

# Chapter 2

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## Creating Sketches

### Learning Objectives

**After completing this chapter, you will be able to:**

- *Start a new part file*
- *Draw a line, rectangle, ellipse, circle, arc, point, and spline*
- *Create fillets*
- *Work with guides*
- *Write text in working environment*
- *Trim and mirror the sketched entities*
- *Measure a sketch entity*
- *Work with display filters*



## INTRODUCTION

Almost all the models design in Creo Direct consist of datums, sketched features, and placed features. For creating datums and placed features, you do not need to draw sketches. However, to create a sketched feature, it is necessary to draw or import its two-dimensional (2D) sketch. In Creo Direct, there is no separate sketch environment for creating sketches. You can draw sketches directly in the **Part** environment. The sketch created in the **Part** mode can be saved in the \*.prt, \*.igs, \*.neu, \*.stp, or \*.zip format. After creating the sketch, you can easily create the required feature without switching to any other mode.

To create any sketch in the **Part** mode of Creo Direct, certain basic steps have to be followed. The following points outline the steps to draw a sketch in the Creo Direct.

### 1. Starting a New Part File

As discussed earlier, you can create sketches in the **Part** mode of Creo Direct, as there is no separate sketching environment. Therefore to create sketches, first you need to invoke the **Part** mode.

### 2. Specifying the Plane for Sketching

By default, the Top plane is selected for sketching. However, you can select any other default plane: Right and Front for creating sketch. In addition to the default planes Front, Right, and Top, in Creo Direct, you can create additional reference plane as per your requirement. You will learn more about selecting planes and creating new reference datum planes in Chapter 4.

### 3. Sketching the Section Geometry

The different sketcher tools available in the **Part** mode can be used to sketch the required section geometry.

### 4. Measuring the Sketch

You can measure a sketch by using the measuring tools available in the **Measure** drop-down of the **Info** group in the **Home** tab. You can measure the sketch in terms of length, diameter, angle, and so on.

## STARTING A NEW PART FILE

To start a new Creo Direct Part file, choose **New** from the **File** menu or choose the **New** button from the **Data** group in the **Home** tab of **Ribbon**; the **New** dialog box will be displayed with two buttons in the **Type** area. To start a new file in the **Part** mode, select the **Part** button from the dialog box, if not already chosen, refer to Figure 2-1; a default name of the part file appears in the **Name** edit box. You can change the part name as required.

You can also change the template file to setup the drawing units. To do so, clear the **Use default template** check box and choose the **OK** button; the **New File Options** window will be opened. You can select the required template file for the drawing and then choose the **OK** button from the dialog box; the part mode will be invoked with the **Home** tab chosen in the **Ribbon**, as shown in Figure 2-2.

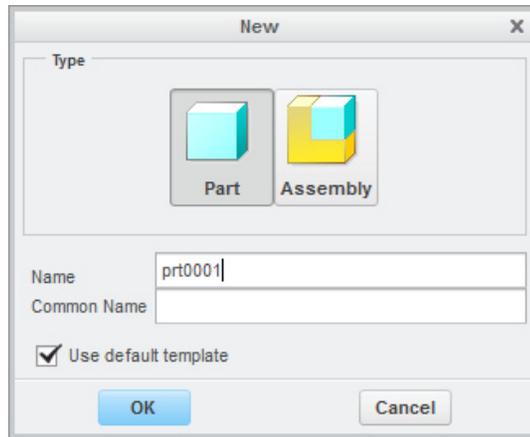


Figure 2-1 The New dialog box

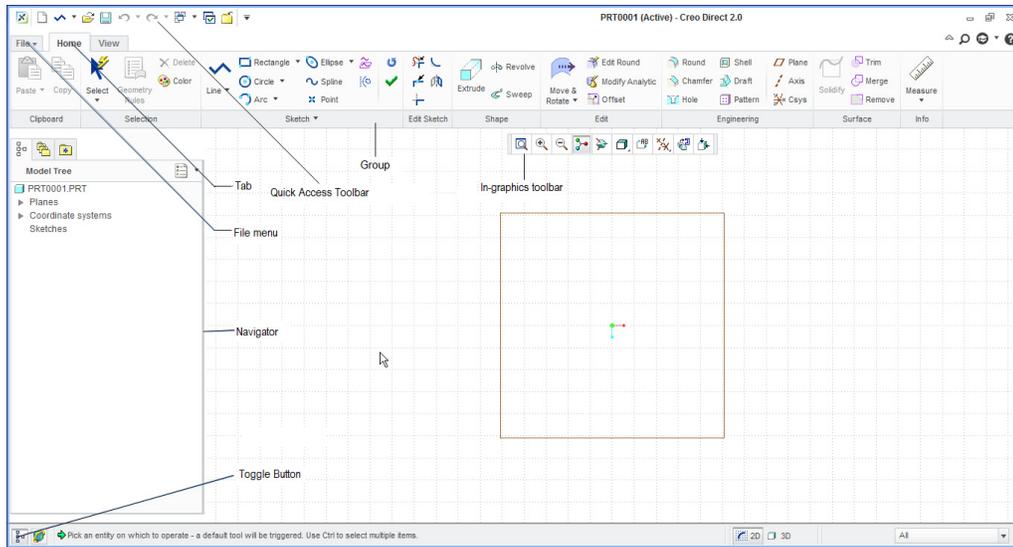


Figure 2-2 Initial screen displayed in the Part mode

The navigator is displayed on the left of the drawing area. In the navigator, the **Model Tree** tab is activated by default. You can hide the **Model Tree** by choosing the **Toggle the display of the navigation area** button available at the bottom left corner of the window.

Once the part mode of Creo Direct is invoked, you can draw sketches for creating the sketched features by using the sketching tools available in the **Home** tab of the **Ribbon**. The tools available for creating sketches in the **Home** tab are discussed next.

## SKETCHING THE SECTION GEOMETRY

In the **Part** environment, by default, the **Home** tab is chosen in the **Ribbon**. The **Sketch** and **Info** groups of the **Home** tab contain tools to draw and measure sketch entities. In this section, you will learn to draw and measure sketch entities by using these tools.

### Drawing a Line

In Creo Direct, there are two tools that can be used to draw lines: **Line/Arc Chain** and **Tangent** tool. These tools are available in the **Line** drop-down in the **Sketch** group of the **Home** tab in the **Ribbon**. The **Line/Arc Chain** tool is used to draw a continuous chain of lines, arcs, or combination of lines and arcs by specifying points in the drawing area. The **Tangent** tool is used to create a line tangent to any arc or circle. The procedure to create lines by using these tools is discussed next.

### Drawing a Line Using the Line/Arc Chain tool

**Ribbon:** Home > Sketch > Line > Line/Arc Chain



The following steps explain the procedure to create lines by using the **Line/Arc Chain** tool available in the **Line** drop-down:

1. Choose the **Line/Arc Chain** tool; the line tool will be invoked and a dynamic input edit boxes will be displayed attached to the cursor. Also, you will be prompted to specify the start point of the line to be created.
2. Specify the X and Y coordinates value of the point location where you want to specify the start point of the line. Note that after entering the X value in the **X** dynamic input edit box, you need to press the DOWN key from the keyboard to switch to **Y** edit box. You can also left click in the drawing area to directly specify the point.

As soon as you specify the start of the line, a rubber-band line appears from the selected point with the other end attached to the cursor and you will be prompted to specify the second point of the line. Note that the **X** and **Y** edit boxes of the dynamic input will be changed to **Length** and **Angle** edit boxes.

3. Specify the length and angle of the line to be created in the **Length** and **Angle** edit boxes of the dynamic input, respectively. On specifying the length and angle value, the line will be created and displayed in the drawing area.

As soon as you specify the second point of the line by defining its length and angle, a rubber-band line is still appears from the specified point with the other end attached to the cursor and you will be prompted to specify the third point of the line. In this way, you can keep on creating continuous chain of lines by specifying their length and angle values in the dynamic edit boxes.



#### Note

While drawing the sketch entities in Creo Direct, the symbols of guides will be displayed along with the entity being drawn. These symbols help you to specify the creation point for the entity,

by displaying the respective guide. For example, after specifying the start point of the line when you move the cursor horizontally toward the right or left, the symbol of horizontal guide will be displayed along with the rubber band line. Similarly, when you move the cursor vertically upward or downward, the symbol of vertical guide will be displayed. In other words, in *Creo Direct*, the guides will be displayed to the entities while drawing them. You will learn more about different type of guides later in this chapter.

You can also invoke the arc mode and create arcs by using the **Line/Arc Chain** tool. To invoke the arc mode while creating lines, press the S key. Once the arc mode invoked, specify the end point of the arc. Figure 2-3 shows a combination of lines and arcs created by using the **Line/Arc Chain** tool.

4. Press the middle mouse button to end the line creation.



#### Note

If you draw a single line, the color of the line drawn will be green. If you draw multiple lines, the color of all the lines drawn will be orange.

After drawing a line(s), when you press the middle mouse button to end the line creation, the line(s) drawn is highlighted in green color. In the **Part** environment, the green color of an entity indicates that it is selected. If you press the **DELETE** key, the selected entity will be erased from the drawing area.

## Drawing a Line Using the Tangent Tool

**Ribbon:** Home > Sketch > Line > Tangent

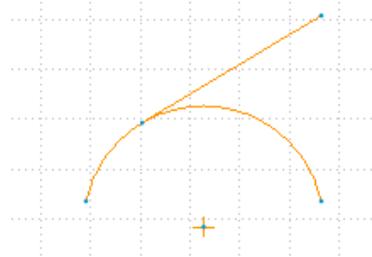


The **Tangent** tool is used to draw a line tangent to entities such as arc, ellipse, circle, or a combination of these. To draw a tangent using this tool follow the steps given next:

1. Choose the **Tangent** tool from the **Line** drop-down in the **Sketch** group; you will be prompted to select the start point on an arc or circle.
2. Select the entity to specify the start point; a rubber-band line will be attached with the cursor. As soon as you specify the second point, a line tangent to the arc will be drawn. Figure 2-4 shows a tangent line created by using the **Tangent** tool.



