
- Save the file and close it.

Starting Solid Edge and Selecting the Sketching Plane

You need to create the profile of the model in the sketching environment of Solid Edge. Therefore, you need to start a new part file first.

- Choose the **Start** button available at the lower left corner of the screen to display a menu.
- Choose **All Programs** (or **Programs**) > **Solid Edge ST2** > **Solid Edge** from the start menu to start the Solid Edge session.

As Solid Edge gets started, the welcome screen is displayed. Now, you need to start a new part file to draw the sketch of the given model.

3. Click the link **Traditional ISO Part** in the **Create** area of the welcome screen; a new Solid Edge part file is started. If the **Traditional ISO Part** link is not available, then you need to do some settings, as discussed next. Else, you can skip the steps 4, 5 and 6.
4. Choose the **Application Button** from the top left corner of the screen; a flyout is displayed. Choose the **Solid Edge Options** button from the flyout; the **Solid Edge Options** dialog box is displayed. 
5. Choose the **User Profile** option from the left pane of the dialog box.
6. Select the **Traditional and Synchronous** option from the **User type** drop-down list and then choose **OK** from the **Solid Edge Options** dialog box.
7. Choose **Home > Sketch > Sketch** from the **Ribbon**; the **Sketch** command bar is displayed and you are prompted to select a planar face or a reference plane. 
8. Select the front plane from the graphics window or select the **Front (XZ)** node from the docking window; the sketching environment is invoked and the selected plane orients itself parallel to the screen. Also, the **Line** tool is invoked by default. If the **Line** tool is not invoked, choose the **Line** button from the **Draw** group in the **Home** tab of the **Ribbon** to invoke it.

Drawing Lines for the Outer Loop

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

As the **Line** tool is active by default, the **Line** command bar is displayed on the left of the window and you are prompted to specify the start point of the line. The prompts are displayed in the **Prompt bar** at the bottom of the screen. You can start drawing the line from the origin, which is the point where the top, right, and front planes intersect and hence its coordinates are 0,0,0. In the current view, the origin is the intersection of the two planes displayed as the horizontal and vertical lines.


1. Make sure that the **Line** tool is active and then move the cursor close to the origin. One of the two planes, displayed as grey horizontal or vertical lines, is highlighted and the **Midpoint** relationship handle is displayed on the right of the cursor.
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 get dynamically modified in the command bar.

Next, you need to specify the other points to define the first line and the remaining lines by using the **Length** and **Angle** edit boxes in the **Line** command bar.

3. Enter **200** in the **Length** edit box of the **Line** command bar and press ENTER. Next, enter **0** 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 by using the **Fit** tool.

4. Choose the **Fit** button from the Status bar; 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. Enter **90** in the **Length** edit box and press ENTER. Again, enter **90** in the **Angle** edit box and press ENTER; a vertical line of 90 mm is drawn. You can use the **Fit** tool to modify the current drawing display area as discussed above.
6. Enter **40** in the **Length** edit box and press ENTER. Enter **180** in the **Angle** edit box and press ENTER; a horizontal line of 40 mm is drawn toward the left of the last line.
7. Enter **40** in the **Length** edit box and press ENTER. Enter **-90** in the **Angle** edit box and press ENTER; a vertical line of 40 mm is drawn downward.
8. Enter **120** in the **Length** edit box and press ENTER. Enter **180** in the **Angle** edit box and press ENTER; a horizontal line of 120 mm long is drawn.
9. Move the cursor vertically upward; a rubber-band line is displayed with its start point at the endpoint of the previous line and the endpoint attached to the cursor. When the line becomes vertical, the vertical relationship handle is displayed.
10. Move the cursor vertically upward until the horizontal alignment indicator is displayed from the top endpoint of the vertical line of 40 mm long. Note that at this point, the value in the **Length** edit box is 40 and the value in the **Angle** edit box is 90. Now, click to specify the endpoint of this line.
11. Move the cursor horizontally toward the left above the vertical plane, and then make sure that the horizontal relationship handle is displayed. Next, move the cursor once on the vertical plane and then move it vertically upward. Click when the intersection relationship handle is displayed.
12. Move the cursor vertically downward to the origin. If the first line is not highlighted in orange, 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.

13. Click to specify the endpoint of the line when the endpoint relationship handle is displayed. Choose the **Fit** button to fit the sketch into the drawing window.
14. Choose the **Select** button from the **Select** group in the **Home** tab of the **Ribbon** to exit the **Line** tool. The sketch after drawing the lines is shown in Figure 2-31.



