

# Chapter 1

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## Creating Sketches in the Sketch Mode-I

### Learning Objectives

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

- Use various tools to create a geometry.
- Dimension a sketch.
- Apply constraints to a sketch.
- Modify a sketch.
- Use Modify Dimensions dialog box.
- Edit geometry of a sketch by trimming.
- Mirror a sketch.
- Use drawing display options.



## THE SKETCH MODE

Almost all models designed in Pro/ENGINEER Wildfire 2.0 consists of datums, sketched features, and placed features. For creating datums and placed features, you do not need to draw sketches. However, to create a three-dimensional (3D) feature, it is necessary to draw its two-dimensional (2D) sketch. When you enter the **Part** mode and select the options to create any sketched feature, the system automatically takes you to the sketcher environment. In the sketcher environment, the sketch of the feature is created, dimensioned, and constrained. The sketches created in the **Sketch** mode are stored in the .sec format. Then you return to the **Part** mode to create the required feature.



### Note

*You will learn about datums and placed features in later chapters.*

In Pro/ENGINEER, a sketch can be drawn in the **Sketch** mode or in the sketcher environment. A designer can draw a 2D sketch of the product and assign the required dimensions and constraints to it. By assigning the dimensions, the designer can make sure that the 2D sketch of the product or model is satisfying the necessary conditions. He can then continue to create the 3D model of the product in the **Part** mode.

## Using the Sketch Mode

To create any section in the **Sketch** mode of Pro/ENGINEER Wildfire 2.0, certain basic steps have to be followed. The following steps outline the procedure to use the **Sketch** mode:

### 1. Sketch the required section geometry

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

### 2. Add the constraints and dimensions to the sketched section

While sketching the section geometry, weak constraints and dimensions are automatically added to the section. The sketch can also be dimensioned and constrained manually. After adding the dimensions you can modify them as required.

### 3. Add relations to the sketch if needed

The geometry of the sketch can be controlled by adding relations.

### 4. Regenerate the section

If the sketch is fully dimensioned and constrained, the sketch is automatically regenerated. Throughout this book, it is assumed that you are sketching in the **Sketch** mode with the **Intent Manager** on. Pro/ENGINEER has the capability to analyze the section, and if the section is not complete for any reason, the section will not be regenerated. You will learn about these reasons as you go through this chapter.



**Tip:** *Throughout this book, sketcher environment is referred to the environment in Pro/ENGINEER where you can draw 2D geometries. Apart from the **Sketch** mode, the sketcher environment can be accessed in other modes of Pro/ENGINEER also.*

## Invoking the Sketch Mode

To invoke the **Sketch** mode, choose **New** from the **File** menu or choose the **Create a new object** button from the **File** toolbar. The **New** dialog box is displayed with available Pro/ENGINEER modes. Select the **Sketch** radio button to start a new file in the **Sketch** mode, see Figure 1-1. When you select the **Sketch** radio button, a default name of the sketch file appears in the **Name** edit box. You can change the sketch name as required and then choose the **OK** button to enter the **Sketch** mode.

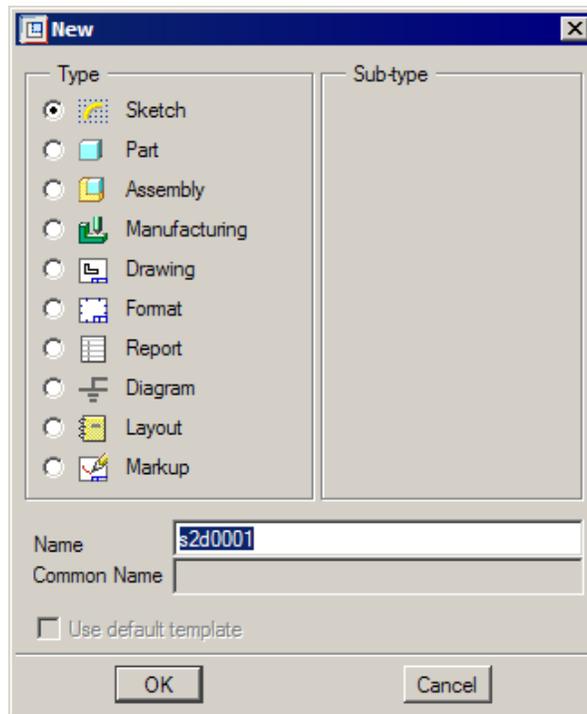


Figure 1-1 The New dialog box

## THE SKETCHER ENVIRONMENT

When you invoke the **Sketch** mode, the initial screen appearance is similar to the one shown in Figure 1-2. This figure also shows the **Sketcher Tools** toolbar that is displayed on the right side of the graphics window. The buttons available in this toolbar are used to draw sketches. The drawing tools are also available in the **Sketch** menu in the menu bar. When you enter the sketcher environment, the **Intent Manager** is on by default. Also, when you are in the selection mode, shortcut menus can be invoked by holding down the right mouse button in the graphics window. The options in these shortcut menus vary depending on the item selected. These shortcut menus also contain the tools to draw sketches.



### Note

The selection mode in the sketcher environment is discussed later in this chapter.

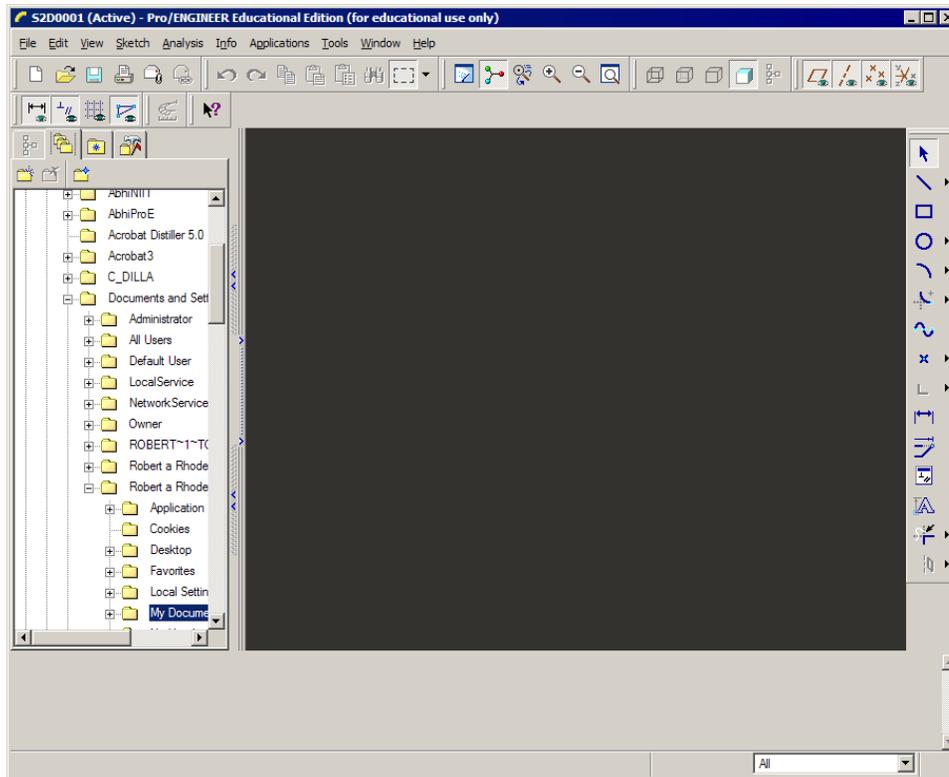


Figure 1-2 Initial screen appearance in the **Sketch** mode

The Navigator is displayed on the left side of the graphics window. It covers a part of the graphics window and therefore the drawing area is decreased. You can make the Navigator slide in by clicking the Navigator sash. Now, the area available for sketching is increased.



**Note**

*The functions of Navigator are discussed in the Introduction chapter.*

## WORKING WITH THE SKETCH IN THE SKETCH MODE

When you invoke the sketcher environment, the **Select items** button is chosen by default. If this button is chosen, the sketcher environment is said to be in the selection mode. In the selection mode, you can select entities of the sketch to edit or to invoke the shortcut menu. The options in the shortcut menu can be used to apply various operations on the selected item.



**Note**

*The sketch is saved with a .sec file extension. While drawing a sketch in the part file, if you save the sketch in the sketcher environment, this sketch is also saved with the .sec file extension.*

**Note**

You can create a simple sketch by using the options available in the shortcut menu. To invoke the shortcut menu, hold down the right mouse button in the graphics window. Note that, once the shortcut menu is displayed, the right mouse button can be released.

## DRAWING A SKETCH USING THE SKETCHER TOOLS TOOLBAR

When you are in the sketcher environment, the **Sketcher Tools** toolbar that is available on the **Right Toolchest**, which contains the tools to draw a sketch, dimension it, and modify the dimensions. In this section, you will learn how to draw sketched entities using the tools available in the **Sketcher Tools** toolbar.

### Placing a Point



Points are generally used for dimensioning the vertices that are removed while applying fillets. For example, if the sketch is to be dimensioned using these vertices, you need to place points on them before applying fillets. Now, since a point is placed at the vertex, you can easily use it for dimensioning the sketch. The following steps explain the procedure to sketch a point:

1. Choose the **Create points** button from the **Sketcher Tools** toolbar; the system prompts you to select a location for the point.
2. As soon as you select the location of the point by clicking, the point is placed at the desired location in the graphics window.

**Note**

To increase the number of visible command prompt lines in the **Message Area**, select the top sash of the **Message Area** using the left mouse button and drag it upwards, toward the screen.

*When you draw a single point no dimensions appear. But, when you draw two points, they are dimensioned with each other.*

### Drawing a Line

To draw lines, there are three tool buttons available in the **Sketcher Tools** toolbar. To view these buttons, choose the black arrow on the right of the **Create 2 point lines** button; the flyout appears with three tool buttons. The first button is **Create 2 point lines**. This button is used to create a line by selecting two points in the graphics window. The second button on the flyout is **Create lines tangent to 2 entities**. This button is used to create a tangent between two entities. The third button is **Create 2 point centerlines**. This button is used to create a centerline by selecting two points in the graphics window. The centerline is used for creating revolved features, mirroring, and so on.

The procedure to create lines using these tools are discussed next.

## Drawing a Line Using the Create 2 point lines Tool



The following steps explain the procedure to create a line using the **Create 2 point lines** tool:

1. Choose the **Create 2 point lines** button. Click on graphics window to start the line; a rubber-band line appears from the selected point with the other end attached to the cursor. The symbols **V** and **H** that appear while drawing the vertical and horizontal lines are called constraints. Constraints are discussed in detail later in this chapter.
2. The system prompts you to specify the endpoint. Move the cursor in the graphics window to a desired location and click to specify the endpoint of the line; the line appears in yellow color. The rubber-band line continues and you can draw the second line.
3. Repeat step 2 until all lines are drawn. You can end the line creation by pressing the middle mouse button. To abort line creation, use the middle mouse button.



### Note

*When you draw a single line, the color of the line after you have drawn it is red. If you draw multiple lines, the color is yellow.*

*After drawing a line, when you press the middle mouse button to end the line creation, the line drawn is highlighted in red color. In the sketcher environment, the red color of an entity indicates that it is selected. If you press the DELETE key, the line will be erased from the graphics window. After drawing a line, weak dimensions in gray color are applied to the sketch. The weak dimensions are applied automatically to the sketched entities as you draw them.*

## Drawing a Line Using the Create lines tangent to 2 entities Tool



The **Create lines tangent to 2 entities** button is used to draw a tangent between two entities such as arcs, circles, splines, or a combination of these. The following steps explain the procedure to draw a tangent using this tool:

1. Choose the arrow on the right of the **Create 2 point lines** button and then choose the **Create lines tangent to 2 entities** button from the **Sketcher Tools** toolbar. You are prompted to select start location on arc or circle.
2. Select the first entity from where the tangent line will be drawn; the color of the selected entity changes to red. Now, you are prompted to select the end location on arc or circle. As soon as you select the second entity, a line that is tangent to both the selected entities is drawn.



### Note

*Whenever you are prompted to select an entity in the sketcher environment, the SELECT dialog box is displayed. You can ignore this dialog box because it appears automatically and disappears without any confirmation.*

### Drawing a Centerline Using the Create 2 point centerlines Tool



You can draw horizontal, vertical, or inclined centerlines using the **Create 2 point centerlines** button. This button is available in the flyout that is displayed when you choose the black arrow on the right of the **Create 2 point lines** button. The centerline in a sketch is used as an axis of rotation, for mirroring, aligning, and dimensioning entities.

The following steps explain the procedure to draw a centerline:

1. In the **Sketcher Tools** toolbar, choose the black arrow on the right of the **Create 2 point lines** button and then choose the **Create 2 point centerlines** button; you are prompted to select the start point.
2. Click in the graphics window to specify the start point. Now, you are prompted to select the end point.
3. Click in the graphics window to specify the endpoint. A centerline of infinite length is drawn.

### Drawing a Rectangle



The following steps explain the procedure to sketch a rectangle using the **Create rectangle** tool:

1. Choose the **Create rectangle** button from the **Sketcher Tools** toolbar; you are prompted to select two points to indicate the diagonal of box. Click to specify the first point; a yellow rubber-band box appears with the cursor attached to the opposite corner of the box.
2. Move the cursor to the desired location in the graphics window to size the diagonal of the rectangle. Click to specify the second point for the diagonal of the rectangle.

### Drawing a Circle

In the **Sketcher Tools** toolbar, there are four tools to draw a circle and one tool to draw an ellipse. To view the tools available to draw circles and ellipses, choose the black arrow on the right of the **Create circle by picking the center and a point on the circle** button. The flyout appears with five buttons. The procedures to create circle and ellipse using the various tools are discussed next.

### Drawing a Circle Using the Create circle by picking the center and a point on the circle Tool



As the name suggests, the **Create circle by picking the center and a point on the circle** tool is used to draw a circle by specifying the center of the circle and a point on it. The following steps explain the procedure to draw a circle using this tool.

1. Choose the **Create circle by picking the center and a point on the circle** button; you are prompted to select the center of a circle.
2. Click in the graphics window to specify the center point of the circle; you are prompted to select a point on the circle. A yellow rubber-band circle appears with the center at the specified point and the cursor attached to its circumference.
3. Move the cursor to size the circle. Click to complete the circle creation; you are again prompted to select the center of the circle.
4. Repeat steps 2 and 3 until you have drawn all required circles. If you want to abort circle creation, press the middle mouse button.

### Drawing a Construction Circle

A construction circle is a circle that is used to align entities, create diametrical or radial dimensioning, and to reference entities. Figure 1-3 shows an application of construction circle. 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.