**Figure 2-3** The line created by using the **Line/Arc Chain** tool



**Figure 2-4** The tangent line created by using the **Tangent** tool

**Note**

By default, Top plane is selected as the sketch part. Therefore, when you create any sketch or sketch entities, it will be drawn on the Top Plane, by default. However, you can select any plane other than default plane for creating sketch by selecting them from the Navigator Tree.

## Creating a Rectangle

In Creo Direct, there are three tools available in the **Rectangle** drop-down that can be used to draw different types of rectangles: the **Corner**, **Slant**, and **Center**. The procedures to create various types of rectangles are given next.

### Creating a Corner Rectangle

**Ribbon:** Home > Sketch > Rectangle > Corner



You can create a rectangle by specifying two diagonally opposite corners. First corner point is the starting point of the rectangle and the second corner point is the end point of the rectangle. To create a rectangle by using the **Corner** tool, you need to follow the steps given below:

1. Invoke the **Corner** tool from the **Rectangle** drop-down; you will be prompted to select two points as corners of the rectangle. Click to specify the first point; you will be prompted to specify the second point. Also, the cursor will be attached to the opposite corner of the box.
2. Move the cursor to any non-collinear desired location in the drawing area and then click to specify the second point for the diagonal of the rectangle. Figure 2-5 shows the rectangle created by using the **Corner** tool.



Figure 2-5 The rectangle created by using the **Corner** tool

### Creating a Slant Rectangle

**Ribbon:** Sketch > Rectangle > Slant



You can create an inclined rectangle by using the **Slant** tool. To do so, you need to follow the steps given next.

1. Invoke the **Slant** tool from the **Rectangle** drop-down; you will be prompted to select two points to indicate the first side of the rectangle. Click to specify the first point; a red rubber-band line will be displayed with the cursor attached to its end point.
2. Specify the second point to create the first side of the rectangle; you will be prompted to specify the end point.
3. Move the cursor in the perpendicular direction in the drawing area and then click to specify the end point for creating rectangle.

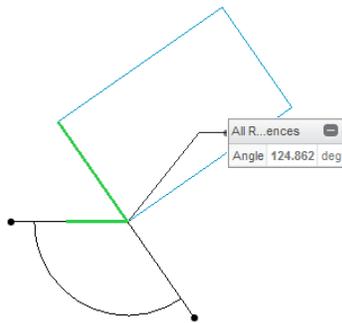
## Creating a Center Rectangle

**Ribbon:** Sketch > Rectangle > Center

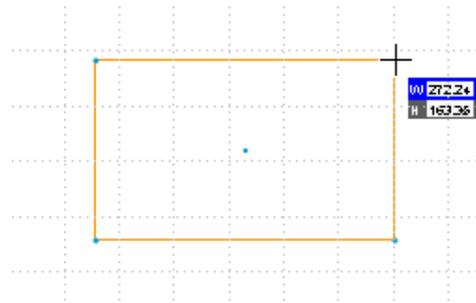
 You can create a rectangle by specifying its center and a corner using the **Center** tool. To create a rectangle by using the **Center** tool, you need to follow the steps given next:

1. Invoke the **Center** tool from the **Rectangle** drop-down available in the **Sketch** group; you will be prompted to specify the center point of the rectangle. Click to specify the center point; a red rubber-band box appears with the cursor attached to the end point of the diagonal of the rectangle.
2. Click at desired point to create the rectangle.

Figure 2-6 and 2-7 show the rectangles created by using the **Slant** and **Center** tools, respectively.



*Figure 2-6 The rectangle created with slant side using the **Slant** tool*



*Figure 2-7 The rectangle created using the **Center** tool*

## Drawing a Circle

You can draw circles by using the **Center and Point**, **Concentric**, **2-Point**, **3-Point**, and **Tangent** tools. These tools are available in the **Circle** drop-down in the **Sketch** group. The steps to create a circle by using these tools are given next.

### Drawing a Circle Using the Center and Point Tool

**Ribbon:** Sketch > Circle > Center and Point

 The **Center and Point** tool is used to draw a circle by specifying the center of the circle and a point on its circumference. To draw a circle using the **Center and Point** tool, follow the steps given next.

1. Choose the **Center and Point** tool; you will be prompted to select the center of the circle.

2. Click in the drawing area to specify the center of the circle; you will be prompted to select a point on the circumference of the circle. Also, an orange rubber-band circle will be displayed with its center at the specified point and the cursor attached to its circumference.
3. Move the cursor to specify the size of the circle. Click at a desired point to complete the creation of the circle; you will be prompted again to select the center of the circle.
4. Repeat steps 2 and 3 if you want to draw more circles, else press the middle mouse button to abort the process.

### Drawing a Circle Using the Concentric Tool

**Ribbon:** Sketch > Circle > Concentric



To draw a circle using the **Concentric** tool, follow the steps given next:

1. Choose the **Concentric** tool from the **Circle** drop-down in the **Sketch** group; you will be prompted to select an arc to determine the center. You can select an arc or a circle to specify the center point.
2. Click on an arc or a circle; the center point of the circle will be defined as the center point of the selected arc or circle. Move the mouse and click at the required location to specify the radius of the circle to be drawn.
3. To finish the process of creating circle, press the middle mouse button.

### Drawing a Circle Using the 2-Point Tool

**Ribbon:** Sketch > Circle > 2-Point



To draw a circle using the **2-Point** tool, follow the steps given next:

1. Choose the **2-Point** tool from the **Circle** drop-down; you will be prompted to specify the first point on the circumference of the circle.
2. As you specify the first point, an orange rubber-band circle will be attached to the circle and you will be prompted to select the second point. Move the mouse and click to specify the second point; a circle is drawn and you will be prompted again to select the first point on the circle to draw the next circle.
3. You can press the middle mouse button to finish the process of creating the circle.

### Drawing a Circle Using the 3-Point Tool

**Ribbon:** Sketch > Sketching > Circle > 3-Point



To draw a circle using the **3-Point** tool, follow the steps given next:

1. Choose the **3-Point** tool from the **Circle** drop-down; you will be prompted to specify the first point on the circumference of the circle.
2. Click to specify the first point at the desired location in the drawing area; you will be prompted to select the second point on the circumference of the circle. Move the cursor and click to specify the second point in the drawing area.
3. As you specify the second point, an orange rubber-band circle will be displayed with the cursor attached to it and you are prompted to specify it. Move the cursor to specify the third point on the circumference of the circle the circle and click to specify the third point; a circle is drawn and you are prompted again to select the first point on the circle to draw the next circle.
4. You can press the middle mouse button to finish the process of creating the circle.

### Drawing a Circle Using the Tangent Tool

**Ribbon:** Sketch > Circle > Tangent



The **Tangent** tool is used to draw a circle tangent to two or three existing entities. This tool references other entities to draw a circle. The circle created using this tool is drawn irrespective of the points selected on the entities. To draw a circle using the **Tangent** tool between three entities, follow the steps given next:

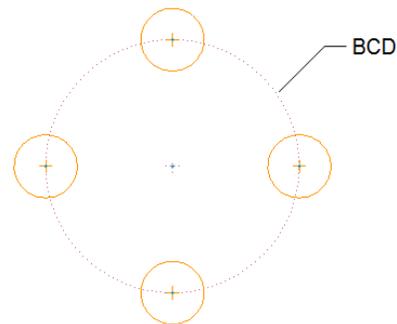
1. Choose the **Tangent** tool from the **Circle** drop-down; you will be prompted to select the start location on an arc, circle, or line.
2. Select the first entity; the color of the selected entity changes to green and you will be prompted to select the second location on an arc, circle, or line. Select the second tangent entity; you will be prompted to select the third location on an arc, circle, or line. Select the third tangent entity; a circle tangent to these three entities is drawn. Note that you can click in the screen to specify the third point if you are creating arc tangent to two entities.
3. To end the process of circle creation, press the middle mouse button.

### Drawing a Construction Entities

**Ribbon:** Sketch > Construction Mode



A construction entity is an entity that is used to align entities, create diametrical or radial dimensions, and to reference the entities. For example, in the sketch of a flange, centers of the circles lie on a particular bolt circle diameter (BCD) that is defined using a construction circle, refer to Figure 2-8.



**Figure 2-8** Construction circle defining BCD

To create a construction entity, choose the **Construction Mode** toggle button and then draw an entity using the tools available in the **Sketch** group; the entity will appear dotted in green color, indicating that it is a construction entity.

## Drawing an Ellipse

You can draw an ellipse by using the **Center & Axis**, and **2-Point** tools. These tools are available in the **Ellipse** drop-down in the **Sketch** group. The methods to create an ellipse by using these tools are given next.

### Drawing an Ellipse Using the 2-Point Tool

**Ribbon:** Sketch > Ellipse > 2-Point



To draw an ellipse using the **2-Point** tool, follow the steps given next:

1. Choose the **2-Point** tool from the **Ellipse** drop-down; you will be prompted to select the start point of the major axis of the ellipse.
2. Click in the drawing area to specify the start point of the major axis; you will be prompted to specify the endpoint of the major axis.
3. Click in the drawing area to specify the endpoint; an orange rubber-band ellipse will be displayed with the cursor. Also, you will be prompted to select a point on the minor axis to define the ellipse.
4. Click to specify the point; an ellipse will be created. You can press the middle mouse button to end the creation of the ellipse.