Filletting Sharp Corners

Next, you need to fillet the sharp corners so that sharp edges are not in the final model. You can fillet the corners by using the **Fillet** tool.

1. Invoke the **Fillet** tool from the **Draw** group of the **Ribbon**; the **Fillet** command bar is displayed.



To fillet any sharp corner, first you need to specify the fillet radius. You can fillet the lower left and lower right corners first and then the remaining corners. This is because the fillet radii of the bottom left and bottom right corners are equal and the fillet radii of the remaining corners are also equal.

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

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

5. Enter **10** in the **Radius** edit box in the command 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 2-32.

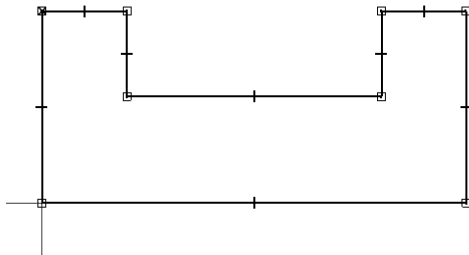


Figure 2-31 Sketch after drawing lines

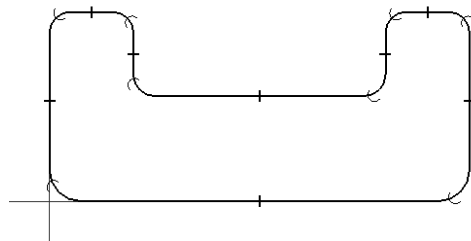



Figure 2-32 Sketch after creating fillets

Drawing Circles

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

1. Choose the **Circle by Center Point** button from the **Draw** group of the **Ribbon**; the **Circle by Center Point** command bar is displayed and you are prompted to select the center point of the circle. 
2. Enter **15** in the **Diameter** edit box of the **Circle** command bar and press ENTER; a circle of the specified diameter is attached to the cursor. The circle attached to the cursor moves as you move the cursor on the screen.
3. Move the cursor over the fillet at the lower left corner once; the fillet is highlighted in orange 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 a plus sign; the fillet is highlighted in orange 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 prompted again to specify the center point of the circle.
6. Move the cursor over the lower right fillet so that its center point is also displayed.
7. Move the cursor over the center point of the lower right fillet and click when the concentric relationship handle is displayed. The final profile for Tutorial 1 is shown in Figure 2-33.

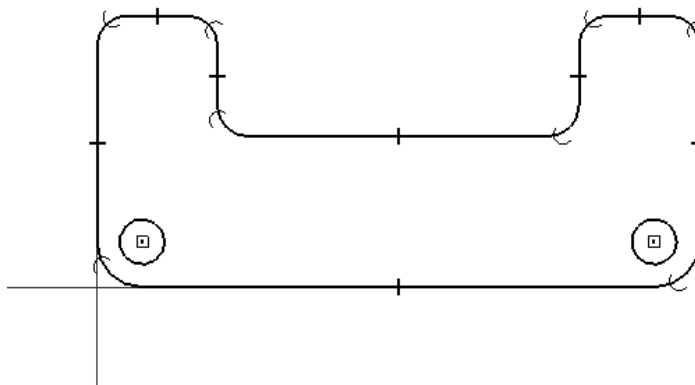
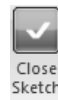


Figure 2-33 Final profile for Tutorial 1

8. Press the ESC key to exit the **Circle** tool.

9. Choose **Close > Close Sketch** from the **Ribbon** to close the sketching environment; the **Sketch** command bar is displayed. Also, the current view changes automatically to the isometric view. Choose the **Fit** button to fit the sketch into the drawing window.
10. Enter the name of the sketch as **Base Sketch** in the **Name** edit box of the **Sketch** command bar and choose the **Finish** button from the **Sketch** command bar; the sketch is displayed with the specified name in the docking window.