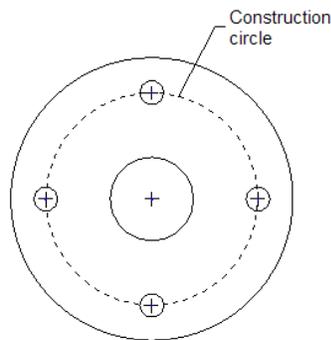


Figure 1-3 Sketch of a flange

To create a construction circle, draw a circle or select a previously drawn circle. Then, hold down the right mouse button on it to invoke the shortcut menu, as shown in Figure 1-4. Choose the **Construction** option from the shortcut menu. The circle appears yellow in color and with a dashed line style, indicating that it is a construction circle.

### Drawing a Circle Using the Create concentric circle Tool



The following steps explain the procedure to draw a concentric circle using the **Create concentric circle** tool:

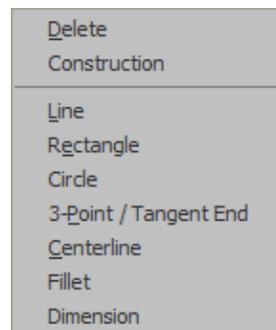


Figure 1-4 Construction option in the shortcut menu



**Tip:** To convert a construction circle back to a solid entity, select the construction circle and hold down the right mouse button to invoke the shortcut menu. Choose the **Solid** option from this shortcut menu.

1. Choose the **Create concentric circle** button from the flyout in the **Sketcher Tools** toolbar; you are 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 to determine the concentricity of the circle to be drawn. Move the mouse to size the circle.
3. After sizing the circle, finish circle creation by clicking and then pressing the middle mouse button to end circle creation.

### Drawing a Circle Using the Create circle by picking its 3 points Tool



The following steps explain the procedure to draw a circle using the **Create circle by picking its 3 points** tool:

1. Choose the **Create circle by picking its 3 points** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to specify the first point on the circle.
2. Click for the first point at the desired location in the graphics window; you are prompted to select the second point on the circle. Move the cursor and click to select the second point in the graphics window.
3. As soon as you select the second point, a yellow rubber-band circle appears with the cursor attached to it. You are prompted to select the third point. Move the mouse to size the circle and click to specify the third point; a circle is drawn. You will be again prompted to select first point on the circle to draw the next circle.
4. You can draw another circle or press the middle mouse button to end circle creation or to abort circle creation before the next circle is completed.

### Drawing a Circle Using the Create a circle tangent to 3 entities Tool



The **Create a circle tangent to 3 entities** tool is used to draw a circle that is tangent to 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.

The following steps explain the procedure to draw a circle using the **Create a circle tangent to 3 entities** tool:

1. Choose the **Create a circle tangent to 3 entities** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to select the start location on an arc, circle, or line.
2. Click to select the first entity. The color of the entity changes to red and you are prompted to select an end location on an arc, circle, or line. Select the second tangent entity. Next,

you are prompted to select the third location on an arc, circle, or line. Select the third tangent entity.

As you select the three entities, a circle that is tangent to these three entities is drawn.

3. To end the creation of circle, press the middle mouse button.

### Drawing an Ellipse Using the Create a full ellipse Tool



The following steps explain the procedure to draw an ellipse using the **Create a full ellipse** tool:

1. Choose the **Create a full ellipse** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to specify the center of an ellipse.
2. Click at the desired location in the graphics window to specify the center point; a yellow rubber-band ellipse appears with the cursor attached to the ellipse. Move the cursor in the graphics window to size the ellipse.
3. An ellipse is drawn when you click to specify the second point.

### Drawing an Arc



To draw an arc there are five tools in the **Sketcher Tools** toolbar. To view them, choose the black arrow on the right of the **Create an arc by 3 points or tangent to an entity at its endpoint** button; the flyout appears with five buttons. The procedures to draw arcs using various tool buttons in the flyout are discussed next.

### Drawing an Arc Using the Create an arc by 3 points or tangent to an entity at its endpoint Tool

The **Create an arc by 3 points or tangent to an entity at its endpoint** tool is used to draw arcs tangent from the endpoint of an existing entity or by defining three points in the graphics window.

When you choose this button to draw an arc from an endpoint, the **Target** symbol is displayed as soon as you select the endpoint. This **Target** symbol is in the form of a circle that is divided into four quadrants. The following steps explain the procedure to draw an arc from the endpoint of an existing entity by using this tool:

1. Choose the **Create an arc by 3 points or tangent to an entity at its endpoint** button from the **Sketcher Tools** toolbar; you are prompted to select the start point of the arc.
2. Specify three points in the graphics window to draw an arc. If you want to draw an arc from the endpoint of an existing entity, select the endpoint of that entity. As soon as you select the endpoint, the **Target** symbol appears at the endpoint of the entity. Move the cursor along the tangent direction through a small distance, a yellow rubber-band arc appears with one end attached to the endpoint of the entity and the other end attached to the cursor. Note that when you move the cursor out of the **Target** symbol perpendicular

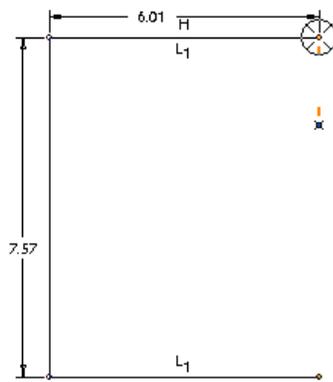
to the endpoint, the arc is drawn by specifying three points. In this case the rubber-band arc does not appear, as shown in Figure 1-5.

On the other hand, if you move the cursor out horizontally from one of the quadrants of the **Target** symbol, the arc is drawn tangent to the endpoint, as shown in Figure 1-6.

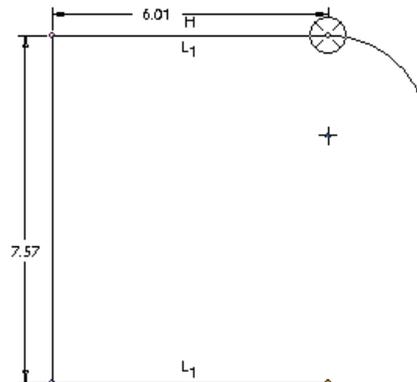
3. Move the cursor to the desired position in the graphics window to size the arc. Use the left mouse button to complete the arc.



**Tip:** If you do not want to draw a tangent arc, move the cursor out of the **Target** symbol perpendicular to the endpoint.



**Figure 1-5** Cursor moved out of **Target** symbol perpendicular to the endpoint



**Figure 1-6** Cursor moved out of the **Target** symbol along the tangent direction.

### Drawing an Arc Using the Create concentric arc Tool



The **Create concentric arc** tool is used to draw an arc concentric to an existing arc. You will have to select an entity to which the arc will be concentric. The entity selected must be an arc or a circle. The following steps explain the procedure to draw an arc using this tool:

1. Choose the **Create concentric arc** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to select an arc to determine the center of the arc to be created.
2. As soon as you select an entity, a dotted circle appears on the screen and you are prompted to select the start point of the arc. After doing so, a yellow rubber-band arc will appear with one end attached to the start point. The length of the arc will change as you move the cursor. Next, you are prompted to select the endpoint of the arc.
3. Click to specify the endpoint; the arc is created.
4. You can continue drawing another arc or end arc creation by pressing the middle mouse button.

### Drawing an Arc Using the Create an arc by picking its center and endpoints Tool



The following steps explain the procedure to draw an arc using the **Create an arc by picking its center and endpoints** tool:

1. Choose the **Create an arc by picking its center and endpoints** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to select the center of the arc.
2. Click to specify a center point for the arc in the graphics window; a yellow colored center mark appears at that point. Now, you are prompted to select the start point of the arc. As you move the cursor, a dotted circle appears and is attached to the cursor.
3. Select the start point of the arc on the circumference of the dotted circle; a yellow rubber-band arc appears from the start point. The length of this arc changes dynamically as you move the cursor.
4. You are prompted to select the endpoint of the arc. Move the cursor to size the arc, and then click to select the endpoint of the arc. An arc is drawn between the two selected points.



#### Note

*You can draw only one arc with one center. If you want to draw another arc you will have to select the center again.*

### Drawing an Arc Using the Create an arc tangent to 3 entities Tool



The **Create an arc tangent to 3 entities** tool is used to draw an arc that is tangent to three selected entities. The following steps explain the procedure to draw an arc using this tool:

1. Choose the **Create an arc tangent to 3 entities** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to select the start location on an arc, circle, or line.
2. As soon as you select the first entity, the color of the entity changes to red. Now, you are prompted to select the end location on an arc, circle, or line.
3. After doing so, you are prompted to select a third location on an arc, circle, or line. Select a third entity. An arc is drawn tangent to the three entities selected.
4. You can continue drawing arcs or press the middle mouse button to abort arc creation.

### Drawing an Arc Using the Create a conic arc Tool



The **Create a conic arc** tool is used to draw a conic arc. The following steps explain the procedure to draw a conic arc using this tool:

1. Choose the **Create a conic arc** button from the flyout in the **Sketcher Tools** toolbar; you are prompted to specify the first endpoint of the conic entity.

2. Specify a point in the graphics window; you are prompted to specify the second endpoint of the conic entity.
3. Specify the second endpoint; a centerline is drawn between the two points. Now, you are prompted to specify the shoulder point of the conic. Specify a point on the screen; the conic arc is drawn.



**Note**

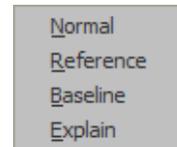
*If you delete the centerline of the conic arc, the arc will not be deleted.*

*Remember that if the conic arc is the only entity in the graphics window, then you cannot delete its centerline.*

## DIMENSIONING THE SKETCH

After you draw a sketch, the next step involves the dimensioning of the sketch. The basic purpose of dimensioning in Pro/ENGINEER is to control the size of the sketch and to locate it with some reference. In Pro/ENGINEER, a sketch cannot be regenerated unless it is fully dimensioned and constrained. The phrase “the sketch cannot be regenerated” means that the sketch cannot be accepted by Pro/ENGINEER.

By default, the sketched entities are dimensioned and constrained automatically while sketching or as soon as you are done with the sketch. However, sometimes you need to add additional dimensions to the sketch. The **Create defining dimension** button in the **Sketcher Tools** toolbar is used to manually dimension the entities. You can also choose **Sketch > Dimension** from the menu bar to display a cascading menu, as shown in Figure 1-7. This menu contains different dimensioning options.



*Figure 1-7  
Dimensioning options*



**Note**

*When the intent manager is off, the dimensions and constraints are not automatically applied. You need to manually add the dimensions by choosing the **Dimension** option from the **SKECHER** menu.*

### Dimensioning a Sketch Using the Create defining dimension button or the Normal Option



The **Create defining dimension** button or the **Normal** option are used for normal dimensioning of the sketch. The following steps explain the procedure to dimension a sketch:

1. Choose the **Create defining dimension** button from the **Sketcher Tools** toolbar. Click on the entity you want to dimension; the color of the entity changes from yellow to red.
2. Move the cursor and place the dimension at the desired place by pressing the middle mouse button. You can modify the dimension values using the modifying options discussed later in this chapter.

The remaining options in the cascading menu are not used while sketching and therefore, they are discussed in Chapter 2.

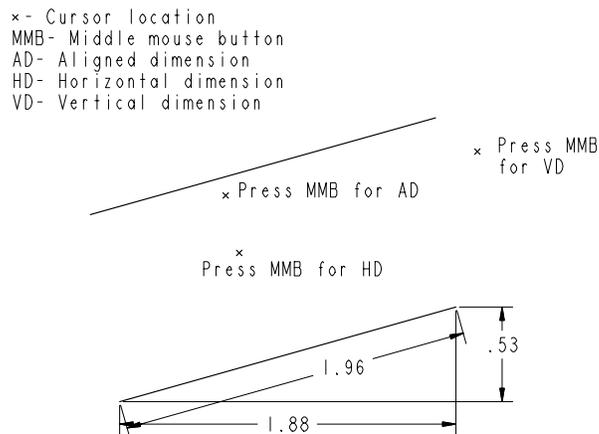
## DIMENSIONING THE BASIC SKETCHED ENTITIES

Choose the **Create defining dimension** button and follow the procedures given below to dimension the sketched entities.

### Linear Dimensioning a Line

You can dimension a line by selecting its endpoints or by selecting the line. After selecting the two endpoints or the line, press the middle mouse button to place the dimension. If the line is inclined and you select the two endpoints to dimension, then the location where you press the middle mouse button is important, because it defines the orientation of the dimension that is displayed on the screen.

Figure 1-8 explains the three possible orientations of dimension that can be displayed when you dimension a line.



*Figure 1-8 Approximate locations of the cursor to achieve different dimensions*



#### Note

*It is not possible to dimension a line in the three orientations at the same time in the sketcher environment. The dimensions in Figure 1-8 are only for explanation.*

### Angular Dimensioning of an Arc

To add angular dimension to an arc, select both ends of the arc by pressing the left mouse button and then select a point on the arc. Next, place the dimension at the desired point by pressing the middle mouse button. The dimension appears, as shown in Figure 1-9. You can modify the dimension using tools that are discussed later.

### Diameter Dimensioning

For diameter dimensioning, click on a circle twice. Then place the dimension at the desired location by pressing the middle mouse button. The diameter dimension is displayed, as shown in Figure 1-10. The same diameter dimensioning technique can also be used for arcs.

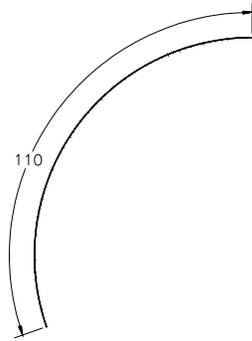


Figure 1-9 Angular dimensioning

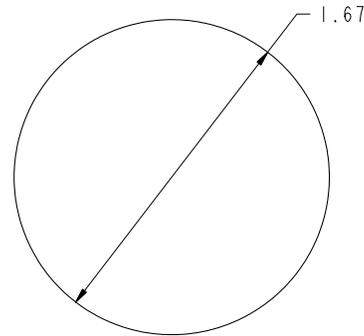


Figure 1-10 Diameter dimensioning

### Radial Dimensioning

For radial dimensioning, click on the entity once. Then place the dimension at the desired location by pressing the middle mouse button. The radial dimension is displayed, as shown in Figure 1-11.

### Dimensioning Revolved Sections

Revolved sections are used to create revolved features such as flanges, couplings, and so on. To dimension a revolved section, click on the entity to be dimensioned. Next, select the centerline about which you want the section to be revolved. Once again select the original entity that you want to dimension. Now, place the dimension at the desired location by pressing the middle mouse button. The dimension is displayed, as shown in Figure 1-12. This dimension represents the diameter of a revolved section.