### Drawing an Ellipse Using the Center & Axis Tool

**Ribbon:** Sketch > Ellipse > Center & Axis



To draw an ellipse using the **Center and Axis** tool, follow the steps given next:

1. Choose the **Center & Axis** tool from the **Ellipse** drop-down; you will be prompted to specify the center of the ellipse.
2. Click at the desired location in the drawing area to specify the center point; you will be prompted to select the endpoint of the major axis of the ellipse. Click in the drawing area to specify the point; an orange rubber-band ellipse will be displayed attached with the cursor. Move the cursor in the drawing area to specify the size of the ellipse.
3. Specify the endpoint on the minor axis of the ellipse; the ellipse will be drawn.

## Drawing an Arc

There are four tools available in the **Arc** drop-down in the **Sketch** group to draw an arc. The procedures to draw arcs using these tools are discussed next.

## Drawing an Arc Using the 3-Point / Tangent End Tool

**Ribbon:** Sketch > Arc > 3-Point / Tangent End



The **3-Point / Tangent End** tool is used to draw arcs tangent to an existing entity at the endpoint. You can also draw an arc using this tool by defining three points in the drawing area. The steps to draw these arcs are discussed next.

1. Choose the **3-Point / Tangent End** tool from the **Arc** drop-down; you will be prompted to select the start point of the arc.
2. Specify three points in the drawing area to draw an three point arc.

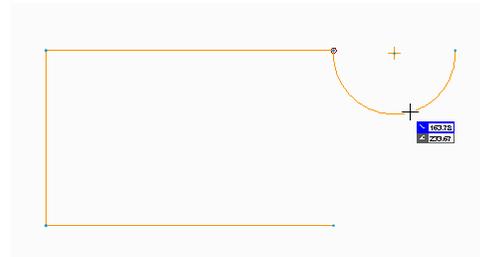
If you want to draw an tangent arc using this tool, atleast one sketch entity should be available in the drawing area. To do so, invoke this tool, select the endpoint of the existing entity. As you select the endpoint, the **Target** symbol appears at the endpoint of the entity, refer to Figure 2-9. This target symbol is green colored and divided into four quadrants.

To draw a tangent arc, move the cursor along a tangent direction through a small distance; a rubber-band arc appears with one end attached to the specified endpoint and the other end attached to the cursor. Next, specify the second point; a arc tangent to the existing entity is created.

To draw a normal arc using this tool, when the Tangent symbol displayed, move the cursor through a small distance in the direction normal to the line, a dotted line displayed and you will be prompted to specify the second point of the arc. Click to specify the second point; the preview of the normal arc displayed, refer to Figure 2-10 and you will be prompted to specify third point. Specify the third point; a normal arc will be created, refer to Figure 2-10.



**Figure 2-9** The **Target** symbol appears after selecting the endpoint of the entity



**Figure 2-10** Cursor moved out of the **Target** symbol along the tangent direction

- Once you are done with creating three point arc, tangent arc, or normal arc using this tool, press the middle mouse button to exit the arc creation.

### Drawing an Arc Using the Center and Ends Tool

**Ribbon:** Sketch > Arc > Center and Ends



To draw an arc using the **Center and Ends** tool, follow the steps given next:

- Choose the **Center and Ends** tool from the **Arc** drop-down; you will be prompted to select the center of the arc.
- Click to specify the center point for the arc in the drawing area; an orange colored center mark will appear at that point and you will be prompted to select the start point of the arc. As you move the cursor, a dotted circle appears attached to the cursor.
- Specify the start point of the arc on the circumference of the dotted circle; a red rubber-band arc will appear from the start point. The length of this arc will change dynamically as you move the cursor and you will be prompted to select the endpoint of the arc.
- Move the cursor to specify the size the arc and then click to specify the endpoint of the arc; an arc will be drawn between the two specified points.

### Drawing an Arc Using the Tangent Tool

**Ribbon:** Sketch > Arc > Tangent



The **Tangent** tool is used to draw an arc that is tangent to two or three selected entities. To draw an arc between three entities using the **Tangent** tool, follow the steps given next:

- Choose the **Tangent** tool from the **Arc** drop-down; you will be prompted to select the start point on an arc, circle, or line.
- As soon as you select the first entity, the color of the selected entity will change to green and you will be prompted to select the second location on an arc, circle, or line.
- On selecting the second location, you will be prompted to select the third location on arc, circle, or line. Select the third entity; an arc is drawn tangent to the three selected entities.

Note that you can click on the screen to specify the third point if you are creating an arc tangent to two entities. You can continue drawing arcs or press the middle mouse button to abort arc creation.

## Drawing an Arc Using the Concentric Tool

**Ribbon:** Sketch > Arc > Concentric



The **Concentric** tool is used to draw an arc concentric to an existing arc or circle. To draw an arc using the **Concentric** tool, follow the steps given next:

1. Choose the **Concentric** tool from the **Arc** drop-down; you will be prompted to select an arc to determine the center of the arc to be created.
2. Select an entity; a dotted circle will appear on the screen and you will be prompted to select the start point of the arc. Click to specify the start point; an orange rubber-band arc will appear with one end attached to the start point. As you move the cursor, the length of the arc will change and you will be prompted to select the endpoint of the arc.
3. Click to specify the endpoint; the arc will be created concentric to the arc or circle selected.

You can continue drawing another arc or end the arc creation by pressing the middle mouse button.

## Placing a Point

**Ribbon:** Sketch > Point



Points are used to specify locations. You can use a point as reference for creating other geometric elements. Points can also be used for dimensioning the vertices that are removed while applying fillets. To draw a point using the **Point** tool, follow the steps given next:

1. Choose the **Point** tool from the **Sketch** group; you will be prompted to select a location for the point.
2. As you click to select the location of the point, the point will be placed at that location in the drawing area.

## Creating a Spline

**Ribbon:** Sketch > Spline



To draw a spline, choose the **Spline** tool from the **Sketch** group. The steps to create a spline are discussed next.

1. Choose the **Spline** tool from the **Sketch** group; you will be prompted to specify the location for spline.
2. Use the left mouse button to specify the start point for the spline. Similarly, specify additional points in the graphics window; a spline will be drawn passing through all

specified points. Press the middle mouse button to end the creation of spline. All points through which the spline passes are called interpolation points.

## Creating Fillets

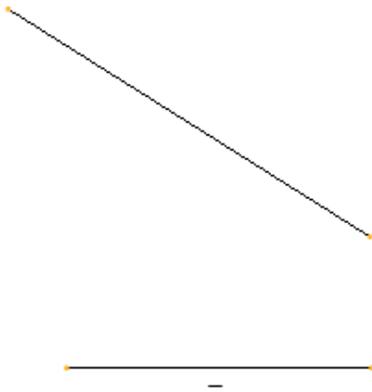
**Ribbon:** Edit Sketch > Fillet



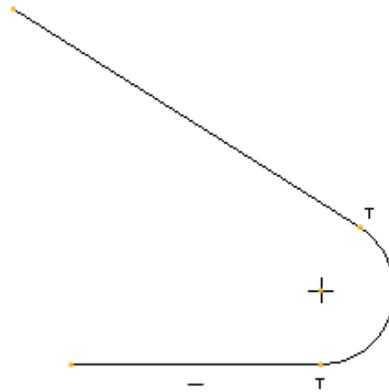
A fillet is the arc formed at the intersection of two lines, a line and an arc, or two arcs. It is controlled by the radius or diameter dimension of the fillet. The circular fillet thus created is an arc with its endpoints tangent to the two lines. The procedure to create a fillet is discussed next:

1. Choose the **Fillet** tool from the **Edit Sketch** group of the **Home** tab; you will be prompted to select the first entity.
2. Click on the entity to select as the first entity; you will be prompted to specify the final entity.
3. Click on the entity to select as the final entity; a fillet will be created between the entities selected.

Figure 2-11 shows two unjoined lines and Figure 2-12 shows the circular fillet created between them.



**Figure 2-11** Two unjoined lines



**Figure 2-12** Fillet created between the two unjoined lines

Figure 2-13 shows two lines that join at a point and Figure 2-14 shows the circular fillet created at the joint. The corner is automatically deleted.

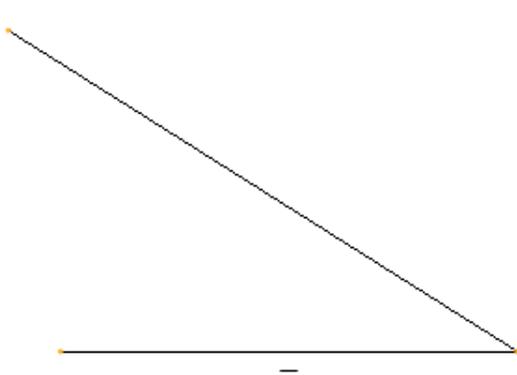


Figure 2-13 Two lines joining at a point

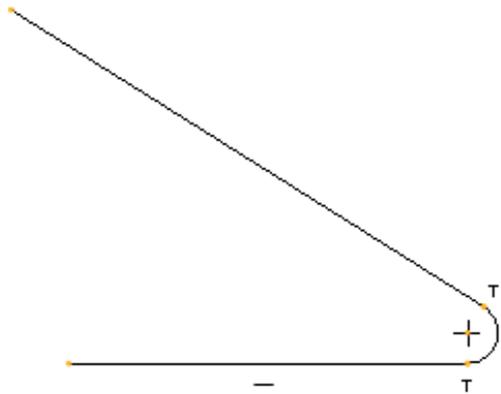


Figure 2-14 Filleted corner

## WORKING WITH GUIDES

Guides are the symbols that are displayed while creating the sketches. By using the guides, you can place and orient an entity with respect to other entity while creating the sketches. For example, while creating a line, if a line becomes parallel to another line, Creo Direct displays the parallel guide symbol on both the lines. Now, if you confirm the line creation, the line is drawn parallel to the other line.