Saving the File

1. Choose the **Save** button from the **Quick Access toolbar**; the **Save As** dialog box is displayed.

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

2. Browse to the *C:* drive and then create a folder with the name *Solid Edge* in it. Make the *Solid Edge* folder current and then create a folder with the name *c02* in this folder.
3. Make the *c02* folder current and save the file with the name *c02tut1.par*. The location of this file is given below:

C:\Solid Edge\c02

4. To close the file, choose the **Application Button**; a flyout is displayed. Now, choose **Close** from the flyout.

Tutorial 2

In this tutorial, you will draw the profile of the model shown in Figure 2-34. The profile to be drawn is shown in Figure 2-35. Do not dimension the profile because the dimensions are given only for your reference.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

- a. Start a new part file.
- b. Invoke the sketching environment.
- c. Draw the profile of the model by using the **Line** tool.
- d. 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 **Quick Access toolbar**, which is available on the screen after you close all files.

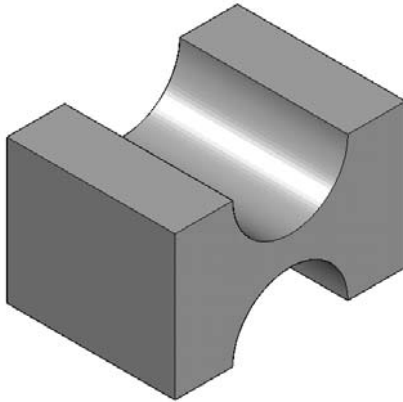


Figure 2-34 Model for Tutorial 2

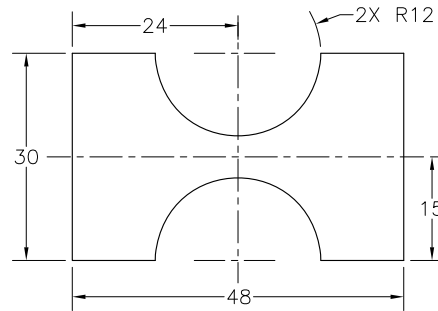


Figure 2-35 Profile for Tutorial 2

1. Choose the **New** button from the **Quick Access toolbar**; the **New** dialog box is displayed, as shown in Figure 2-36.
2. Select **iso part.par** from the **General** tab of the dialog box, as shown in Figure 2-36, and then choose **OK** to start a new part file.

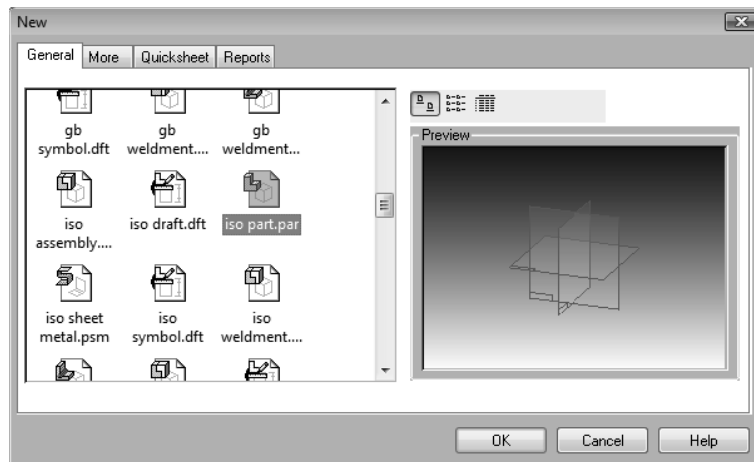


Figure 2-36 The New dialog box to start a new file in Solid Edge

3. Next, choose **Sketch > Sketch** from the **Ribbon**; the **Sketch** command bar is displayed and you are prompted to select a planar face or a reference plane.
4. Select the right 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 invoked by default. If the **Line** tool is not invoked, choose the **Line** button from the **Draw** group in the **Home** tab of the **Ribbon** to activate it.



Drawing the Profile

As the **Line** tool is active, its command bar is displayed at the left of the window 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 grey 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 that you specified 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 varies in the command bar dynamically.

3. Enter **12** in the **Length** edit box of the **Line** command bar and press ENTER. Enter **0** 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 as the next entity is an arc, you need to invoke the arc mode.

4. Press the A key to invoke the arc mode. Alternatively, you can also invoke the arc mode by choosing the **Arc** button from the command bar.

On doing so, 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** and **180** in the **Radius** and **Sweep** edit boxes of the command bar, respectively. 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 command bar.

7. Move the cursor close to the horizontal plane and click when the plane is highlighted in orange; the arc is drawn and the line mode is invoked again.

8. Enter **12** and **0** in the **Length** and **Angle** edit boxes, respectively, and then press ENTER. Choose the **Fit** button from the status bar to fit the sketch into the drawing window.