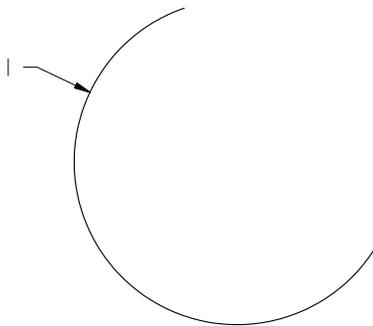


Figure 1-11 Radial dimensioning

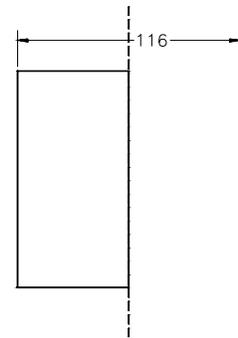


Figure 1-12 Dimensioning for revolved sections



**Tip:** To add dimension to a revolved section, you can also first select the centerline, next the entity to dimension, and then again the centerline.

## WORKING WITH CONSTRAINTS

In Pro/ENGINEER, the entities in a sketch have to be fully specified in terms of size, shape, orientation, and location. This is achieved by setting constraints. Using constraints in the sketch reduces the number of dimensions in that sketch.

Constraints are the logical operations that are performed on the selected geometry to make it more accurate in defining its position with respect to the other geometry. For example, if a line is nearly parallel to another line, Pro/ENGINEER snaps the parallel line and displays the parallel constraint symbol. Now, if you confirm the line creation, the line is drawn parallel to the other line. You can also apply constraints manually.

There are two types of constraints in Pro/ENGINEER, **Geometry** constraints and **Assembly** constraints. Here, you will learn about the **Geometry** constraints and the **Assembly** constraints will be discussed in later chapters.



To apply constraints manually, choose the **Impose sketcher constraints on the section** button from the **Sketcher Tools** toolbar to display the **Constraints** dialog box. This dialog box is shown in Figure 1-13.

This dialog box is used to apply constraints manually. Although the constraints are applied automatically as you draw the sketch, you can use this dialog box if you want to manually apply additional constraints. The constraints that are applied automatically are weak constraints and they appear in gray color. Weak constraints can be made strong and this is discussed later in this chapter. The constraints in the **Constraints** dialog box are discussed next.

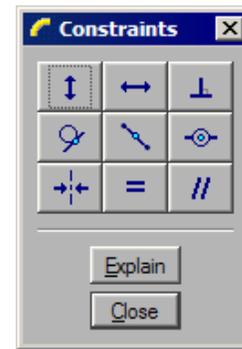


Figure 1-13 The Constraints dialog box

### Make a line or two vertices vertical



This constraint forces the selected line segment to become a vertical line. This constraint also forces the two vertices to be placed along a vertical line.

### Make a line or two vertices horizontal



This constraint forces the selected line segment or two vertices that are apart by some distance to become horizontal or to lie in a horizontal line.

### Make two entities perpendicular



This constraint forces the selected entity to become normal to another selected entity.

### Make two entities tangent



This constraint forces the two selected entities to become tangent to each other.

### Place point on the middle of the line



This constraint forces a selected point or vertex to lie on the middle of a line.

## Create same points, points on entity or collinear constraint



This constraint performs three functions. This constraint can be used to force the two selected points to become coincident to constrain a point on the selected entity, and to make two selected entities collinear, so that they lie on the same line. This constraint aligns two vertices or entities.

## Make two points or vertices symmetric about a centerline



This constraint makes a section symmetrical about the centerline. When you select this constraint, you are prompted to select a centerline and two vertices to make them symmetrical.

## Create Equal Lengths, Equal Radii, or Same Curvature constraint



This constraint forces any two selected entities to become equal in dimension. When you select this constraint, you are prompted to select two lines to make their lengths equal, or you are prompted to select two arcs, circles, or ellipses to make their radii equal.

## Make two lines parallel



This constraint is used to force two lines to become parallel. When selected, this constraint prompts you to select two entities that you want to make parallel.

## Explain Option

The **Explain** option of the **Constraints** dialog box provides information about the constraints that are applied to a sketch. The constraints in the sketch are displayed as symbols. When you choose the **Explain** button, you are prompted to select the constraint or dimension on which you want the explanation. Select the symbol using the left mouse button. The information about the selected constraint is displayed in the **Message Area**.



### Note

*This option is generally helpful when you view a sketch drawn by some other person. By using the **Explain** option you can obtain information about the various constraints applied in the sketch.*

## Disabling the Constraints

The need to disable a constraint arises when you are drawing an entity. For example, if you draw a circle at some distance apart from a circle. While drawing it, the system tends to apply the equal radius constraint when the sizes of the two circles become equal. If at this moment you do not want to apply the equal radius constraint, right-click to disable the equal radius constraint. When you right-click to disable a constraint, an orange line / appears across the symbol. To enable the constraint, right-click once again.

## Converting a Weak Constraint into a Strong Constraint

As discussed earlier, when you draw a sketch, some weak dimensions are automatically applied

to the sketch. As you proceed to complete the sketch, these dimensions are automatically deleted from the sketch without any confirmation.

Select a weak dimension or a weak constraint from the graphics window. The selected dimension or constraint is highlighted in red. Press and hold the right mouse button to invoke the shortcut menu, as shown in Figure 1-14. Choose the **Strong** option from the shortcut menu.



*Figure 1-14* Shortcut menu to convert the weak dimensions to strong

You can also choose **Edit > Convert To > Strong** from the menu bar. The color of the selected dimension is changed from gray to yellow, indicating that the selected constraint or dimension is made permanent.

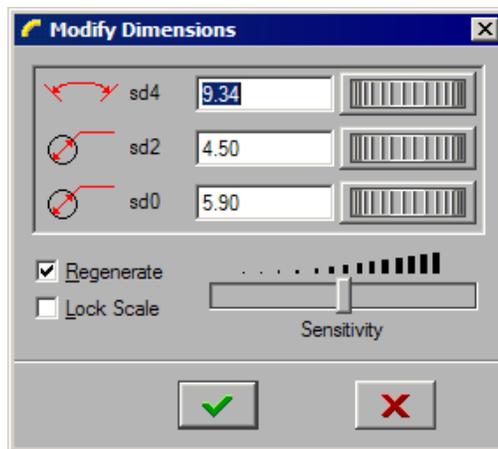
## MODIFYING THE DIMENSIONS OF A SKETCH

There are four ways to modify the dimensions of a sketch. These methods are discussed next.

### Using the Modify the values of dimensions, geometry of splines, or text entities Tool



You can select one or more dimensions from the sketch to modify. When you select dimension(s) from a sketch, they are highlighted in red. If you want to select more than one dimension, hold down the CTRL key and select the dimensions by clicking on them. You can also use CTRL+ALT+A or define a window to select the dimensions in the sketch. Choose the **Modify the values of dimensions, geometry of splines, or text entities** button from the **Sketcher Tools** toolbar to modify the dimensions; the **Modify Dimensions** dialog box is displayed, as shown in Figure 1-15.



*Figure 1-15* The *Modify Dimensions* dialog box

To modify dimensions using this dialog box, you can either enter a value in the edit box or

use the thumbwheel that is available on the right of the edit box. The **Sensitivity** slider is used to set the sensitivity of the thumbwheel.

By default, the **Regenerate** check box is selected and any modifications in the dimensions are automatically updated in the sketch. If you want to delay the modification process of the sketch based on the new value of the selected dimension, you need to clear this check box. If this check box is cleared, the dimensions will not be modified until you exit this dialog box. This means that Pro/ENGINEER allows you to make multiple modifications before updating the sketch.

**Note**

*It is recommended that you clear the **Regenerate** check box and then modify the dimensions if you have to modify more than one dimension.*

The **Lock Scale** check box is used to lock the scale of the selected dimensions. After locking the scale, if you modify any dimension, all other dimensions will also be modified by the same scale.

## Using the Edit Menu

The **Modify** option available in the **Edit** menu in the menu bar can also be used to modify the dimensions. When you choose the **Modify** option from the **Edit** menu, a check mark appears to the left of the **Modify** option in the **Edit** menu. Now, you can select a dimension from the sketch to modify. When you select a dimension, the **Modify Dimensions** dialog box is displayed. By default, the **Regenerate** check box is selected. Therefore, the sketch will be regenerated dynamically as you modify the dimension.

## Modifying a Dimension by Double-Clicking

You can also modify a dimension by double-clicking on it. When you double-click on a dimension, the pop-up text field appears. Enter a new dimension value in this field and press ENTER or use the middle mouse button. Remember that you can select a dimension only when you choose **Select items** button from the **Sketch Tools** toolbar.

## Modifying Dimensions Dynamically

In the sketcher environment, Pro/ENGINEER is always in the selection mode, unless you have invoked some other tool. When you bring the cursor to an entity, the color of the entity changes to cyan. Now, if you hold down the left mouse button, you can modify the entity by dragging the mouse. You will notice that as the entity is modified, the dimensions referenced to the selected entity are also modified.

## RESOLVE SKETCH DIALOG BOX

While applying constraints or dimensions, the system may sometimes prompt you to delete one or more highlighted dimensions or constraints. This is because while adding dimensions or constraints some strong dimensions or constraints conflict with the existing dimensions or constraints. As soon as the conflict occur, the **Resolve Sketch** dialog box is displayed, as shown in Figure 1-16 and the constraints or dimensions under conflict are displayed in red.

When you select a dimension or constraint from the **Resolve Sketch** dialog box, the corresponding dimension or constraint in the graphics window is enclosed in a yellow box. The buttons available in the **Resolve Sketch** dialog box are discussed next.

## Undo

When you choose the **Undo** button, the section is brought back to the state that was just before the conflict occurred.

## Delete

The **Delete** button is used to delete a selected dimension or constraint that is enclosed within the yellow box. Select the dimension or the constraint to delete before you choose the **Delete** button from the **Resolve Sketch** dialog box.

## Dim > Ref

When you choose the **Dim > Ref** button, the selected dimension is converted to a reference dimension.



### Note

*The reference dimensions are used only for reference. They do not participate in feature creation.*

## Explain

When you choose the **Explain** button, the system provides you with information about the selected constraint or dimension. The information is displayed in the **Message Area**.

## DELETING THE SKETCHER ENTITIES

To delete a sketched entity, select it by defining a window. You can specify a window by picking two points so that the entity or entities are enclosed in the window. After specifying the window, the color of the selected entity changes to red. Right-click in the graphics window and hold down the right mouse button until a shortcut menu appears. Now, choose the **Delete** option from this menu to delete the selected item.

You can also delete an item by selecting it and pressing the DELETE key when the selected item turns red in color.

To delete more than one item from the graphics window, press the CTRL key and click to select the entities to be deleted. Press the DELETE key to delete the selected entities. You can also specify a window to select the entities.

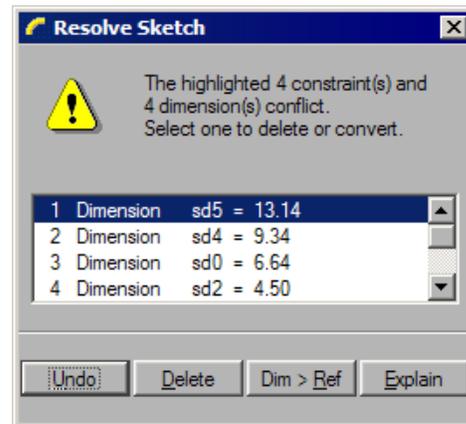


Figure 1-16 The **Resolve Sketch** dialog box



**Note**

It is necessary to be in the selection mode while selecting the items. The term “items” used in this chapter refers to dimensions and entities.

The **Geometry**, **Dimension**, and **Constraint** filters are available in the drop-down list located in the status bar. These filters allow you to select exactly the item that you need to select. This means, if you want to select all constraints in the sketch, choose the **Constraint** filter and specify a window to select. You will notice that only the constraints are selected.



To restore the last deleted item, choose the **Undo** button. This button is available in the **Edit** toolbar on the **Top Toolchest**.

## TRIMMING THE SKETCHER ENTITIES

When 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 tools available for trimming that are available in the **Sketcher Tools** toolbar. You can trim entities using three tools. These tools are discussed next.

### Dynamically trim section entities Tool



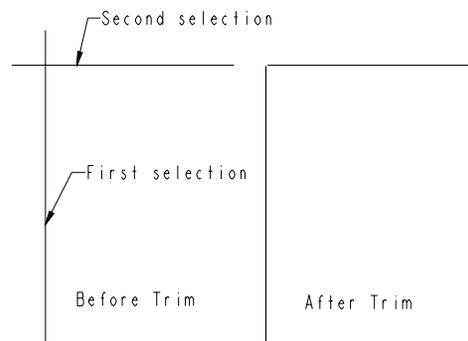
This tool button deletes the selected entities. After choosing the **Dynamically trim section entities** button, when you move the cursor over an entity, the entity is highlighted in cyan color. Press the left mouse button to trim the entity. This tool button also trims entities that extend beyond the point of intersection.

### Trim entities (cut or extend) to other entities or geometry Tool



The **Trim entities (cut or extend) to other entities or geometry** button is used to trim two entities at their corners. Note that when you trim entities using this option, the portion from where you select the entities is retained and the other portion is trimmed. The following steps explain the procedure to trim entities using this tool button:

1. Choose the black arrow on the right of the **Dynamically trim section entities** button to display the flyout. From this flyout, choose the **Trim entities (cut or extend) to other entities or geometry** button; you are prompted to select two entities to be trimmed.
2. Click to select the two entities on the sides you want to keep after trimming, see Figure 1-17. These two entities must be intersecting entities. The entities are trimmed from the point of intersection.



**Figure 1-17** Trimming the lines

## Divide an entity at the point of selection Tool



The **Divide an entity at the point of selection** button is used to divide an entity into any number of parts or entities by specifying points on the entity.

This button is available on the flyout that is displayed when you choose the black arrow that is on the right side of the **Dynamically trim section entities** button.

The following steps explain the procedure to divide an entity:

1. Choose **Divide an entity at the point of selection** button from the flyout; you are prompted to select an entity to be divided.
2. Click to select the entity 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 smaller entities.

## MIRRORING THE SKETCHER ENTITIES



The **Mirror selected entities** button is used to mirror sketched geometries about a centerline. This tool helps to reduce the time used for creation of symmetrical geometries and dimensioning them.

The following steps explain the procedure to mirror a sketched geometry:

1. Sketch a geometry and then sketch a centerline about which you need to mirror the geometry.
2. Select the entities that you need to mirror. The selected entities turn red in color.
3. Choose the **Mirror selected entities** button from the **Sketcher Tools** toolbar. You are prompted to select the centerline about which you need to mirror, hence do so. The selected entities are mirrored about the centerline.



**Tip:** *In case of symmetrical parts, you can save time involved in dimensioning a sketch by dimensioning half of the section and then mirroring it. Pro/ENGINEER will assume that the mirrored half has the same dimensions as the sketched half.*

## 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, you are drawing a sketch of a piston and you have to work on the minute details of the grooves for the piston rings. To work on these minute details, you have to enlarge the display of these grooves. You can enlarge or reduce the drawing display using various drawing display tools provided in Pro/ENGINEER. These tools are available in the **View** toolbar. Some of these drawing display options are discussed next. The remaining drawing display options will be discussed in later chapters.