### Types of Guides

There are various types of guides in Creo Direct such as Vertical, Horizontal, Tangent, and so on, refer to Figure 2-15.

While creating the sketch, the guides will be displayed by default. You can toggle the display of guides by using the check boxes available in the **Aids** dialog box. To invoke the dialog box, click on the inclined arrow  available in the right side of the **Aids** group of the **View** tab; the **Aids** dialog box will be displayed, refer to Figure 2-15. Now, you can toggle the display of guides by using these check boxes. You can also select the **Show guides** check box to enable or disable the display of all the guides.

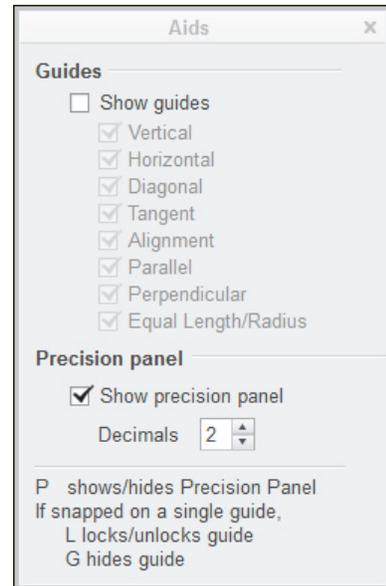


Figure 2-15 The Aids dialog box



**Tip:** 1. You can specify the decimals value for the precision by entering the value of decimals in the **Decimal** edit box of the **Precision panel** group.

2. You can lock the guides while creating the sketches. To do so, press the **L** key while creating the sketch; the guides of the entity will be locked. To unlock the guides, press the **L** key again.

## WRITING TEXT IN THE SKETCHER ENVIRONMENT

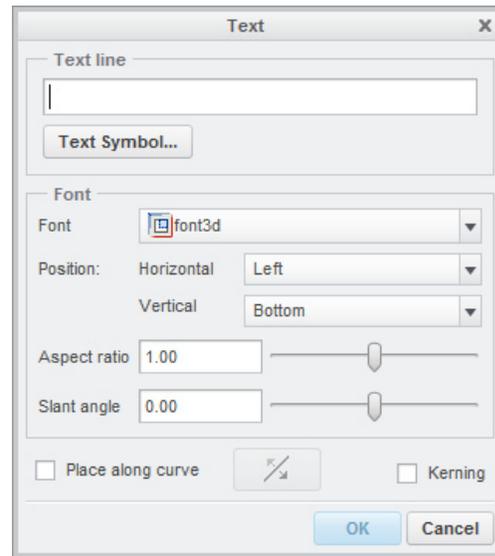
**Ribbon:** Sketch > Text

There are various instances when a designer needs to write text on the model, for example, to create a label, model number, company name, and so on. In Creo Direct, you can write this text in the sketcher environment.



In the sketcher environment, the text is written using the **Text** tool available in the **Sketch** group. The following steps explain the procedure to write text in the sketcher environment:

1. Choose the **Text** tool from the expanded **Sketch** group; you will be prompted to select the start point of line.
2. Click to specify the start point; you will be prompted to select the second point of line to determine the text height and orientation.
3. To write the text upright, the second point should be above the start point and in a straight line. If the second point is below the start point, the text will be written down from right to left. Click to specify the second point; the **Text** dialog box will be displayed, as shown in Figure 2-16.



*Figure 2-16 The Text dialog box*

After specifying the second point, a construction line will be drawn with height equal to the distance between the two points.

The height and orientation of the text depends on the height and angle of the construction line, created by specifying two points. If the construction line is drawn at an angle, then the text is written at that angle.

4. Enter the text in the **Text line** edit box, which can be up to 79 characters. As you enter the text, the text will be displayed dynamically in the graphics window. You can choose the desired font of the text from the **Font** drop-down list. The aspect ratio and the slant angle of the text can be controlled by using the slider bars.
5. Choose the **OK** button from the **Text** dialog box to exit it.

Figure 2-17 shows the text created by specifying the text height as 6.



*Figure 2-17 The text created*

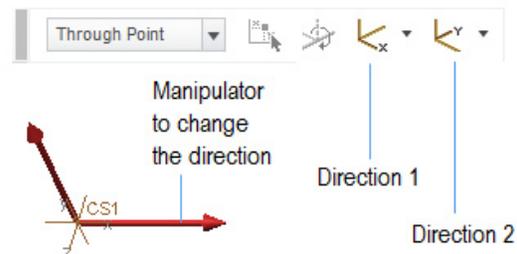
## CREATING A REFERENCE COORDINATE SYSTEM



The Coordinate System is created based on the selected geometry. The **Csyt** tool in the **Engineering** group is used to create a coordinate system that will act as a reference for dimensioning. You can dimension the splines using the coordinate system. Thus, it provides you the flexibility to modify the spline points by specifying different coordinates with respect to the coordinate system.

Also, it is used in the **Assembly** mode of Creo Direct. You will learn more about **Assembly** mode in later chapters. The following steps explain the procedure to create a coordinate system:

1. Choose the **Csyt** tool from the **Engineering** group; you will be prompted to select a location for the coordinate system.
2. Click to place the coordinate system at the desired location on the geometry or on the plane; a preview of coordinate system with dashboard will be displayed, refer to Figure 2-18. You can change the direction of the axes by using the options available in the **Direction 1** and **Direction 2** drop-downs. You can also create Coordinate System by selecting two edges. After creating the Coordinate System, press the middle mouse button to exit the tool.



*Figure 2-18 The default coordinate system displayed after using the Csyt tool*

## EDITING THE SKETCHED ENTITIES

While creating a design, there are a number of places where you need to remove the unwanted and extended entities. You can do this by using the trimming tools that are available in the **Edit Sketch** group. These tools are discussed next.

### Trim Segment

**Ribbon:** Edit Sketch > Trim Segment



This tool is used to trim the entities that extend beyond the point of intersection. It is also used to delete the selected entities. After choosing the **Trim Segment** tool, when you move the cursor over an entity, the entity will be highlighted in green color. Now, if you click on line entity, the highlighted entity will be trimmed.

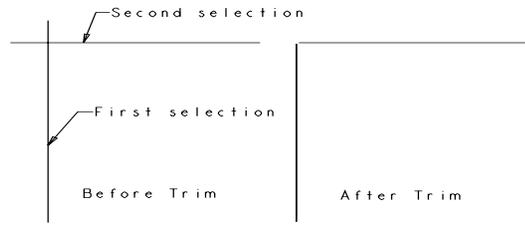
## Corner

**Ribbon:** Edit Sketch > Corner



The **Corner** tool is used to trim two entities at their corners. When you trim entities using this option, the portion from where you select the entities is retained and the other portion is trimmed. To trim entities using this tool, follow the steps discussed next:

1. Choose the **Corner** tool from the **Edit Sketch** group; you will be prompted to select two entities to be trimmed.
2. Click to select the two entities on the sides that you want to retain after the trimming action. These two entities must be intersecting entities. The entities are trimmed from the point of intersection, refer to Figure 2-19.



*Figure 2-19 Trimming the lines*



**Tip:** You can also join two entities by extending them. To do so, choose the **Corner** tool and select the two entities that you want to join; the two entities will get joined at a point.

## Divide

**Ribbon:** Edit Sketch > Divide



The **Divide** tool is used to divide an entity into a number of parts or entities by specifying points on it. To do so, follow the steps given next:

1. Choose the **Divide** tool from the **Edit Sketch** group; you will be prompted to specify a point on entity to be divided.
2. Click to select the entity such as line, spline, and so on, at the point where you want to divide it; the entity is divided into two different entities. They can now be treated as two separate entities.
3. Similarly, you can break other entities like circles or arcs into several small entities.

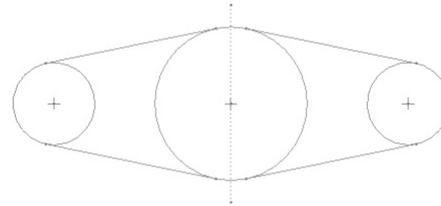
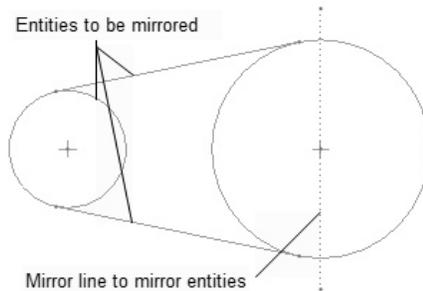
## Mirror

**Ribbon:** Edit Sketch > Mirror



The **Mirror** tool is used to mirror sketched geometries about a line and thus helps in saving the time required to create the sketched entities. To mirror the sketched entity, follow the steps given next:

1. Select the entities that you want to mirror, refer to Figure 2-20; the selected entities turn green in color.
2. Choose the **Mirror** tool from the **Edit Sketch** group; you will be prompted to select the line about which you want to mirror. Select the line, refer to Figure 2-20; the selected entities will be mirrored about the line, refer to Figure 2-21.



**Figure 2-20** Selecting the entities to be mirrored about the mirror line

**Figure 2-21** Sketch after mirroring the geometries

## MEASURING THE BASIC ENTITIES

You can measure the sketched entities by using the measuring tools. Procedures to measure the sketch are given below.