9. Enter **30** and **90** in the **Length** and **Angle** edit boxes, respectively, and then press ENTER.

10. Move the cursor horizontally toward the left. 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 endpoint of the arc. If the alignment indicator is not displayed, move the cursor once on the endpoint of the arc and then move it back.

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

11. Press the A key 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. A horizontal dotted line is displayed originating from the upper right horizontal line. At the point where the cursor is vertically in line with the start point of the lower arc, a vertical dotted line appears from the start point of the lower arc, as shown in Figure 2-37.
14. Click to define the endpoint of the arc when the horizontal and vertical alignment dotted lines 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 orange.
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 2-38.
18. Press the ESC key button to exit the current tool.

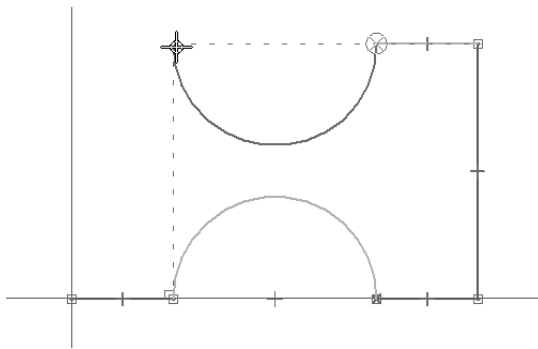


Figure 2-37 Horizontal and vertical dotted lines displayed to define the endpoint of the arc

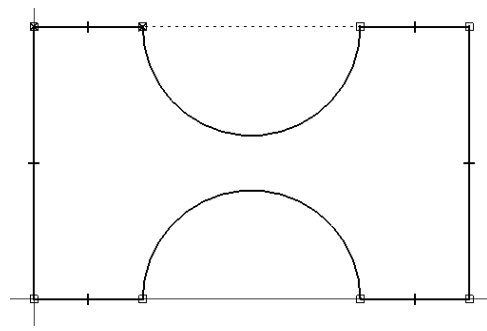



Figure 2-38 Final profile for Tutorial 2

19. Choose **Close > Close Sketch** from the **Ribbon** to close the sketching environment. You will notice that the current view is automatically changed to the isometric view. 
20. Enter **Base Sketch** as the name of the sketch in the **Name** edit box in the command bar and then choose the **Finish** button from the command bar. The sketch is displayed with this name in the **PathFinder** of the docking window.

Saving the File

1. Choose the **Save** button from the **Quick Access toolbar**; the **Save As** dialog box is displayed.
2. Browse to the *C:\Solid Edge\c02* folder, if it is not the current folder. Save the file with the name *c02tut2.par*. The location of this file is as follows:

C:\Solid Edge\c02

3. Choose the **Application Button**; a flyout is displayed. From the flyout, choose **Close** to close the file.

Tutorial 3

In this tutorial, you will draw the profile of the base feature of the model shown in Figure 2-39. The profile to be drawn is shown in Figure 2-40. Do not dimension the profile because the dimensions are given for your reference only. **(Expected time: 30 min)**

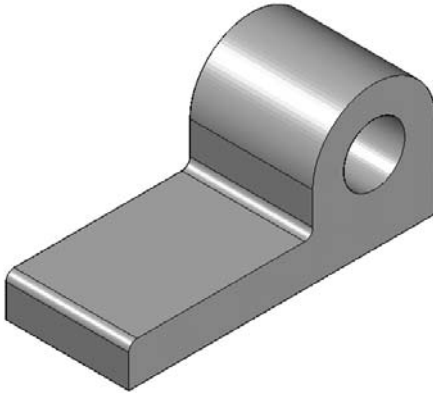


Figure 2-39 Model for Tutorial 3

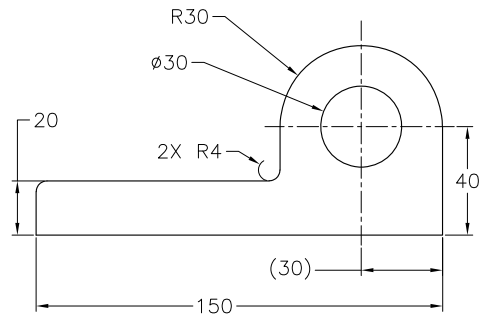


Figure 2-40 Profile for Tutorial 3


The following steps are required to complete this tutorial:

- a. Start a new part file.
- b. Invoke the sketching environment.
- c. Draw the profile of the model by using the **Line** tool.