## Zoom In



This tool enlarges the view of the drawing on the screen. After choosing the **Zoom In** button, you will be prompted to define a box. The area that you will enclose inside the box will be enlarged and displayed in the graphics window. Note that when you enlarge the view of the drawing, the original size of the entities is not changed. To exit the zoom tool right-click in the graphics window.

## Zoom Out



This tool reduces the view of the drawing on the screen, thus increasing the drawing display area. Each time you choose this button to zoom out, the display of the sketch in the graphics window is reduced.

## Refit object to fully display it on the screen



This option reduces or enlarges the display such that all entities that comprise the sketch are fitted inside the current display. Note that the dimensions may not necessarily be included in the current display.

## Redraw the current view



While working with complex sketches, some unwanted temporary information is retained on the screen. The unwanted information may include the shadows of the deleted sketched entities, dimensions, and so on. This unwanted information can be removed from the graphics window using the **Redraw the current view** button. This option is extensively used when designing in Pro/ENGINEER.



### Note

To remove the temporary information you can also choose **View > Repaint** from the menu bar or **CTRL+R** to repaint the screen.

*If you have a mouse that has a middle mouse button wheel, then scrolling the wheel will zoom in and out.*

*One more way to zoom in and out is to use the middle mouse button and the CTRL key. 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.*

*In the **Sketch** mode, you can pan the sketch using the middle mouse button but in the **Part** mode, use SHIFT+middle mouse button to pan the model.*

## TUTORIALS

### Tutorial 1

In this tutorial, you will draw the sketch for the model shown in Figure 1-18. The sketch is shown in Figure 1-19. **(Expected time: 30 min)**

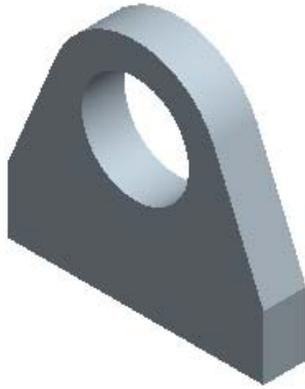


Figure 1-18 Model for Tutorial 1

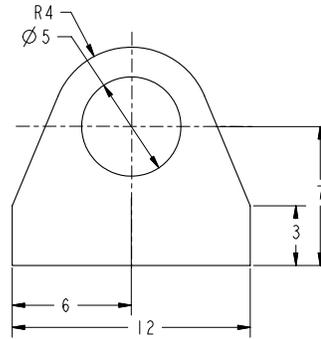


Figure 1-19 Sketch of the model

The following steps are required to complete this tutorial:

- Start Pro/ENGINEER Wildfire 2.0 session.
- Set the working directory and create a new sketch file.
- Draw lines using the line tool, refer to Figures 1-20 and 1-21.
- Draw an arc and a circle, refer to Figures 1-22 and 1-23.
- Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 1-24
- Save the sketch and close the file.

### Starting Pro/ENGINEER

- Start Pro/ENGINEER Wildfire 2.0 by double-clicking on the Pro/ENGINEER icon on the desktop of your computer or by using the **Start** menu.

### Setting the Working Directory

When the Pro/ENGINEER session is started, the first task is to set the working directory. A working directory is a directory on your system where you can save the work done in the current session of Pro/ENGINEER. You can set any directory existing on your system as the working directory. Because this is the first tutorial of this chapter, you need to create a folder named *c01*, if it does not exist.

- Choose the **Set Working Directory** option from the **File** menu. The **Select Working Directory** dialog box is displayed.
- Browse and select *C:\ProE-WF-2.0*. If this folder is non existing then create it prior to setting the working directory.
- Choose the **New Directory** button in the **Select Working Directory** dialog box; the **New Directory** dialog box is displayed. 
- Type **c01** in the **New Directory** edit box and choose **OK** from the dialog box. You have created a folder named *c01* in *C:\ProE-WF-2.0*.

5. Choose **OK** from the **Select Working Directory** dialog box. You have set the working directory to *C:\ProE-WF-2.0\c01*. A message is displayed in the **Message Area** that the directory successfully changed to *C:\ProE-WF-2.0\c01* directory.

### Starting a New Object File

Any sketch drawn in the **Sketch** mode is saved with the *.sec* file extension. This file format is one of the file formats available in Pro/ENGINEER.

1. Choose the **Create a new object** button from the **File** toolbar. The **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box. A default name of the sketch appears in the **Name** edit box. 
2. Enter *c01tut1* in the **Name** edit box and choose the **OK** button.

You are in the sketcher environment of the **Sketch** mode. When you enter the sketcher environment, the Navigator is displayed to the left in the graphics window.

3. Slide in the Navigator to the left by clicking on the sash present on its right edge. Now, the drawing area is increased.

### Drawing the Lines of the Sketch

Start drawing the sketch with the right vertical line.

1. Choose the **Create 2 point lines** button from the **Sketcher Tools** toolbar. 
2. Specify the start point to the right in graphics window by clicking. One end of the line is attached to the cursor. Move the cursor down to get an approximate size of the line.

Notice that when the cursor is moved vertically downward, a red colored constraint **V** appears in the graphics window next to the line. This indicates that if you draw a line now, the vertical constraint will be applied to the line.

3. Click to specify the endpoint of the line. The vertical constraint **V** is applied to the line and the symbol **V** appears in yellow. The color of the constraint indicates that this constraint is strong. This means that you cannot change the orientation of this line until you delete the constraint that is applied on the line.

Another rubber-band line is attached to the cursor with its start point at the endpoint of the last line.

4. Move the cursor horizontally toward the left; a horizontal rubber-band line extends to the left as you move the mouse.
5. After you get the desired size of the line, click to end the line. Notice that a horizontal constraint **H** that is yellow in color is applied to the line.
6. Move the cursor upward in the graphics window; a vertical rubber-band line extends as

you move the mouse. As you move the cursor upward, notice that at a particular point where the length of the left vertical line is equal to the length of the right vertical line,  $L_1$  symbol is displayed on both the vertical lines. This symbol suggests that the equal length constraint is applied to the two vertical lines.

- When the  $L_1$  constraint appears on the vertical line, click to specify the endpoint of the vertical line. Notice that the  $L_1$  constraint is displayed in gray color, as shown in Figure 1-20. This suggests that it is a weak constraint. The rubber-band line is still attached to the cursor.

You can also apply the constraints later. But to save an extra step of adding the constraints, you will use the constraints that are applied automatically while drawing.

- Move the cursor to size the line and specify the endpoint of the left inclined line, see Figure 1-20.
- Press the middle mouse button to end line creation. You will notice that gray colored dimensions are applied to the sketch, see Figure 1-20. The color of these dimensions indicates that these dimensions are weak dimensions. These dimensions are automatically deleted when you have completed the sketch or when you are adding dimensions and constraints manually. When system deletes weak dimensions, it does not confirm their deletion.
- The line option is still active. Move the cursor close to the top end of the right vertical line. You will notice that as you bring the cursor close to the top end, the cursor snaps to that point. Select the point by clicking.
- Size the inclined line and specify the endpoint of the right inclined line. Press the middle mouse button to end line creation.

Figure 1-21 shows the lines that you have drawn. Now, the arc and circle will be drawn.

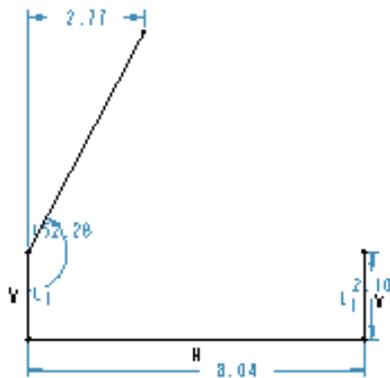


Figure 1-20 Lines with weak dimensions

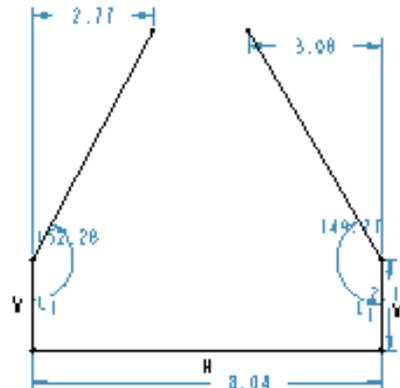


Figure 1-21 Partial sketch with weak dimensions

### Drawing the Arc

1. Choose the **Create an arc by 3 points or tangent to an entity at its endpoint** button from the **Sketcher Tools** toolbar. You are prompted to select the start point of the arc. 
2. Select the endpoint of the left inclined line; the **Target** symbol appears in green color.
3. Move the cursor along the tangent direction through a small distance. A rubber-band arc that is tangent to the endpoint of the line appears. As you move the cursor to the endpoint of the right inclined line, at a particular point the tangent constraint is applied at both the ends of the arc. This is indicated by the symbol **T** that appears on the endpoints of the inclined lines.
4. As the tangent constraint appears, click to end arc creation. You will notice that the tangent constraint with a symbol **T** appears at the endpoints of the arc, as shown in Figure 1-22. Press the middle mouse button to end arc creation.

The tangent constraint **T** will appear in white, which suggests that it is a strong constraint and the tangency of the inclined line with the arc cannot be modified until you delete the tangent constraint.

Note that in Figure 1-21 there are some weak dimensions that are not displayed in Figure 1-22. This is because the weak dimensions are deleted without confirming their deletion. Hence, after drawing the arc some weak dimensions get deleted automatically.



#### Note

*If the tangent constraint symbol is not displayed on any of the inclined lines, apply the constraint manually using the **Constraints** dialog box that is displayed when you choose the **Impose sketcher constraints on the section** button from the **Sketcher Tools** toolbar (see page 1-16 for more information).*

### Drawing the Circle

1. Choose the black arrow on the right of the **Create circle by picking the center and a point on the circle** button to display the flyout. From this flyout, choose the **Create concentric circle** button; you are prompted to select an arc. 
2. Select the arc by clicking by clicking on it. Move the mouse and a circle appears.
3. To draw the circle, click to select a point inside the sketch.
4. Press the middle mouse button to end circle creation. The sketch is complete and appears similar to that shown in Figure 1-23.

### Dimensioning the Sketch

The right vertical line, the bottom horizontal line, the arc, and the circle are dimensioned automatically and weak dimensions are applied to them. You will use these dimensions. Hence, there is no need to dimension these entities again.

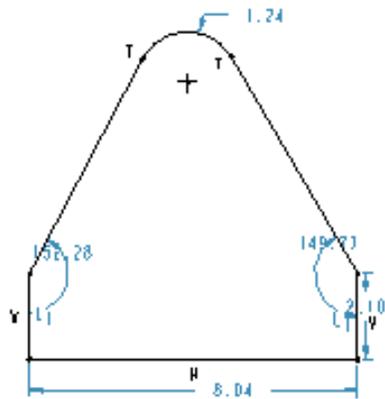


Figure 1-22 Sketch with arc

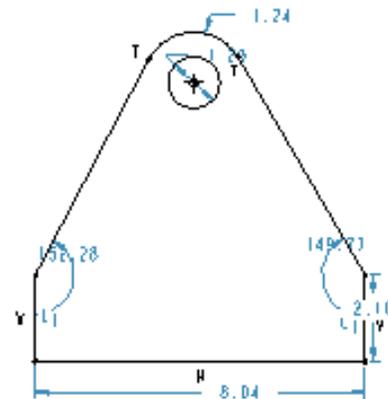


Figure 1-23 Sketch with all entities, weak dimensions, and weak constraints

1. Choose the **Create defining dimension** button from the **Sketcher Tools** toolbar. 
2. Select the center of the circle and then select the bottom horizontal line by clicking on them. Both, the center and the line turn red in color.
3. Place the dimension on the right of the sketch by pressing the middle mouse button.
4. Select the center of the circle and then select the left vertical line. Both the center and the vertical line turn red in color.
5. Press the middle mouse button to place the dimension below the sketch, refer to Figure 1-24.

### Modifying the Dimensions

The sketch is dimensioned with default values. You need to modify these values to the given values.

1. Choose the **Select items** button. 
2. Select all dimensions by specifying a window around them.



#### Note

You can also use **CTRL+ALT+A** to select the entire sketch with dimensions.

3. When all dimensions turn red in color, choose the **Modify the values of dimensions, geometry of splines, or text entities** button. The **Modify Dimensions** dialog box is displayed. 

All dimensions in the sketch are displayed in this dialog box and each dimension has a separate thumbwheel and an edit box. You can use the thumbwheel or the edit box to

modify the dimensions. It is recommended that you use the edit boxes to modify the dimensions if the change in the dimension value is large.

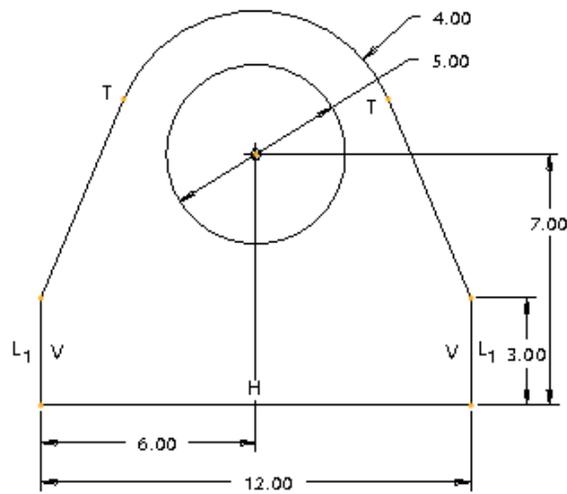
4. Clear the **Regenerate** check box and then modify the values of the dimensions.

When you clear this check box, any modification in a dimension value does not update the sketch. It is recommended that you clear the **Regenerate** check box when more than one dimension has to be modified.

Notice that the dimension you select in the **Modify Dimensions** dialog box gets enclosed in a yellow box in the graphics window.

5. Modify all dimensions, as shown in Figure 1-24. After modifying the dimensions, choose the **Regenerate the section and close the dialog** button from the **Modify Dimensions** dialog box. A message **Dimension modifications successfully completed** is displayed in the **Message Area**.

The sketch is complete and is shown in Figure 1-24.



**Figure 1-24** The complete sketch with dimensions and constraints



**Tip:** You can modify the location of the dimensions as they appear on the screen by selecting and dragging them to a new location.

### Saving the Sketch

The sketch will now be saved. You have to save the sketch because you may need the sketch later in the **Part** mode in order to create a 3D model.

1. Choose **Save the active object** button from the **File** 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 **Window > Close** from the menu bar.

## Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 1-25. The sketch is shown in Figure 1-26. For your reference, all entities in the sketch are labeled alphabetically. **(Expected time: 30 min)**

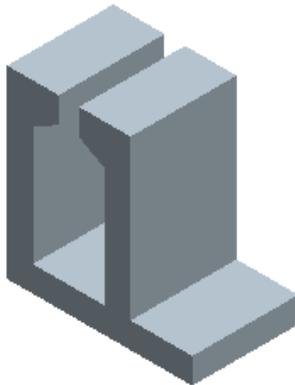


Figure 1-25 Model for Tutorial 2

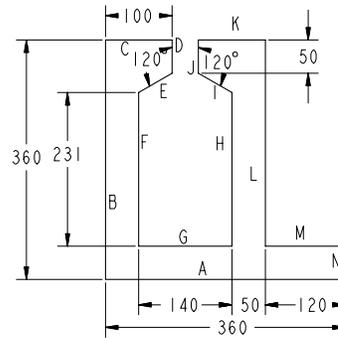


Figure 1-26 Sketch of the model

The following steps are required to complete this tutorial:

- a. Set the Working directory and create a new object file.
- b. Draw the sketch using the line tool, refer to Figure 1-27.
- c. Dimension the required entities and then modify the dimensions of the sketch, refer to Figures 1-28 and 1-29.
- d. Save the sketch and close the file.

### 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 Pro/Engineer is started, then you have to again set the working directory by following the steps given next.

1. Open the Navigator by clicking the top arrows on the left edge of the Pro/ENGINEER main window; the Navigator slides out.
2. Click on the plus symbol adjacent to the *ProE-WF-2.0* folder in the Navigator; the contents of the *ProE-WF-2.0* folder are displayed.

3. Now right-click on the *c01* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; the working directory is set to *c01*.
4. Close the Navigator by clicking on the sash located at the right edge of Navigator. The Navigator slides in.

### Starting a New Object File

1. Choose the **Create a new object** button from the **File** toolbar; the **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box; a default name of the sketch appears in the **Name** edit box. 
2. Enter *c01tut2* in the **Name** edit box and choose **OK**, you are in the sketcher environment of the **Sketch** mode.

### Drawing the Sketch

The sketch in Figure 1-26 consists of only lines. For ease of understanding, all lines in the sketch are labelled alphabetically.

1. Choose the **Create 2 point lines** button from the **Sketcher Tools** toolbar. Select a point close to the lower right corner of the graphics window by clicking and start drawing the horizontal line A. Here, you will notice that as you draw line A, the **H** symbol is displayed on the line. This indicates that the line is horizontally constrained. Move the cursor toward the left and specify the endpoint of the line. 
2. Move the cursor vertically upwards so that the **V** constraint appears on the line. When you get the appropriate size of the line, click to specify the endpoint of line B; line B is completed.
3. Move the cursor to the right in the graphics window and click to specify the endpoint of line C.
4. Now, to draw line D, move the cursor down and click to specify the endpoint of line D.
5. Line E is inclined. Move the cursor to size the line and click to specify the endpoint of line E.
6. The next line you need to draw is line F. Move the cursor vertically downwards and click to specify the endpoint of line F.
7. Now, to draw line G, move the cursor horizontally toward the left and click to specify the endpoint of line G.
8. Move the cursor vertically upwards and click to specify the endpoint of line H.
9. Now, continue drawing the remaining lines that are shown in Figure 1-27. When the sketch is complete, end line creation by pressing the middle mouse button. Notice that the sketched entities are dimensioned automatically as you draw them. These dimensions are weak dimensions and appear in gray color.

## Applying the Constraints to the Sketch

Constraints are applied to the sketch to maintain the design intent of the feature and this might sometimes result in less dimensions in the sketch.

1. Choose the **Impose sketcher constraints on the section** button from the **Sketcher Tools** toolbar; the **Constraints** dialog box is displayed. 
2. Choose the **Create Equal Lengths, Equal Radii, or Same Curvature constraint** button and select lines F and H. The equal length constraint  $L_2$  is applied to both the lines. The constraint labels such as  $L_2$  or  $L_3$  vary from sketch to sketch. 
3. Select lines C and K; the equal length constraint is applied to both the lines.
4. Now, select lines J and N; the equal length constraint is applied to both the lines.
5. Select lines A and B. The equal length constraint is applied to both the lines.
6. Choose the **Make line or two vertices horizontal** button from the **Constraints** dialog box; you are prompted to select a line or two vertices. 
7. Select the vertex that is joining lines L and M and the vertex that is joining lines G and H. Both the vertices are aligned horizontally, as shown in Figure 1-27.
8. Select the vertex that is joining lines C and D and the vertex that is joining lines J and K. Both the vertices are aligned horizontally, as shown in Figure 1-27.

## Dimensioning the Sketch

Weak dimensions are already applied to the sketch while drawing. You need to dimension only the angle between lines D and E and lines J and I.

1. Choose the **Create defining dimension** button. 
2. Select lines D and E using the left mouse button; the selected lines turn red in color. Now, press the middle mouse button to place the dimension close to the vertex where lines D and E join.
3. Similarly, dimension the angle between lines J and I.

Figure 1-28 shows the sketch after applying dimensions. If your sketch does not have all dimensions shown in this figure, apply them using the **Create defining dimension** button.

## Modifying the Dimensions

The dimensions that are applied to the sketch need modification in dimension values.

1. Choose the **Select items** button and then select all dimensions by specifying a window around them. 

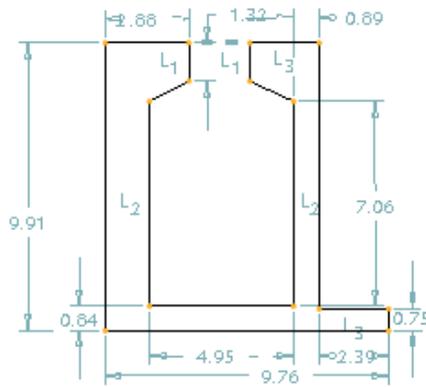


Figure 1-27 Sketch with weak dimensions and constraints

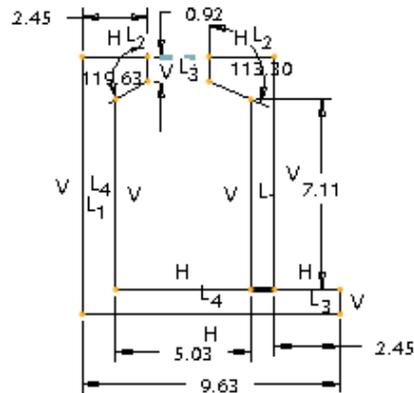


Figure 1-28 Sketch after dimensioning

2. When the dimensions turn red in color, choose the **Modify the values of dimensions, geometry of splines, or text entities** button. The **Modify Dimensions** dialog box is displayed.
3. Clear the **Regenerate** check box and then modify the values of the dimensions. When you clear this check box, the sketch is not regenerated while you modify the dimensions.  
  
Notice that the dimension you select in the **Modify Dimensions** dialog box is enclosed in a yellow box in the graphics window.
4. When all dimensions are modified, choose the **Regenerate the section and close the dialog** button from the **Modify Dimensions** dialog box. A message **Dimension modifications successfully completed** is displayed in the **Message Area**. The completed sketch is shown in Figure 1-29.

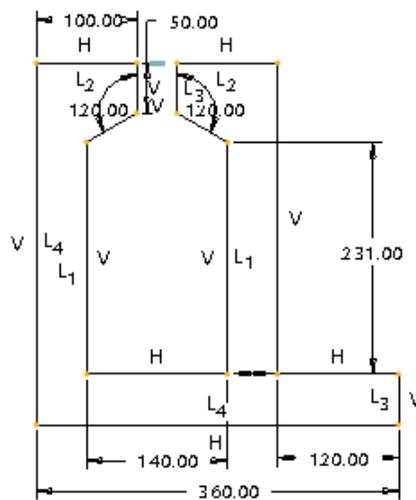


Figure 1-29 Complete sketch with dimensions and constraints

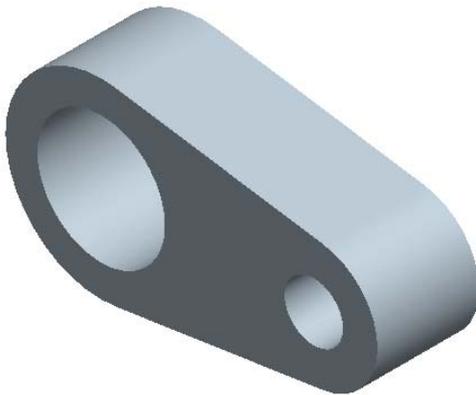
6. Save the sketch as discussed earlier. After saving the sketch, choose **Window > Close** from the menu bar to exit the **Sketch** mode.

**Note**

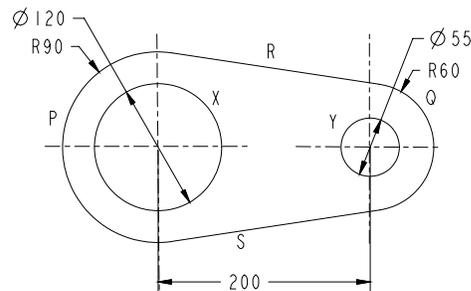
You can also modify dimensions individually. But, individual modification of dimensions is recommended only when either there is a minor change in the dimension value or when only one dimension is required to be modified.

### Tutorial 3

In this tutorial, you will draw the sketch for the model shown in Figure 1-30. The sketch is shown in Figure 1-31. For your reference, all entities in the sketch are labeled alphabetically. **(Expected time: 30 min)**



*Figure 1-30 Model for Tutorial 3*



*Figure 1-31 Sketch of the model*

The following steps are required to complete this tutorial:

- a. Set the working directory and create a new object file.
- b. Draw the sketch using the sketcher tools, refer to Figures 1-32 through 1-35.
- c. Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 1-36
- d. Save the sketch and close the file.

### 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 Pro/Engineer is started, then you have to again set the working directory by following the steps given next.

1. Open the Navigator by sliding it out. Click on the plus symbol adjacent to the *ProE-WF-2.0* folder in the Navigator; the contents of the *ProE-WF-2.0* folder are displayed.
2. Now right-click on the *c01* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; the working directory is set to *c01*. Close the Navigator.

### Starting New Object File

1. Choose the **Create a new object** button from the **File** toolbar. The **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box. A default name of the sketch appears in the **Name** edit box. 
2. Enter *c01tul3* in the **Name** edit box. Choose the **OK** button to enter the sketcher environment of the **Sketch** mode.

### Drawing the Circles

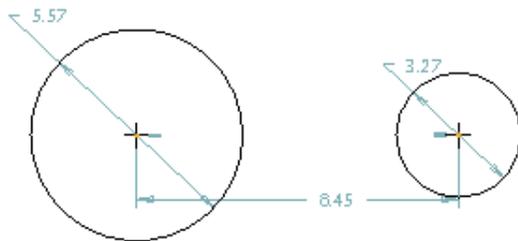
1. Choose **Create circle by picking the center and a point on the circle** button from the **Sketcher Tools** toolbar and specify the center of the circle. 
2. Move the cursor to size the circle and click to complete the circle.
3. Draw another circle whose center is collinear with the center of the previous circle.

Figure 1-32 shows the two collinear circles drawn using the **Create circle by picking the center and a point on the circle** button from the **Sketcher Tools** toolbar.

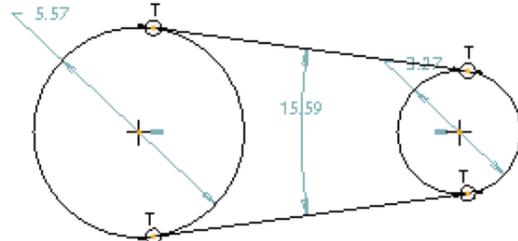
### Drawing the Tangent Lines

1. Choose the **Create lines tangent to 2 entities** button from the flyout in the **Sketcher Tools** toolbar. You are prompted to select the start location on the arc or a circle. 
2. Select the left circle at the top. A rubber-band line appears whose one end is attached to the circle and the other end is attached to the cursor.
3. Click on the top of the right circle; a tangent that connects the two circles is drawn.
4. Similarly, draw a tangent by selecting the two circles at the bottom.

Figure 1-33 shows the sketch after drawing the tangent lines.



**Figure 1-32** The two circles with weak dimensions and constraints



**Figure 1-33** Circles joined by lines and the tangent constraint applied to them

### Trimming the Circles

As evident from Figure 1-33, the tangents that are drawn intersect the circles at the point where they meet the circle. Therefore, the part of the circle that is not required can be dynamically trimmed.

1. Choose the **Dynamically trim section entities** button from the **Sketcher Tools** toolbar. 
2. Select the two circles individually to trim them at the locations shown in Figure 1-34. Figure 1-35 shows the two circles after deleting the unwanted portion of the circle.

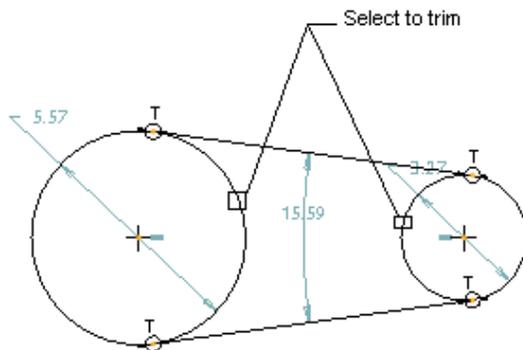


Figure 1-34 Locations to trim



Figure 1-35 Sketch after trimming

### Drawing the Circles

1. Choose the black arrow on the right of the **Create circle by picking the center and a point on the circle** button to display the flyout. From this flyout, choose the **Create concentric circle** button; you are prompted to select an arc. 
2. Select arc P and create circle X concentric to the arc. Similarly, select arc Q to create a concentric circle Y.

Notice that the two arcs are applied radius dimension whereas the circles are applied diameter dimension. This is because by default, the arcs are applied radius dimension and circles are applied diameter dimension.

### Dimensioning the Sketch

In order to fully define a sketch, it should be dimensioned.

1. Choose the **Create defining dimension** button. 
2. Select the centers of the two circles and place the dimension at the bottom of the sketch.

### Modifying the Dimensions

1. Choose the **Select items** button. 

2. Select all dimensions by defining a window.



**Note**

You can also use **CTRL+ALT+A** from the keyboard to select all entities and items in the sketch.