### Measuring the length

**Ribbon:** Info > Measure > Length

 You can measure the length of the sketched entity by using the **Length** tool. To do so, choose the **Length** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select the references for length analysis. When you select a line as reference whose length to be measure; the length of the line will be displayed in a box attached to the line, refer to Figure 2-22.

### Measuring the Angle

**Ribbon:** Info > Measure > Angle

 To measure the angle between two entities, choose the **Angle** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select the references for angle analysis. Now, hold the CTRL key and select two lines for reference between which you want to measure angle. As you select the lines, the angle will be displayed in a box attached to the line, as shown in Figure 2-23.

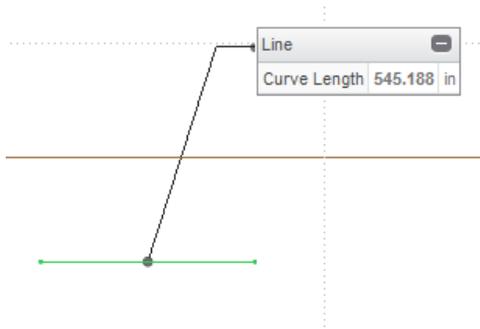


Figure 2-22 Linear measurement

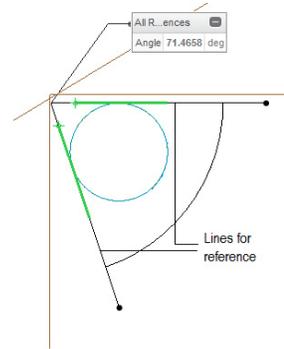


Figure 2-23 Angular measurement

## Measuring the Diameter

**Ribbon:** Info > Measure > Diameter

For diameter measurement, choose the **Diameter** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select the references for diameter analysis. When you select the circle as reference, the diameter of a circle will be displayed in a box with arrow attached with the circle, as shown in Figure 2-24. Note that you can not measure the diameter of circle while creating it.

## Measuring the Distance

**Ribbon:** Info > Measure > Distance

The **Distance** tool is used to measure the distance between the two entities of a sketch. To measure a sketch using this option, follow the steps given next:

1. Choose the **Distance** tool from the **Measure** drop-down; you will be prompted to select the references for measuring the distance.
2. Hold the CTRL key and then click on the entities. The color of the entities changes from orange to green and the distance between them will be displayed in a box, as shown in Figure 2-25.

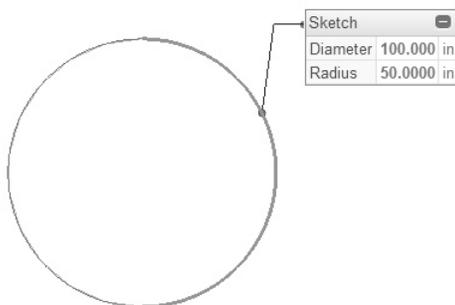


Figure 2-24 Diameter measurement

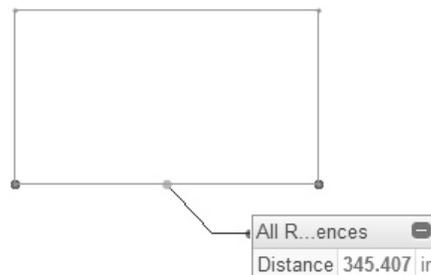


Figure 2-25 Distance measurement

## Measuring the Area

**Ribbon:** Info > Measure > Area

 To measure the area of a closed section, choose the **Area** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select the references for area analysis. Now, select all the entities that define a region by pressing the CTRL key; the measurement will be displayed in a box with an arrow attached to the entity, as shown in Figure 2-26.

## Measuring the Volume

**Ribbon:** Info > Measure > Volume

 To measure the volume of solid body, choose the **Volume** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select the references for volume analysis. Now, select the body whose volume is to be measured; the volume measured will be displayed in a box attached to the entity, as shown in Figure 2-27.

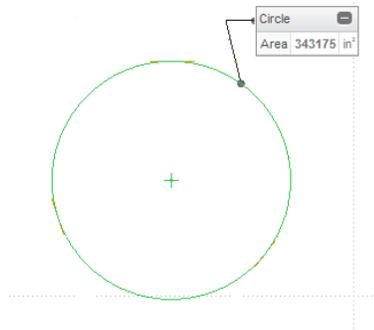


Figure 2-26 Area measurement

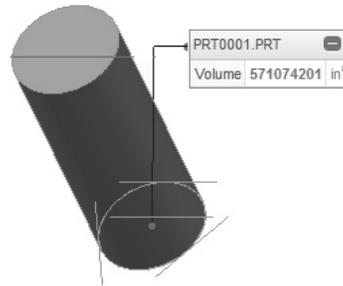


Figure 2-27 Volume measurement

## Generating the Summary

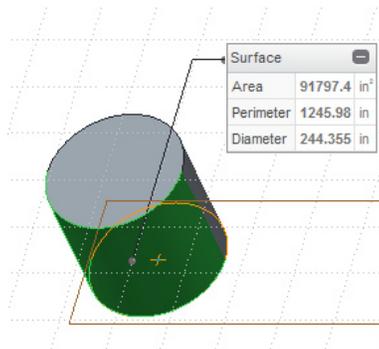
**Ribbon:** Info > Measure > Summary

 You can use the **Summary** tool to get the summary of the measurements related to an entities selected. To do so, choose the **Summary** tool from the **Measure** drop-down in the **Info** group; you will be prompted to select references for summary analysis. Select the surfaces whose summary to be generated. If you select a surface, the summary related to measurements will be displayed, as shown in Figure 2-28.

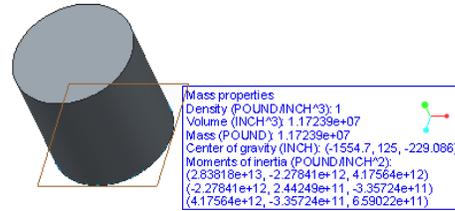
## Measuring Mass Properties

**Ribbon:** Info > Measure > Measure Mass Properties

 By using the **Measure Mass Properties** tool, you can get the mass properties of an object such as density, volume, center of gravity, moment of inertia, and so on. To do so, choose the **Measure Mass Properties** tool available in the **Measure** drop-down of the **Info** group in the **Home** tab; the mass properties of the selected object will be displayed, as shown in Figure 2-29.



**Figure 2-28** The information displayed after using the **Summary** tool



**Figure 2-29** The information displayed after using the **Measure Mass Properties** tool



### Note

You will learn about creating 3D objects in later chapters.

## Clearing all selections



You can remove all the measurements by using the **Clear all selection** tool. This tool will be available in the **Measure** toolbar only when you choose a measuring tool. This tool is used to remove all measurements from the entity. To remove the measurements, choose the **Clear all selections** tool; all the measurements will be removed.

## DRAWING DISPLAY OPTIONS

While working with complex sketches, sometimes you need to increase the display of a particular portion of a sketch so that you can work on the minute details of the sketch. For example, while drawing the sketch of a piston and you need to work on the minute details of the grooves of the piston rings. To work on these minute details, you will have to enlarge the view of these grooves. You can enlarge or reduce the view of the drawing using various drawing display tools provided in Creo Direct. These tools are available in the **In-Graphics** toolbar. Some of these drawing display options are discussed next. The remaining drawing display options will be discussed in the later chapters.

### Zoom In



This tool is available in the **In-Graphics** toolbar and is used to enlarge the view of the drawing on the screen. After choosing the **Zoom In** tool from the **In-Graphics** toolbar, you will be prompted to define a box. The area that you will enclose inside the box will be enlarged and displayed in the drawing area. Note that when you enlarge the view of the drawing, the original size of the objects is not changed. To exit the **Zoom** tool, right-click in the drawing area.

### Zoom Out



This tool is available in the **In-Graphics** toolbar and is used to reduce the view size of the drawing on the screen, thus increasing the drawing display area. Each time you choose this button, the display size of the entity in the drawing area is reduced.

**Note**

1. Scrolling the mouse down will zoom in and scrolling it up will zoom out the view. Alternatively, when you use **CTRL**+middle mouse button and drag the mouse upward, the sketch is zoomed out and when you drag the mouse downward, the sketch is zoomed in.
2. Use **SHIFT**+middle mouse button to pan the model.

**Refit**

This tool is available in the **In-Graphics** toolbar. It is used to reduce or enlarge the view such that all entities that form the sketch are fitted inside the current display. Note that the dimensions may not necessarily be included in the current display.

**Repaint**

While working with complex sketches, some unwanted temporary information is retained on the screen. It may include the shadows of the deleted sketched entities, dimensions, and other information. This unwanted information can be removed from the drawing area by using the **Repaint** tool available in the **In-Graphics** toolbar.

## SKETCHER DISPLAY FILTERS

While working with the sketches, sometimes you need to disable the display of some of the sketcher components such as grids, precision panel, and so on. You can disable or enable their display using the options available in the **Grid** group of the **View** tab. Also, you can modify the grid settings. The options available in the **Grid** group are discussed next.

**Grid**

**Ribbon:** View > Grid > Grid



You can use the **Grid** tool to set the parameters of grids on the screen. When you choose this tool, the **Grid Settings** dialog box will be displayed, as shown in Figure 2-30. The options available in this dialog box are explained next.

**Grid Type**

The options in this area are used to set the type of grids. There are two radio buttons available in this area. When you choose the **Cartesian** radio button, the display of grids will be rectangular. When you choose the **Polar** radio button, the display of grids will be circular.

**Grid Spacing**

The options in this area are used to specify the spacing between grid lines. There are two radio buttons available in this area. When you choose the **Dynamic** radio button, the spacing between both X and Y direction is symmetrical. But, if you choose the **Static** radio button, you can specify the spacing values in the **X Spacing** and **Y Spacing** edit boxes.