- d. Fillet the two corners of the outer loop and then draw the inner circle.
- e. Save the file and close it.

Starting a New Part File and Selecting the Sketching Plane

As mentioned earlier, you need to start a new part file by choosing the **New** button from the **Quick Access toolbar**, which remains on the screen after closing all files.

1. Choose the **New** button from the **Quick Access toolbar**; the **New** dialog box is displayed.
2. Select **iso part.par** from the list box of the **General** tab and choose **OK** to start a new part file.
3. Choose **Sketch > Sketch** from the **Ribbon**; the **Sketch** command bar is displayed and you are prompted to select a planar face or a reference plane. 
4. Select the right 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


As the **Line** tool is active, its command bar is displayed at the left of the graphics window 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 grey 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 that you specified is selected as the start point of the line and the endpoint is attached to the cursor.

3. Enter **150** in the **Length** edit box of the **Line** command bar and press ENTER. Enter **0** in the **Angle** edit box and press ENTER.

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

4. Choose the **Fit** button from the status bar; now the line is completely visible in the current view. 
5. Enter **40** and **90** in the **Length** and **Angle** edit boxes of the command bar, respectively, and then press ENTER.

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

6. Press the A key 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** and **180** 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** and **-90** in the **Length** and **Angle** edit boxes, respectively, and then press ENTER.
10. Move the cursor horizontally toward the left and make sure that the horizontal relationship handle is displayed. Click to define the endpoint of the line when the vertical plane is highlighted in orange.
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 orange 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 2-41.

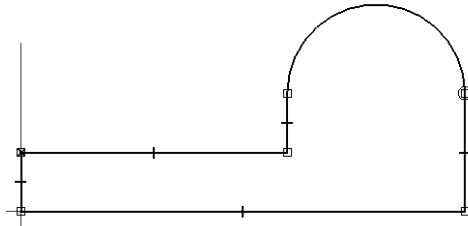


Figure 2-41 Outer loop of the profile for Tutorial 3

Filleting Sharp Corners


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

1. Choose **Draw > Fillet** from the **Ribbon**; the **Fillet** command bar is displayed.
2. Enter **4** in the **Radius** edit box of the **Fillet** command 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 orange.
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 orange.



Drawing the Circle

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

1. Choose **Draw > Circle by Center Point** from the **Ribbon**; the **Circle by Center Point** command bar is displayed. 
2. Enter **30** in the **Diameter** edit box in this command bar and press ENTER; a circle of 30 mm is attached to the cursor.
3. Move the cursor over the arc of 30 radius; the arc is highlighted in orange 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 2-42.

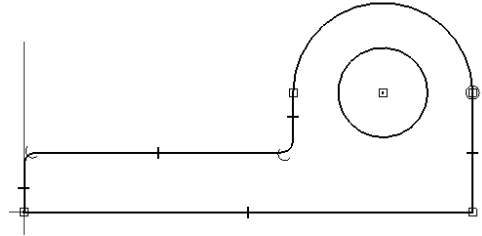



Figure 2-42 Final profile for Tutorial 3

5. Press the ESC key to exit the current tool.
6. Choose **Close > Close Sketch** from the **Ribbon**; the sketching environment is closed and the **Sketch** command bar is displayed. Also, the current view is automatically changed to the isometric view. 
7. Enter **Base Sketch** as the name of the sketch in the **Name** edit box of the command bar and choose **Finish**; the sketch is displayed by this name in the docking window.

Saving the File

1. Choose the **Save** button from the **Quick Access toolbar**; the **Save As** dialog box is displayed.
2. Browse to the *C:\Solid Edge\c02* folder, if it is not the current. Save the file with the name *c02tut3.par*. The location of this file as follows:

C:\Solid Edge\c02
3. Choose the **Application Button**; a flyout is displayed. Now, choose **Close** from the flyout to close the file.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. Most of the designs created in a solid modeling consist of 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 by using the **Sketch** tool. (T/F)
3. You can use the command bar 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 by using the _____ tool in the status bar.
6. You can invoke the arc mode within the **Line** tool by pressing the _____ key.
7. You can bevel corners in a sketch by using the _____ tool.
8. You can retain sharp corners even after filleting them by choosing the _____ button from the **Fillet** command bar.
9. In Solid Edge, you can draw a polygon of sides ranging from _____ to _____.
10. The _____ tool is used to draw an arc by specifying its start point, endpoint, and the third point on it.