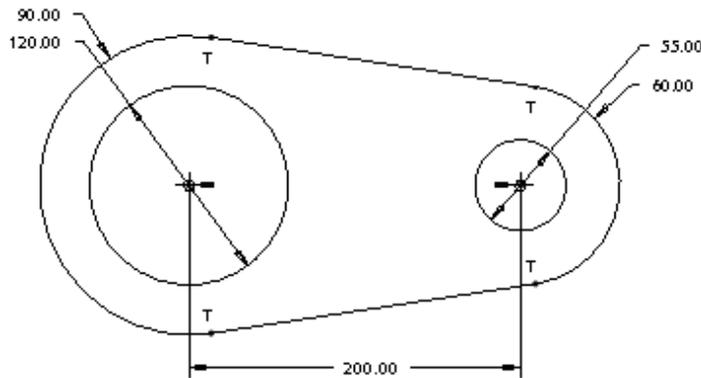
3. When all dimensions turn red in color, choose the **Modify the values of dimensions, geometry of splines, or text entities** button; the **Modify Dimensions** dialog box is displayed.
4. Clear the **Regenerate** check box and then modify the values of the dimensions.



You will notice that the dimension you edit in the **Modify Dimensions** dialog box is enclosed by a yellow box in the graphics window.

5. When all dimensions are modified, choose the **Regenerate the section and close the dialog** button from the **Modify Dimensions** dialog box; a message **Dimension modifications successfully completed** is displayed in the **Message Area**.

The sketch is completed and is shown in Figure 1-36.



*Figure 1-36 The complete sketch with dimensions and constraints*

6. Save the sketch as discussed earlier. After saving the sketch, choose **Window > Close** from the menu bar to exit the **Sketch** mode.

**Self-Evaluation Test**

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

1. Dimensions and constraints are automatically applied to a sketch when you draw it. (T/F)

2. In Pro/ENGINEER, you can create lines that are tangent to two circles. (T/F)
3. If the **Intent Manager** is on and you draw a line, the cursor snaps to the endpoint of the previous line. (T/F)
4. You can convert a weak constraint to strong by using the shortcut menu that is displayed when you right-click on the weak constraint. (T/F)
5. While drawing a circle, first you need to specify its diameter. (T/F)
6. The \_\_\_\_\_ menu in the menu bar has the **Modify** option in it.
7. The sketch can be modified by changing its \_\_\_\_\_.
8. **Intent Manager** is \_\_\_\_\_ by default when you enter the **Sketch** mode. (on/off)
9. In the **Sketch** mode the tangent constraint is represented by \_\_\_\_\_ symbol.
10. The **Sketch** mode file is saved with a \_\_\_\_\_ file extension.

### Review Questions

Answer the following questions:

1. What is the need of **Sketch** mode in Pro/ENGINEER?
2. What are the four basic steps to create a sketch?
3. What are the various types of lines you can sketch using the buttons available in the **Sketcher Tools** toolbar?
4. Why is it important to select the working directory before creating a new file?
5. Write all steps involved in creating a sketch that is accepted by Pro/ENGINEER.
6. You can dynamically modify the geometry of a sketch. (T/F)
7. You can use the **Create rectangle** button from the **Sketcher Tools** toolbar to draw a square. (T/F)
8. The \_\_\_\_\_ button is used to apply constraints manually.
9. You cannot undo a previous operation in the sketcher environment. (T/F)
10. You can also use the options in the menu bar to draw a sketch from the **Sketch** menu. (T/F)

**Exercises**

**Exercise 1**

In this exercise, you will draw the sketch for the model shown in Figure 1-37. The sketch is shown in Figure 1-38. **(Expected time: 30 min)**

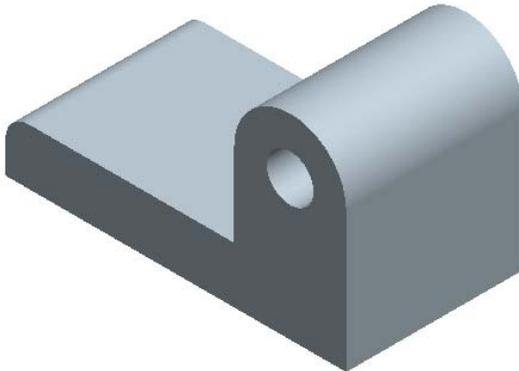


Figure 1-37 Solid model for Exercise 1

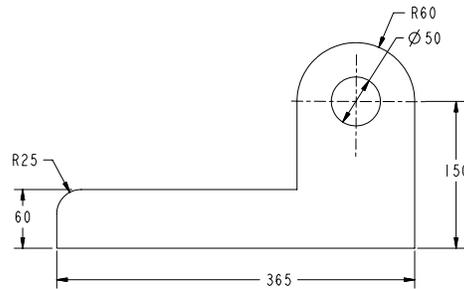


Figure 1-38 Sketch of the model

**Exercise 2**

In this exercise, you will draw the sketch for the model shown in Figure 1-39. The sketch is shown in Figure 1-40. **(Expected time: 30 min)**

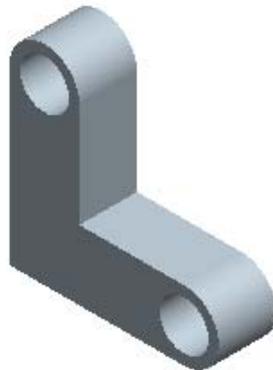


Figure 1-39 Solid model for Exercise 2

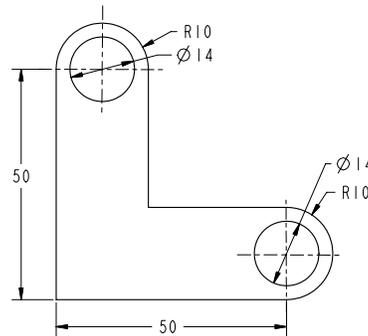


Figure 1-40 Sketch of the model

### Exercise 3

In this exercise, you will draw the sketch for the model shown in Figure 1-41. The sketch is shown in Figure 1-42. **(Expected time: 30 min)**

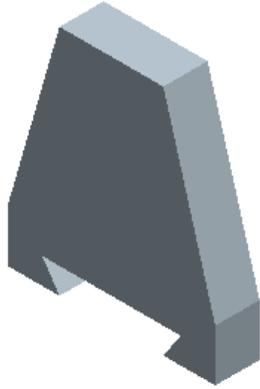


Figure 1-41 Solid model for Exercise 3

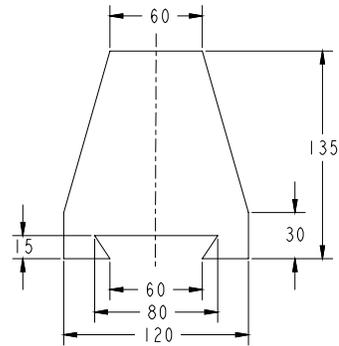


Figure 1-42 Sketch of the model

### Exercise 4

In this exercise, you will draw the sketch for the model shown in Figure 1-43. The sketch is shown in Figure 1-44. **(Expected time: 30 min)**

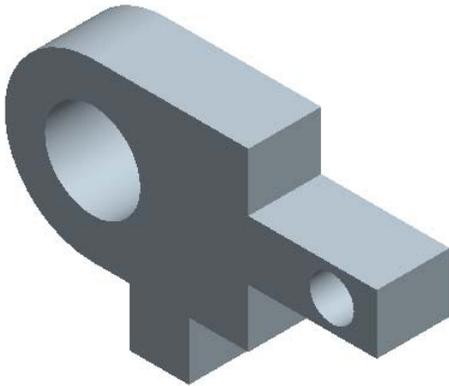


Figure 1-43 Solid model for Exercise 4

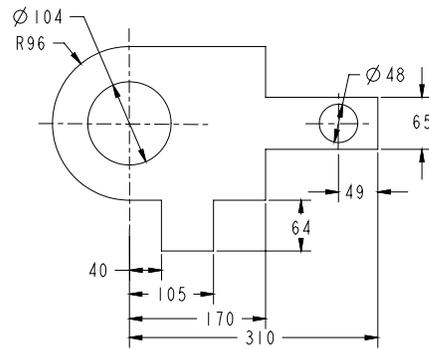


Figure 1-44 Sketch of the model

#### Answers to Self-Evaluation Test

1 - T, 2 - T, 3 - T, 4 - T, 5 - F, 6 - Edit, 7 - dimensions, 8 - on, 9 - T, 10 - .sec

# Chapter 2

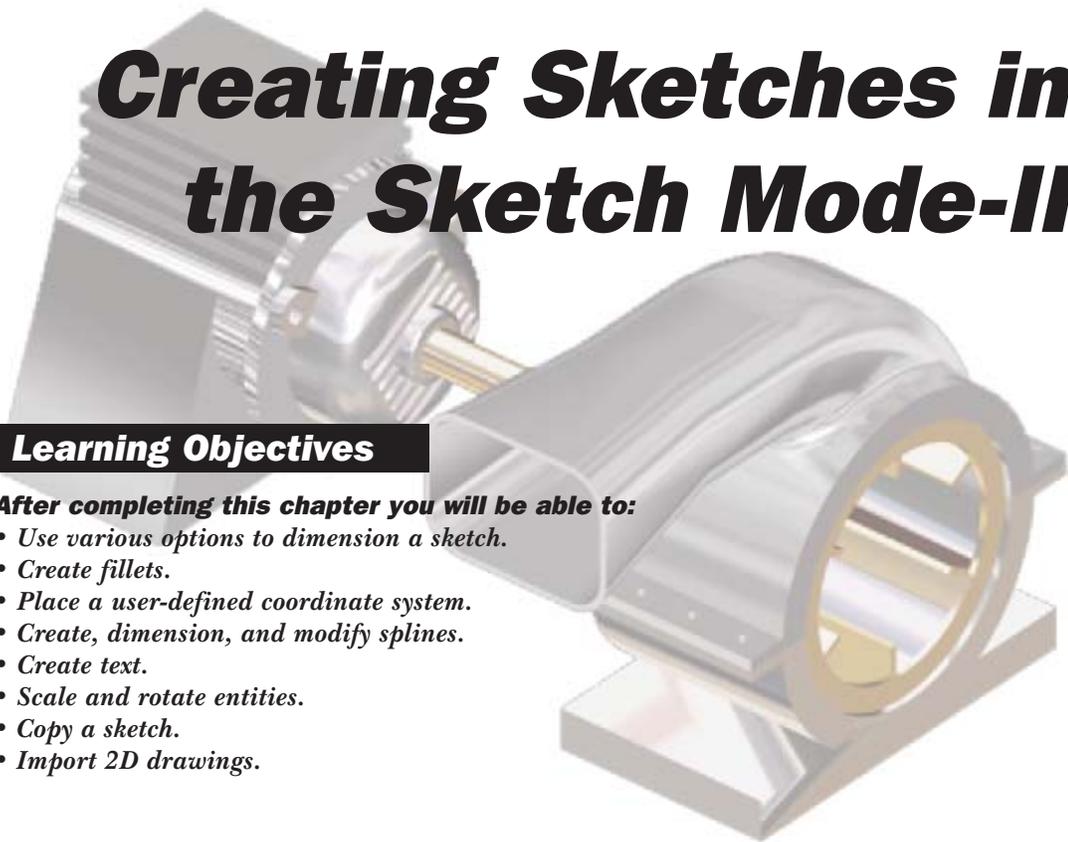
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## Creating Sketches in the Sketch Mode-II

### Learning Objectives

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

- Use various options to dimension a sketch.
- Create fillets.
- Place a user-defined coordinate system.
- Create, dimension, and modify splines.
- Create text.
- Scale and rotate entities.
- Copy a sketch.
- Import 2D drawings.



## DIMENSIONING THE SKETCH

In Chapter 1 you have learned how to dimension a sketch using the **Normal** option or using its equivalent tool, that is, **Create defining dimension** button in the **Sketcher Tools** toolbar. In this chapter, you will learn the **Baseline** option of dimensioning a sketch.

Normal  
Reference  
Baseline  
Explain

Choose **Sketch > Dimension** from the menu bar; the cascading menu is displayed, as shown in Figure 2-1. The options that are available to dimension a sketch are shown in this figure and are discussed next.

**Figure 2-1**  
*Dimensioning options*

### Dimensioning a Sketch Using the Baseline Option

In Pro/ENGINEER, the **Baseline** option of dimensioning is used to create dimensions in terms of horizontal and vertical location values of an entity with respect to a specified baseline. This type of dimensioning in a drawing can be used for writing a CNC program to manufacture a component.

The **Baseline** option can be used to dimension lines, conics, arcs, and so on. The following steps explain the procedure to create dimensions using the **Baseline** option:

1. Choose **Sketch > Dimension > Baseline** from the menu bar.
2. Select the entity that will act as the baseline (origin or reference). Press the middle mouse button to place the dimension, the dimension **0.00** is displayed where you place the dimension. Note that since the location value of the baseline is taken as the origin, the dimension value of the baseline entity will become 0.00. The dimension values of all other entities dimensioned with reference to the baseline will be measured from this origin.

Depending upon the entity selected to act as the baseline, the horizontal or the vertical dimension value of the location of the entity will be placed. For example, if you select a vertical line, the value of its location will be placed vertically. Similarly, if you select a horizontal line, the value of its location will be placed horizontally.

For arcs, circles, and splines there are two options to dimension using the **Baseline** option. When you select a circle center or an arc for baseline dimensioning and press the middle mouse button, the **Dim Orientation** dialog box is displayed, as shown in Figure 2-2. You are prompted to select the dimension orientation; the dimension is placed according to the orientation selected.



**Figure 2-2** *The Dim Orientation dialog box*

3. Next, choose the **Create defining dimension** button from the **Sketcher Tools** toolbar. Select the baseline dimension that was placed earlier and then select the entity to dimension. Now, press the middle mouse button to place the dimension.

The orientation of the dimension will depend upon the baseline dimension and the entity selected. Figure 2-3 shows a sketch dimensioned using the above-mentioned method. In this figure, the two baselines are dimensioned using the **Baseline** option. Therefore, the dimensions of these lines are displayed as 0.00. The remaining lines are dimensioned by using the **Create defining dimension** button by first selecting the baseline dimension and then the entity to dimension.

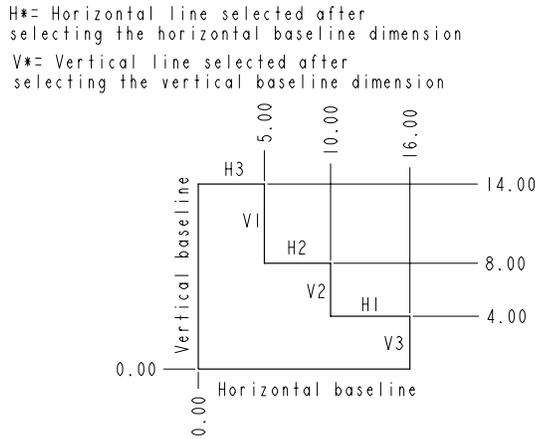


Figure 2-3 Baseline dimensioning of a sketch

## Replacing the Dimensions of a Sketch Using the Replace Option

The **Replace** option is used to replace a dimension from a sketch. To use this option you must have a dimensioned sketch. The following steps explain the procedure to dimension a sketch using the **Replace** option:

1. Choose **Edit > Replace** from the menu bar; you will be prompted to select a dimension to be replaced.
2. Select the dimension to be replaced; the selected dimension is erased. Select an entity to dimension and place the dimension at the desired place. The previous dimension is replaced by a new dimension.