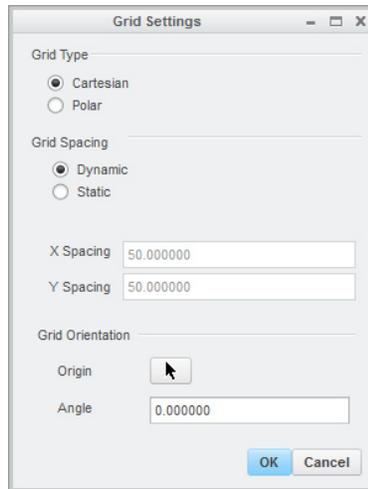


Figure 2-30 The *Grid Settings* dialog box

### Grid Orientation

The options in this area are used to set the orientation of grids with respect to an active plane. You can specify the value of orientation by setting the values in the **Angle** edit box.

### Disp Grid

**Ribbon:** View > Grid > Disp Grid

You can use this check box to enable or disable the view of the grid. Sometimes you may not be able to see the grid as it becomes very dense. To see the grid in such cases, you need to zoom in the screen to see grid.

## TUTORIALS

### Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 2-31. The sketch is shown in Figure 2-32. The dimensions are given for the reference only.

(Expected time: 30 min)

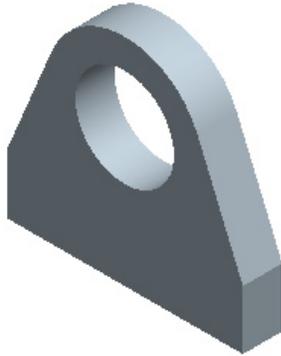


Figure 2-31 Model for Tutorial 1

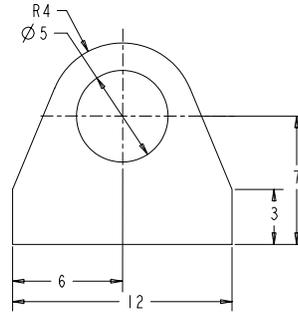


Figure 2-32 Sketch of the model

The following steps are required to complete this tutorial:

- Start Creo Direct session.
- Set the working directory and create a new part file.
- Draw lines by using the **Line/Arc Chain** tool, refer to Figure 2-34.
- Draw circles, refer to Figures 2-35.
- Save the sketch and close the file.

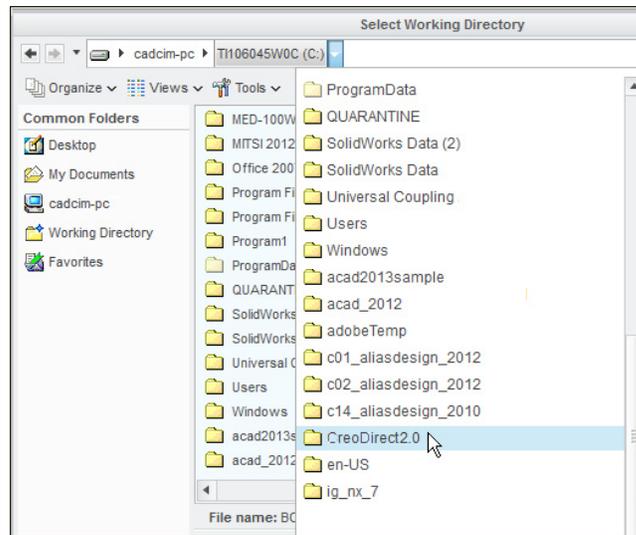
### Starting Creo Direct

Start Creo Direct by double-clicking on the **Creo Direct 2.0** icon on the desktop of your computer. Alternatively, choose **Start > All Programs > PTC Creo > Creo Direct 2.0** from the taskbar.

### Setting the Working Directory

After the Creo Direct session is started, first set the working directory where you will save the work done in the current session. You can also set an existing directory on your system as the working directory. Since it is the first tutorial of this chapter, you also need to create a folder with the name *c02*.

- Choose the **Select Working Directory** option from the **Manage Session** flyout of the **File** menu; the **Select Working Directory** dialog box is displayed. Alternatively, you can select the **Select Working Directory** button from the **Data** group in the **Home** tab to open the **Select Working Directory** dialog box.
- In the dialog box, browse to *C:\CreoDirect2.0* folder in the **Select Working Directory** dialog box, refer to Figure 2-33. If the *CreoDirect2.0* folder does not exist, create this folder before setting the working directory.



*Figure 2-33 The **CreoDirect2.0** folder selected in the **Select Working Directory** dialog box*

3. Choose the **Organize** button from the top of the **Select Working Directory** dialog box to display the flyout. Choose the **New Folder** option from the flyout; the **New Folder** dialog box is displayed.
4. Enter **c02** in the **New Directory** edit box and choose **OK** from the **New Folder** dialog box; a new folder named *c02* is created at *C:\CreoDirect2.0*.
5. Choose **OK** from the **Select Working Directory** dialog box to set the working directory to *C:\CreoDirect2.0\c02*; the message **Successfully changed to C:\CreoDirect2.0\c02 directory** is displayed in the message area.

### Starting a New Object File

1. Choose the **New** tool from the **Quick Access Toolbar** or press CTRL+N; the **New** dialog box is displayed. In this dialog box, select the **Part** button from the **Type** area; the default name of the file appears in the **Name** edit box. 
2. Enter **c02tut01** in the **Name** edit box and choose the **OK** button; the **Part** mode is invoked.

When the **Part** mode is invoked, the **Model Tree** is displayed on the left of the drawing area.

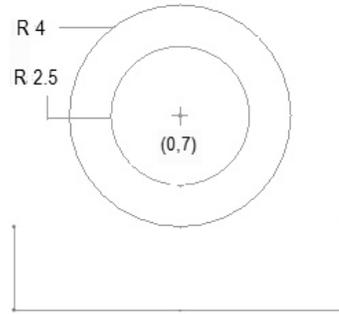
3. Choose the **Toggle the display of the navigation area** button available at the bottom left corner of the screen to close the **Model Tree**. On closing the tree, the drawing area is increased. 

### Creating the Sketch

1. Choose the **Line/Arc Chain** tool from the **Line** drop-down available in the **Home** tab; the cursor is changed to dynamic input cursor with X and Y coordinates values of the current location of the cursor displaced in it. Note that as you move the cursor the coordinate values will be modified accordingly. 
2. Choose the **Sketch Plan View** tool from the **In-graphics** toolbar to make the sketching plane parallel to the screen. 
3. Specify **6** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **3** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. After specifying the coordinates, press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
6. Move the cursor vertical downward; a symbol of vertical guide is displayed.
7. Specify **3** as the length of the line in the dynamic input. Next, press ENTER; the vertical line of length 3 is created.
8. Move the cursor horizontally toward the left; a symbol of horizontal guide is displayed.
9. Specify **12** as the length of the line in dynamic input and press ENTER.
10. Move the cursor vertically upward in the drawing area; a vertical rubber-band line extends with it. Notice that at a particular point where the length of the left vertical line is equal to the length of the right vertical line, the **L** guide symbol is displayed on both the vertical lines.
11. When the **L** guide symbol appears on the vertical line, click to specify the endpoint of the vertical line. The rubber-band line is still attached to the cursor. Press the middle mouse button to end the line creation, refer to Figure 2-34.
12. Choose the **Center and Point** tool from the **Circle** drop-down of the **Sketch** group in the **Home** tab; you will be prompted to specify the center of circle. 
13. Specify the coordinates as **0, 7** in the X and Y edit boxes of the dynamic input as the center of the circle; you will be prompted to specify the radius of circle.
14. Specify the radius of circle as **2.5** in the dynamic input; the circle is created.
15. Similarly, create the other circle with centerpoint and radius values as **0, 7** and **4**, respectively. Figure 2-35 shows the two circles created.

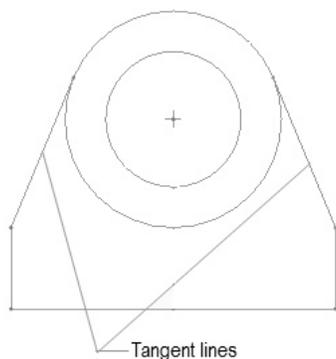


**Figure 2-34** Line created with the center point of the sketch

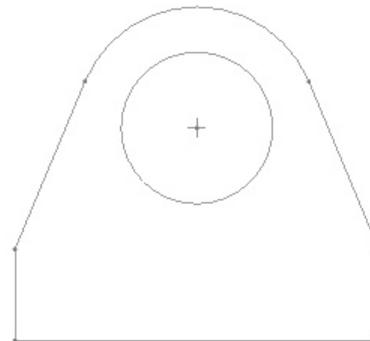


**Figure 2-35** Two circles of radii 2.5 and 4

16. Choose the **Line/Arc Chain** tool from the **Sketch** group of the **Home** tab; you are prompted to specify the start point of the line.
17. Click on the point created in step 5; you will be prompted to specify the next point of the line.
18. Hover the cursor on the circle of radius 4 and click when the tangent symbol appears; the tangent line is created.
19. Similarly, draw the other tangent line, refer to Figure 2-36.
20. Choose the **Trim Segment** tool available in the **Edit Sketch** group of the **Home** tab; you are prompted to select entities to trim.
21. Select the unwanted entities to trim; the sketch of the model is created, as shown in Figure 2-37.



**Figure 2-36** Tangent lines created



**Figure 2-37** Sketch of the model created after trimming the unwanted entities

### Saving the Sketch

1. Choose the **Save** tool from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the sketch that you had entered earlier.
2. Choose the **OK** button; the sketch is saved.
3. After saving the sketch, choose the **Close** button from the **Quick Access Toolbar**.



## Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 2-38. The sketch is shown in Figure 2-39. For your reference, all entities in the sketch are labeled alphabetically. **(Expected time: 30 min)**

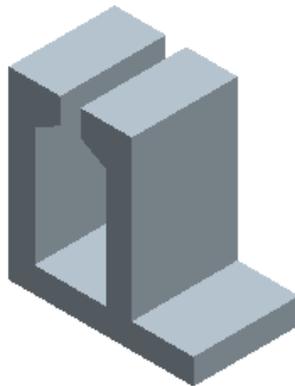


Figure 2-38 Model for Tutorial 2

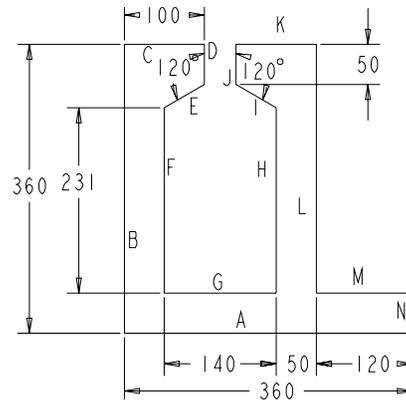


Figure 2-39 Sketch of the model

The following steps are required to complete this tutorial:

- a. Set the working directory and create a new part file.
- b. Draw the sketch, refer to Figure 2-39.
- c. Save the sketch and close the file.

### Setting the Working Directory

The working directory was selected in Tutorial 1, therefore you do not need to select the directory again. But, if a new session of Creo Direct is started, you need to set the working directory again by following the steps given next.

1. Open the Navigator (if it is in collapsed state) by clicking on the **Toggle the display of the navigation area** button on the bottom left corner of the Creo Direct main window; the Navigator is displayed. At the bottom of the Navigator, the **Folder Tree** is displayed in the **Folder Browser** tab. Click on the **Folder Tree** button; the **Folder Tree** expands.



2. Click on the right arrow adjacent to the *CreoDirect2.0* folder in the Navigator; the contents of the *CreoDirect2.0* folder are displayed.
3. Now, right-click on the *c02* folder; a shortcut menu is displayed. From the shortcut menu, choose the **Set Working Directory** option; the working directory is set to *c02*.
4. Close the Navigator by clicking on the **Toggle the display of the navigation area** button located at the bottom left corner of the main window; the Navigator slides in.

### Starting a New Object File

1. Choose the **New** tool from the **Quick Access Toolbar**; the **New** dialog box is displayed. Select the **Part** button from the **Type** area of the **New** dialog box; the default name of the sketch appears in the **Name** edit box. 
2. Enter **c02tut02** in the **Name** edit box and choose **OK**; the sketcher environment of the **Part** mode is invoked.

### Drawing the Sketch

The sketch in Figure 2-38 consists of only lines. For understanding, all the lines in the sketch are labeled alphabetically.

1. Choose the **Line/Arc Chain** tool from the **Line** drop-down of the **Sketch** group; you are prompted to specify the start point of the line. 
2. Choose the **Sketch Plan View** tool from the **In-graphics** toolbar to make the sketching plane parallel to the screen. 
3. Specify **360** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. After specifying the coordinates, press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
6. Next, move the cursor horizontally in left direction to create the line A, refer to Figure 2-32. You will notice that as you move the cursor, the horizontal guide symbol is displayed with the line. Move the cursor toward the left and enter **360** for specifying the endpoint of the line.
7. Next, move the cursor vertically upward so that the vertical guide appears on the line. Enter **360** in the dynamic input to specify the endpoint of line B; the line B is created.
8. Move the cursor to the right direction in the drawing area and enter **100** in the dynamic input to specify the endpoint of line C.

9. Now, to draw line D, move the cursor in vertical downward direction and enter **50** in the dynamic input to specify the endpoint of line D.
10. Next, move the cursor in the South-West, you will notice that in the dynamic input the Length and angle edit boxes will be displayed.
11. Press the SPACE key two times; the Length and Angle edit boxes will be changed to X and Y coordinate edit boxes.
12. Specify the value of X and Y coordinates as **50, 281** and press ENTER; the line E is created.
13. Move the cursor in vertical downward direction and specify the length of line as **231**; the line F is created.
14. Move the cursor in horizontally right direction and specify the length of line as **140**; the line G is created.
15. Move the cursor in vertically upward direction and specify the length of line as **231**; the line H is created.
16. Next, move the cursor in North-West direction, you will notice that in the dynamic input the Length and angle edit boxes will be displayed.
17. Press the SPACE key two times; the Length and Angle edit boxes will be changed to X and Y coordinate edit boxes.
18. Specify the value of X and Y coordinates as **140, 310** and press ENTER; the line I is created.
19. Move the cursor in vertically upward direction and specify the length of line as **50**; the line J is created.
20. Move the cursor in horizontally right direction and specify the length of line as **100**; the line K is created.
21. Move the cursor in vertically downward direction and specify the length of line as **310**; the line L is created.
22. Move the cursor in horizontally right direction and specify the length of line as **120**; the line M is created.
23. Next, move the cursor in vertically downward direction and specify the length of line as **50**; the line N is created.

### Saving the Sketch

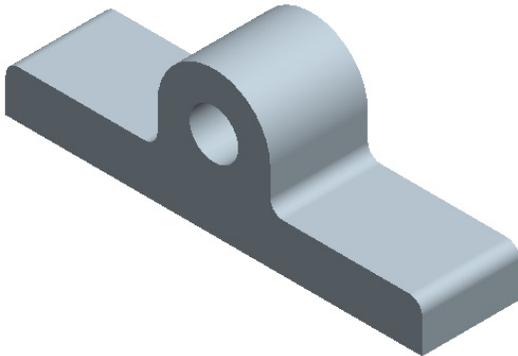
Now, you need to save the sketch because you may need the sketch to create a 3D model.

1. Choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name that you had specified earlier.
2. Choose the **OK** button; the sketch is saved.

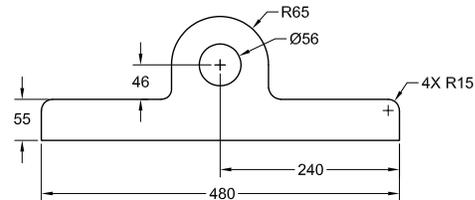


### Tutorial 3

In this tutorial, you will draw the sketch of the model shown in Figure 2-40. The sketch is shown in Figure 2-41. **(Expected time: 30 min)**



*Figure 2-40 Model for Tutorial 3*



*Figure 2-41 Sketch of the model*

The following steps are required to complete this tutorial:

- a. Set the working directory and create a new part file.
- b. Draw the sketch using sketcher tools, refer to Figure 2-42.
- c. Apply fillets at two corners of the sketch, refer to Figure 2-44.
- d. Save the sketch and exit the **Sketch** mode.

### Setting the Working Directory

The working directory was selected in Tutorial 1, and therefore there is no need to select the working directory again. But if a new session of Creo Direct is started, then you have to set the working directory again by following the steps given next.

1. Open the Navigator by clicking on the **Toggle the display of the navigation area** button on the left corner of the Creo Direct window; the Navigator slides out. In the Navigator, the **Folder Tree** is displayed at the bottom. Click on the black arrow that is available at the right-side of the **Folder Tree**; the **Folder Tree** expands.
2. Click on the node adjacent to the *CreoDirect2.0* folder in the Navigator to display the context of this folder.

3. Now, right-click on the *c02* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; *c02* is set as the working directory.
4. Close the Navigator by clicking on the **Toggle the display of the navigation area** button; the Navigator slides in.

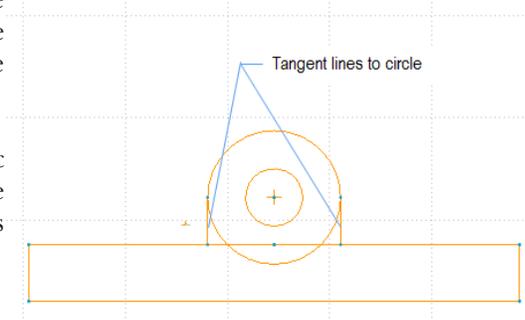
### Starting a New Object File

1. Start a new object file in the **Part** mode. Name the file as *c02tut3*.

### Drawing the Sketch

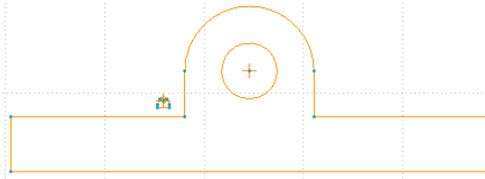
1. Choose the **Line/Arc Chain** tool from the **Line** drop-down of the **Sketch** group; you are prompted to specify the start point of the line. 
2. Choose the **Sketch Plan View** tool from the **In-graphics** toolbar to make the sketching plane parallel to the screen. 
3. Specify **-240** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **55** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. After specifying the coordinates, press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
6. Next, move the cursor in vertical downward direction and enter **55** in the dynamic input to specify the second point.
7. Next, move the cursor in horizontally right direction and enter **480** in the dynamic input to specify the third point.
8. Next, move the cursor in upward direction and enter **55** in dynamic input to specify the fourth point. Next, join the fourth and first point of the sketch.
9. Choose the **Center and Point** tool from the **Circle** drop-down of the **Sketch** group in the **Home** tab; you are prompted to specify the center of the circle. 
10. Specify the coordinates for center of the circle as **0,101**; you will be prompted to specify the radius of the circle.
11. Specify the radius of the circle as **28**; the circle is created. Similarly, create the other circle with center point and radius as **0,101** and **65**, respectively.