Review Questions

Answer the following questions:

1. Which of the following options is selected from the **New** dialog box to start a new part file?

(a) iso assembly.asm	(b) iso draft.dft
(c) iso part.par	(d) iso sheet metal.psm
2. Which one of the following tools is used to round the sharp corners in a sketch?

(a) Fillet	(b) Chamfer
(c) Round	(d) None of these
3. Which of the following edit boxes in the arc mode replaces the **Angle** edit box in the **Line** command bar?

(a) Arc	(b) Sweep
(c) Value	(d) None of these

4. Which of the following tools is used to convert an existing sketched entity into a bezier spline curve?
 - (a) **Convert to Sketch**
 - (b) **Convert to Arc**
 - (c) **Convert**
 - (d) **Convert the Curve**
5. To invoke the sketching environment, choose **Home > Sketch > Sketch** from the **Ribbon** and then select a reference plane. (T/F)
6. The part file in Solid Edge is saved with a *.prt* extension. (T/F)
7. You can select entities by dragging a box around them. (T/F)
8. If **Overlapping** is the current selection mode, all entities that lie inside the box or even intersect the box will be selected. (T/F)
9. 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)
10. In Solid Edge, you can create a rectangle by specifying two diagonally opposite corners by using the **Rectangle by 2 Points** tool. (T/F)

Exercises

Exercise 1

Draw the profile of the base feature of the model shown in Figure 2-43. The profile to be drawn is shown in Figure 2-44. Do not dimension the profile because the dimensions are given for your reference only. **(Expected time: 30 min)**

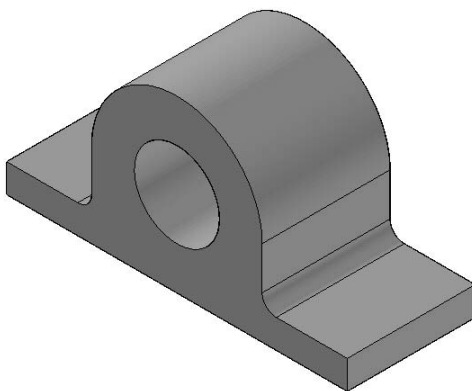


Figure 2-43 Model for Exercise 1

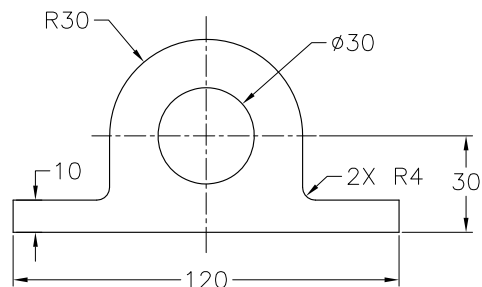


Figure 2-44 Profile for Exercise 1

Exercise 2

Draw the profile of the base feature of the model shown in Figure 2-45. The profile to be drawn is shown in Figure 2-46. Do not dimension the profile because the dimensions are given for your reference only.
(Expected time: 30 min)

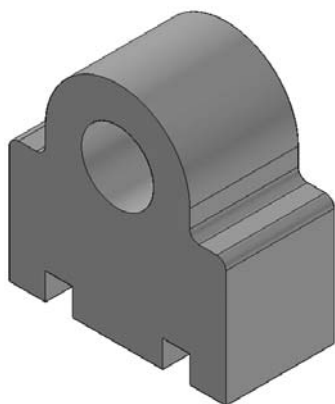


Figure 2-45 Model for Exercise 2

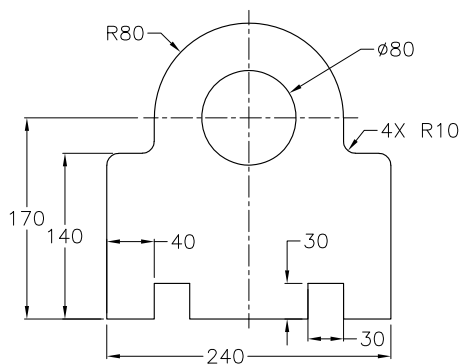


Figure 2-46 Profile for Exercise 2

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. F, 5. Sketch View, 6. A, 7. Chamfer, 8. No Trim, 9. 3,200, 10. Arc by 3 Points.