## CREATING FILLETS

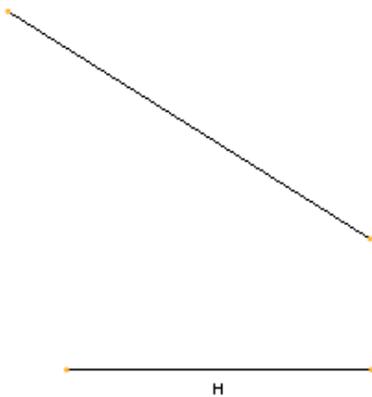
In the sketcher environment, you can create two types of fillets.

1. Circular fillets
2. Elliptical fillets

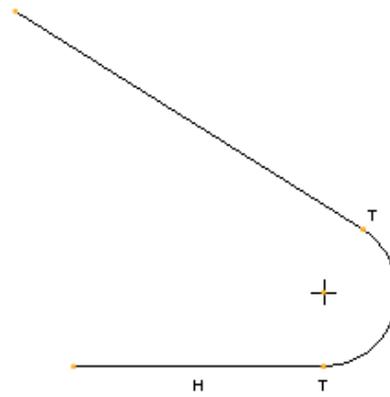
### Creating Circular Fillets

A circular fillet is the arc that is used to join two lines, a line and an arc, or two arcs. The fillet is controlled by its radius or diameter dimension. The resulting fillet will depend on the location where the elements are selected.

Figure 2-4 shows two lines that do not join at the corners and Figure 2-5 shows the circular fillet created between them. The circular fillet that is created is an arc with its endpoints tangent to the two lines. This is evident from the **T** symbol that is automatically applied to the endpoints of the fillet arc.

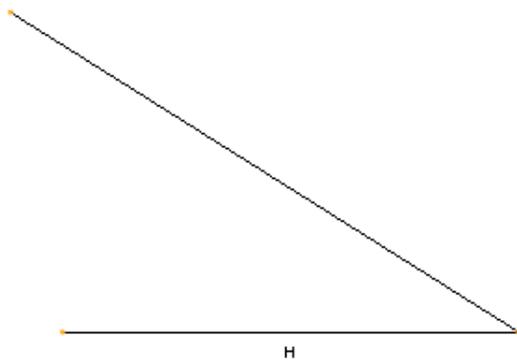


**Figure 2-4** Two lines that do not join

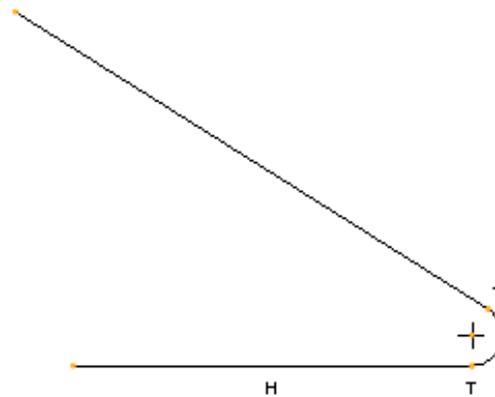


**Figure 2-5** Fillet created between the two lines

Figure 2-6 shows two lines that join at the corner and Figure 2-7 shows the circular fillet created that joins the two lines by an arc. The corner is automatically deleted.



**Figure 2-6** Two lines joining at the corner



**Figure 2-7** Filleted corner

Figure 2-8 shows two sets of arcs and Figure 2-9 shows the circular fillet created between the two arcs. The location where you select the arcs to create the fillet is important. The fillet is created tangent to the selection points on the arcs. Here, the endpoints of the arcs are selected to create the fillet.

If the points of selection on the two arcs are away from the endpoints of the arcs, then the fillet is created at the selection points. The portion of the arc that extends beyond the fillet should be manually deleted or trimmed. This case of creating a fillet is explained in Figures 2-10 and 2-11.

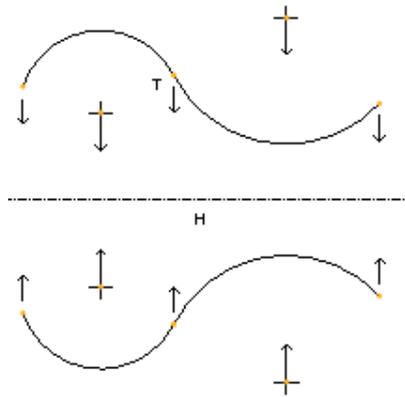


Figure 2-8 Two sets of arcs

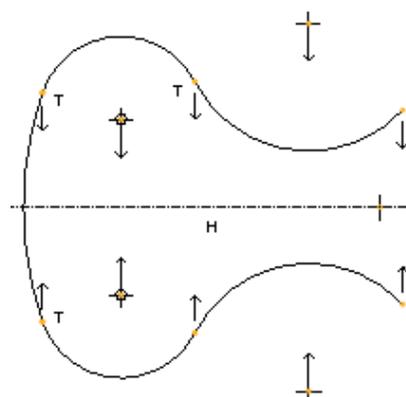


Figure 2-9 Fillet created by selecting the endpoints

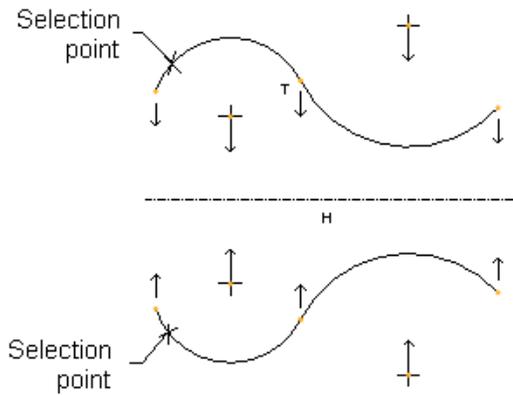


Figure 2-10 Points selected on arcs

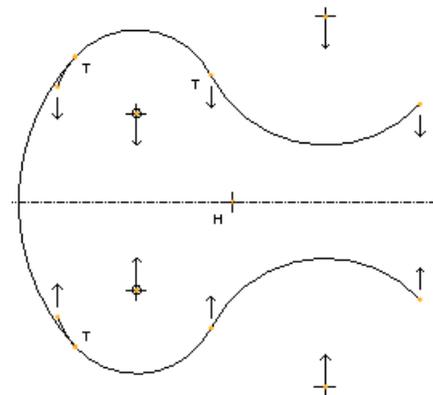


Figure 2-11 Filleted arcs



The **Create a circular fillet between two entities** button in the **Sketcher Tools** toolbar is used to create circular fillets. The following steps explain the procedure to create a circular fillet:

1. Choose the **Create a circular fillet between two entities** button from the **Sketcher Tools** toolbar; you are prompted to select two entities.
2. Select the first entity for filleting by using the left mouse button; the yellow color of the first entity changes to red. Now, select the second entity. As soon as you select the second entity, if possible, a fillet is drawn between the two selected entities.
3. Repeat step 2 until you have created all fillets.



**Note** You can create a fillet between any two entities except between two parallel lines and between a centerline and an entity.

## Creating Elliptical Fillets

An elliptical fillet is the arc in the form of an ellipse that joins two lines, two arcs, or a line and an arc. The geometry of the elliptical fillet depends on the location where you select the entities to create a fillet.

The advantage of elliptical fillets over circular fillets is that elliptical fillets have the geometry of an ellipse. As you know, an ellipse is controlled by dimensions in two directions. Therefore, when an elliptical fillet is dynamically modified, its geometry can be controlled in either the x-direction or the y-direction. Hence, the elliptical fillet when modified, results in more curved geometric shapes than a circular fillet.

Figures 2-12 and 2-13 illustrate the elliptical fillet. Notice, that a strong tangent constraint **T** is automatically applied when you create a fillet.

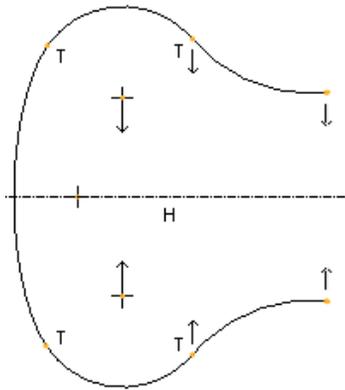


Figure 2-12 Arcs to be filleted

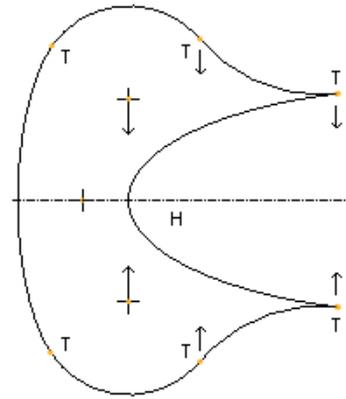


Figure 2-13 Elliptical fillet created



An elliptical fillet is created by choosing the **Create an elliptical fillet between two entities** button in the **Sketcher Tools** toolbar. The following steps explain the procedure to create elliptical fillets:

1. Choose the black arrow on the right of the **Create a circular fillet between two entities** button to display a flyout. Choose the **Create an elliptical fillet between two entities** button from this flyout; you are prompted to select two entities.
2. Select the first entity by clicking; the color of the entity changes to red.
3. Select the second entity. As soon as you select the second entity, the elliptical fillet is created. The shape of the elliptical fillet depends upon the specified points. After the fillet is created, you are again prompted to select two entities for elliptical fillet.
4. Repeat steps 2 and 3 until you have created all fillets.

When you select an elliptical fillet to dimension, the **Ellipse Rad** dialog box is displayed, as shown in Figure 2-14. There are two radio buttons in this dialog box. The **X Radius** radio

button when selected dimensions the elliptical fillet radially in the X-direction. The **Y Radius** radio button when selected dimensions the elliptical fillet radially in the Y-direction.

## CREATING A REFERENCE COORDINATE SYSTEM



The **Create reference coordinate system** button in the **Sketcher Tools** toolbar is used to create a coordinate system and add it to the created sketch. The coordinate system acts as 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.

The user-defined coordinate system is used in blend features to align different sections in a blend. It is also used in the **Assembly** mode and **Manufacturing** mode of Pro/ENGINEER.

The following steps explain the procedure to create a coordinate system:

1. Choose **Create reference coordinate system** button on the flyout that is displayed when you choose the black arrow on the right of the **Create points** button in the **Sketcher Tools** toolbar. You are prompted to select the location for the coordinate system. The coordinate system symbol is attached to the cursor.
2. Place the coordinate system at the desired points on the screen by clicking. The coordinate system will be placed at as many places as you click in the graphics window. You can end coordinate system creation by using the middle mouse button.



### Note

*If you add a coordinate system to a sketch, it must be dimensioned. Unless the coordinate system is placed at the endpoints of a line, an arc, a spline, or at the center of an arc or a circle. In other words, a coordinate system must be referenced to an entity in a sketch.*

## WORKING WITH SPLINES

Splines are curved entities that pass through an infinite number of intermediate points. Generally, splines are used to define the outer surface of a model. This is because the splines can provide different shape to curves and the flexibility to modify the surfaces that result from the splines. Splines find application in automobile and aeroplane body designing.

### Creating a Spline



To draw a spline, choose the **Create a spline curve** button from the **Sketcher Tools** toolbar.

The following steps explain the procedure to create a spline:

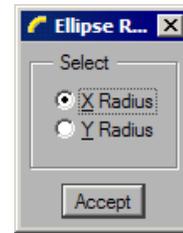


Figure 2-14 Ellipse Radius dialog box

1. Choose the **Create a spline curve** button from the **Sketcher Tools** toolbar; you are prompted to select the location for spline.
2. Use the left mouse button to select the start point for the spline. Similarly, select additional points in the graphics window and a spline will be drawn that passes through the specified points. All points through which the spline passes are called interpolation points.

**Note**

*If you select the start point of the spline as its end point, then the resulting spline will be in form of a continuous loop.*

## Dimensioning of Splines

When a spline is drawn, the weak dimensions are automatically applied to the spline. A spline can be dimensioned manually in the following ways:

1. Dimensioning the endpoints.
2. Radius of curvature dimensioning.
3. Tangency dimensioning.
4. Coordinate dimensioning.
5. Dimensioning the interpolation points.

### Dimensioning the Endpoints

To dimension a spline by selecting the endpoints:

1. Choose the **Create defining dimension** button from the **Sketcher Tools** toolbar.
2. Select the two endpoints of the spline and place the horizontal or vertical dimension by pressing the middle mouse button. Figure 2-15 shows a spline that is dimensioned by selecting the endpoints.

### Radius of Curvature Dimensioning

The radius of curvature of a spline can be dimensioned only if its tangency is defined. In other words, radius of curvature of a spline can be dimensioned only if the spline is tangent to an entity. For radius of curvature dimensioning of a spline:

1. Choose the **Create defining dimension** button from the **Sketcher Tools** toolbar.
2. Select the endpoint of the spline where tangency is defined.
3. Press the middle mouse button to place the dimension. Figure 2-16 shows the radius of curvature dimensioning of a spline.

### Tangency or Angular Dimensioning

A spline can be dimensioned tangentially. This type of dimensioning is also called angular dimensioning. To angular dimension a spline and a line tangent to it:

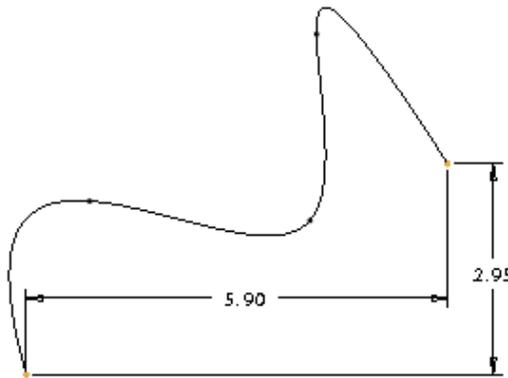


Figure 2-15 Endpoint dimensioning

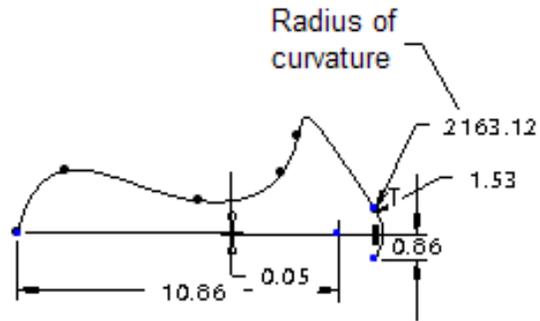


Figure 2-16 Radius of curvature

1. Choose the **Create defining dimension** button from the **Sketcher Tools** toolbar.
2. Select the spline by clicking.
3. Select the entity tangent to the spline by clicking.
4. Select the interpolation point of the spline that is to be dimensioned tangentially.
5. Press the middle mouse button to place the dimension.