12. Choose the **Tangent** tool from the **Line** drop-down of the **Sketch** group in the **Home** tab; you are prompted to select the entity to which the tangent line is to be drawn.
13. Click on the circle of radius 65; the dynamic input is displayed with Length and Angle edit boxes. Also, a tangent symbol is displayed with the point specified.
14. Move the cursor in the vertical downward direction and join the horizontal line; the tangent line is created.

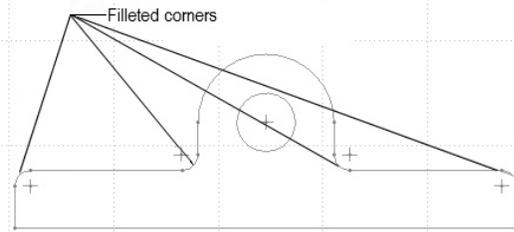


*Figure 2-42 Tangents created*

15. Similarly, create the other tangent line, refer to Figure 2-42.
16. Choose the **Trim Segment** tool available in the **Edit Sketch** group of the **Home** tab and trim the unwanted entities, as shown in Figure 2-43.
17. Choose the **Fillet** tool available in the **Edit Sketch** group of the **Home** tab; you are prompted to select reference entities to create the fillet.
18. Apply fillet to the corners of the sketch with fillet radius as **15**. Figure 2-44 shows the sketch of the model after applying the fillet.



*Figure 2-43 Sketch after trimming the unwanted entities*



*Figure 2-44 Sketch of the model after creating the fillets*

### **Saving the Sketch**

Now, you can save the sketch because you may need the sketch to create a 3D model.

1. Choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the sketch that you had entered earlier.
2. Choose the **OK** button; the sketch is saved.



3. After saving the sketch, choose the **Close** button from the **Quick Access Toolbar**.



## Self-Evaluation Test

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

1. When you draw a sketch, the guides are also displayed by default. (T/F)
2. In Creo Direct, you can create lines that are tangent to two circles. (T/F)
3. You can edit the sketch entities from the **Model Tree**. (T/F)
4. You can measure the number of sketch entities at a time. (T/F)
5. While drawing a circle, first you need to specify its diameter. (T/F)
6. The **Trim** tool lies in the \_\_\_\_\_ group of the **Ribbon**.
7. A sketch can be modified by changing its \_\_\_\_\_.
8. The **Model Tree** is on by default when you enter the **Part** mode. (T/F)
9. In the **Part** mode, the tangent guide is represented by a \_\_\_\_\_ symbol.
10. The file created in the **Part** mode is saved with a \_\_\_\_\_ file extension.

## Review Questions

Answer the following questions:

1. You can dynamically modify the geometry of a sketch. (T/F)
2. You can use the **Rectangle** button from the **Sketch** group to draw a square. (T/F)
3. The tools available in the \_\_\_\_\_ group are used to measure on entity manually.
4. You cannot undo a previous operation in the sketcher environment. (T/F)

## Exercises

### Exercise 1

In this exercise, you will draw the sketch of the model shown in Figure 2-45. The sketch is shown in Figure 2-46. **(Expected time: 30 min)**

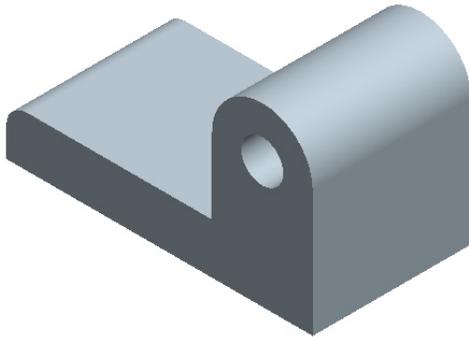


Figure 2-45 Solid model for Exercise 1

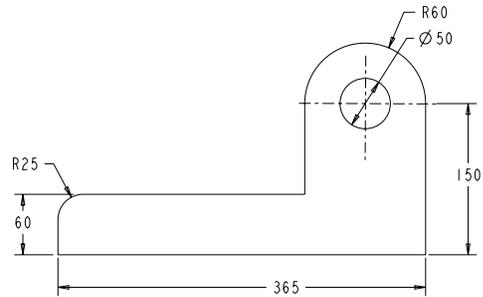


Figure 2-46 Sketch of the model

### Exercise 2

In this exercise, you will draw the sketch of the model shown in Figure 2-47. The sketch is shown in Figure 2-48. **(Expected time: 30 min)**

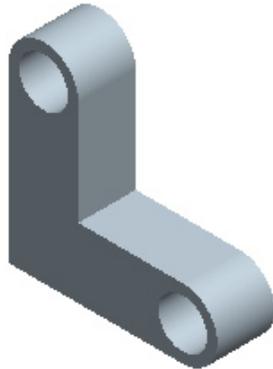


Figure 2-47 Solid model for Exercise 2

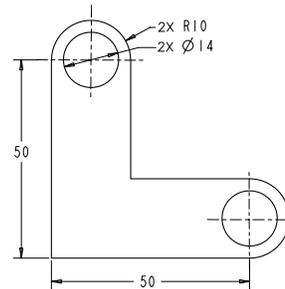


Figure 2-48 Sketch of the model

### Exercise 3

In this exercise, you will draw the sketch of the model shown in Figure 2-49. The sketch is shown in Figure 2-50. **(Expected time: 30 min)**

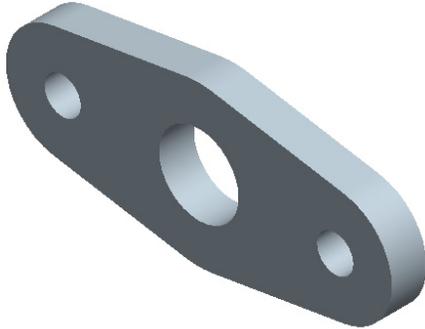


Figure 2-49 Solid model for Exercise 3

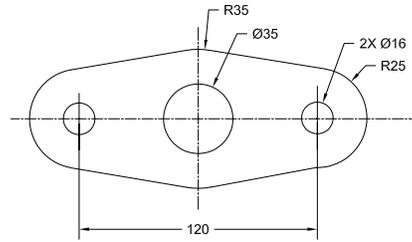


Figure 2-50 Sketch of the model

### Exercise 4

In this exercise, you will draw the sketch of the model shown in Figure 2-51. The sketch is shown in Figure 2-52. **(Expected time: 30 min)**

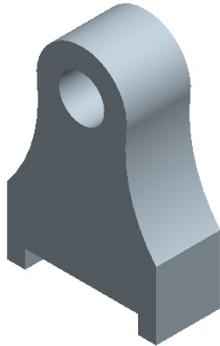


Figure 2-51 Solid model for Exercise 4

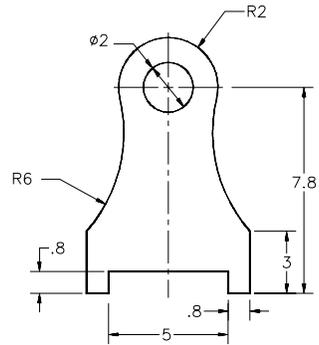


Figure 2-52 Sketch of the model

### Exercise 5

In this exercise, you will draw the sketch of the model shown in Figure 2-53. The sketch is shown in Figure 2-54. **(Expected time: 30 min)**

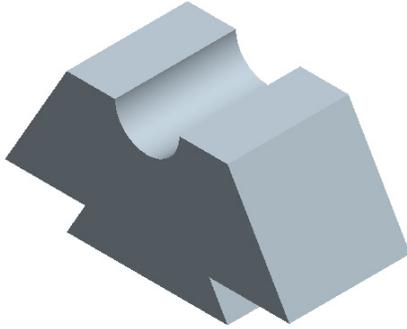


Figure 2-53 Solid model for Exercise 5

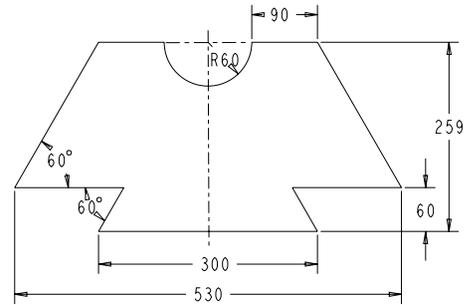


Figure 2-54 Sketch of the model

### Exercise 6

In this exercise, you will draw the sketch of the model shown in Figure 2-55. The sketch is shown in Figure 2-56. **(Expected time: 30 min)**

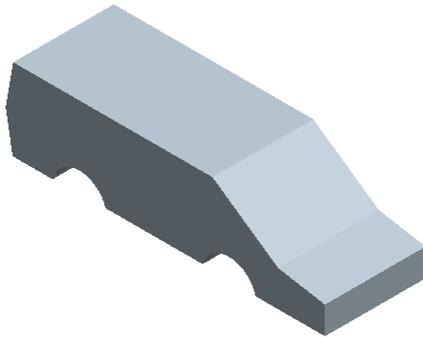


Figure 2-55 Solid model for Exercise 6

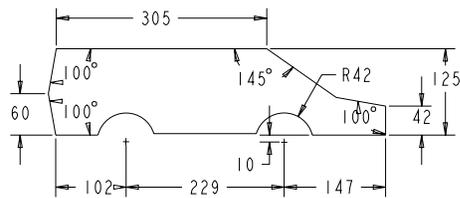


Figure 2-56 Sketch of the model

### Answers to Self-Evaluation Test

1. F, 2. F, 3. T, 4. T, 5. F, 6. Edit Sketch, 7. dimensions, 8. T, 9. T, 10. .prt