### Coordinate Dimensioning

The spline can be dimensioned with respect to a user-defined coordinate system. Choose the **Create reference coordinate system** button from the flyout in the **Sketcher Tools** toolbar. The coordinate system is attached to the cursor. Place the coordinate system in the graphics window. Now, the spline can be dimensioned with respect to the coordinate system.

### Dimensioning the Interpolation Points

A spline can be dimensioned by dimensioning its interpolation points or vertices. This type of dimensioning is used when the designer wants the spline to be standard for all designs. This is because the exact curve can be duplicated if the interpolation points or the vertices of a spline are dimensioned.



**Tip:** A dimension can be moved by pressing and holding the left mouse button on the dimension and moving it. The dimension text is replaced by a red colored box. You can drag the dimension to the desired location in the graphics window and release the left mouse button to place the dimension at that point.

### Modifying a Spline

In Pro/ENGINEER Wildfire 2.0, a spline can be modified in the following ways:

1. Moving the interpolation points of the spline.
2. Adding points to a spline.

3. Deleting points of a spline.
4. Creating a control polygon and moving its control points.
5. Modifying the dimensions of the spline.

### Moving the Points of the Spline

The position of the interpolation points can be dynamically modified. To modify a spline, select an interpolation point on the spline and drag it to modify the shape of the spline.

You can also use the dashboard to modify a spline. To invoke the dashboard, select the spline and hold the right mouse button down to invoke the shortcut menu. Choose the **Modify** option from the shortcut menu. The **Modify Spline** dashboard is displayed at the bottom of the graphics window with the options and buttons to modify a spline .

The interpolation points of the spline appear in white in the graphics window. Drag the interpolation points to modify the shape of the spline. Choose ✓ from the dashboard to exit it.

### Adding Interpolation Points to a Spline

To add interpolation points on a spline, invoke the dashboard. Now, right-click on the spline to invoke the shortcut menu. Choose the **Add Point** option; a point is added to the spline where the spline was selected. The new point appears in white. Note that the **Add Point** option is available only when the dashboard is displayed at the bottom of the graphics window. You cannot increase the length of the spline by adding points before the start point and after the endpoint of the spline.

### Deleting Interpolation Points of a Spline

To delete a point or a vertex, invoke the dashboard. Now, select the vertex and hold down the right mouse button to invoke the shortcut menu. Choose the **Delete Point** option from the shortcut menu. The selected point is deleted. You can continue deleting vertices or points from a spline until only two end points are left in the spline.

### Creating a Control Polygon and Moving its Control Points



When you draw a spline, it is associated with a control frame. The vertices of its frame are called control points. To create a control polygon, choose the **Modify spline using control points** button from the dashboard. The control polygon is displayed in the graphics window. The control points of this polygon can be moved by dragging to modify the spline shape.

### Modifying the Dimensions of the Spline

The shape of the spline is controlled by the position of its interpolation points. Hence by modifying the dimensions, the position of the interpolation points are changed, which results in modification of the shape of the spline.



**Tip:** To dynamically modify the shape of the sketch, you need to select an entity of the sketch and drag the mouse to modify the sketch. Remember that if the selected entity is constrained then you cannot modify it. You can modify it only after disabling the constraints.

## WRITING TEXT IN SKETCHER ENVIRONMENT

There are various instances when a designer needs a text command to write text on the model. For example, for creating a label, model number, company name, and so on. In Pro/ENGINEER, you can write this text in the sketcher environment.



In the sketcher environment, the text is written using the **Create text as a part of a section** button from the **Sketcher Tools** toolbar. The following steps explain the procedure to write text in the sketcher environment:

1. Choose the **Create text as a part of a section** button from the **Sketcher Tools** toolbar; you are prompted to select the start point of line to determine the text height and orientation.
2. Specify the start point on the screen by clicking; you are prompted to select the second point of line to determine the text height and orientation.
3. Note that 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 upside down. Specify the second point on the screen by clicking; the **Text** dialog box is displayed, as shown in Figure 2-17.

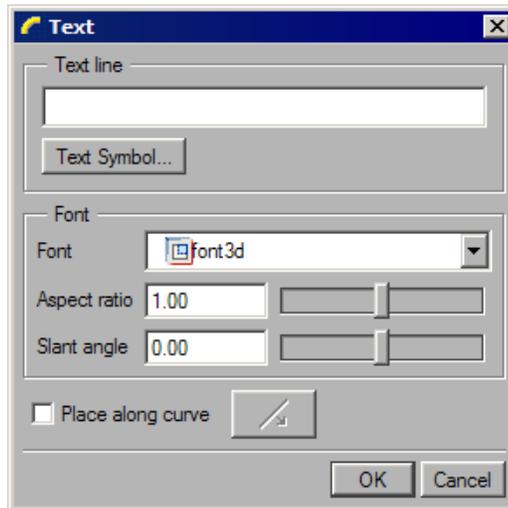


Figure 2-17 The Text dialog

After specifying the second point, a construction line is drawn having 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. If the construction line is drawn at an angle, then the text is written at that angle.

4. In the **Text Line** edit box, enter the text, which can be up to 79 characters. As you enter the text, the text is 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 in the **Text** dialog box to exit it.



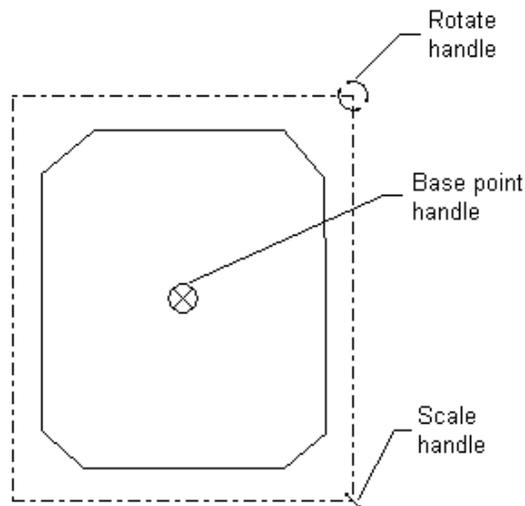
**Note**

*In later chapters of this book, you will learn that to enter the sketcher environment there are other methods also, other than entering through the **Sketch** mode.*

## SCALING AND ROTATING ENTITIES



The sketches can be scaled or rotated by using the **Scale and rotate selected entities** button from the **Sketcher Tools** toolbar. To invoke this button, select the sketch and then choose the arrow on the right of the **Mirror selected entities** button. This button is available on the flyout that is displayed. After choosing this tool button, the sketch that is composed of various entities now acts as a single entity. The sketch appears orange in color and is enclosed in a boundary box, as shown in Figure 2-18.

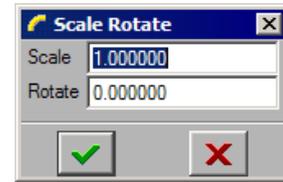


**Figure 2-18** Selected entities enclosed in a boundary box with three handles

There are three handles that facilitate in scaling and rotating the selected sketch. The rotate handle is used to dynamically rotate the selected entities. The scale handle is used to dynamically scale the selected entities. The base point handle is used to pick the sketch and place it at any other location in the graphics window. To change the location of any of the three handles, right-click on the handle. The handle symbol is attached to the cursor. Place the symbol at the desired location. The following steps explain the procedure to scale and rotate a sketch:

1. Select the sketch to be rotated and scaled, and then choose the black arrow on the right of the **Mirror selected entities** button from the **Sketcher Tools** toolbar.

- From the flyout, choose the **Scale and rotate selected entities** button; the **Scale Rotate** dialog box is displayed, as shown in Figure 2-19. This dialog box contains the **Scale** and **Rotate** edit boxes. The sketch can be scaled and rotated dynamically or by entering a value in these edit boxes.
- To dynamically rotate the sketch, select the rotate handle and then move the cursor; the sketch is rotated as you move the cursor. You can also enter the rotation angle in the **Rotate** edit box.



**Figure 2-19** The **Scale Rotate** dialog box

To scale the sketch, select the scale handle and then move the cursor. As you move the cursor, the sketch is scaled dynamically in the graphics window. You can also enter the scale value in the **Scale** edit box.

- After the sketch is rotated and scaled, choose the **Accept the changes and close the dialog** button in the **Scale Rotate** dialog box.

## COPYING SKETCHED ENTITIES IN SKETCH MODE

 The **Make a copy of selected entities** button in the **Sketcher Tools** toolbar is used to copy the entities in the sketcher environment. To invoke this button, select the sketch and then choose the arrow on the right of the **Mirror selected entities** button. Choose the **Make a copy of selected entities** button from the flyout that is displayed. The **Scale Rotate** dialog box is displayed. Using this dialog box, you can simultaneously copy, scale and rotate the selected entities. The following steps explain the procedure to copy the selected entities:

- Select the entities to be copied and then choose the black arrow on the right of the **Mirror selected entities** button from the **Sketcher Tools** toolbar. From the flyout, choose the **Make a copy of selected entities** button.

The copy of the selected entity is placed at the top left corner in the graphics window. This copy appears red in color. There are three handles in the copied entities that facilitate in sizing and orienting them. Figure 2-20 shows the copied sketch enclosed in a boundary with the three handles.

The **Scale Rotate** dialog box is also displayed. Use this dialog box to specify the scale factor and the rotation angle.

- Select the base point handle; the copied entities are attached to the cursor. Place the entities where needed.
- Finally, choose the **Accept the changes and close the dialog** button in the **Scale Rotate** dialog box.

## IMPORTING 2D DRAWINGS IN THE SKETCH MODE

The two-dimensional (2D) drawings when opened in the sketcher environment can be saved

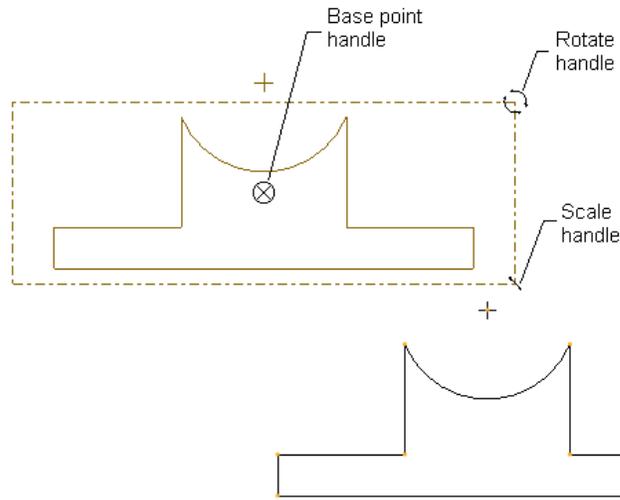


Figure 2-20 Original entities and the copied entities with three handles

in the *.sec* format. The *.sec* file can then be converted to a solid model. The **Data from file** option in the **Sketch** menu in the menu bar is used to import the 2D sketches. This option saves time in drawing the same or similar section again. The file formats from which the data can be imported are shown in Figure 2-21.



Figure 2-21 File formats

When you choose **Sketch** > **Data from file** from the menu bar, the **Open** dialog box is displayed, as shown in Figure 2-22. Use this dialog box to select and open the file.

In the **Open** dialog box, if you select a drawing file that is created in the **Drawing** mode of Pro/ENGINEER Wildfire 2.0, the draft entities in that file are imported. The selected drawing is opened in a sub window. You are prompted to select the entities to copy from the sub window. Select the draft entities and then press the middle mouse button. The selected entities are enclosed in a boundary and the **Scale Rotate** dialog box is displayed. Use this dialog box to set the scale and orientation of the sketch. Note that if the *.dwt* file does not consist of draft entities, no data is imported.

In the **Open** dialog box, if you select a *.sec* file that is created in the sketcher environment, the sketch is either displayed in the graphics window enclosed in a boundary or is directly placed in the graphics window. If there are no entities in the current sketcher environment, the selected sketch will be automatically placed in the graphics window. In case, there are some

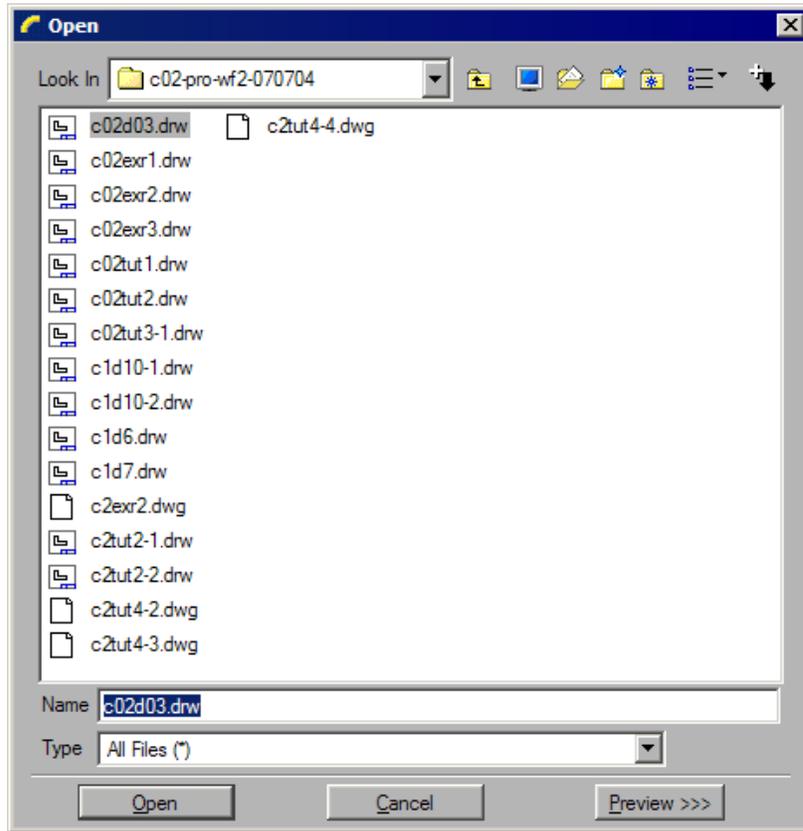


Figure 2-22 The **Open** dialog box

entities present in the current sketcher environment, the selected sketch is enclosed in a boundary and the **Scale Rotate** dialog box is displayed. Use this dialog box to set the scale and orientation of the sketch.

The section imported using the **Data from file** option in the current sketch is an independent copy. The imported section is no longer associated with the source section. The units, dimensions, grid parameters, and accuracy are acquired from the current sketch.



**Tip:** The display of vertices of the section, the display of dimensions, and the display of constraints can be turned on or off from the **Sketcher** toolbar. This toolbar is in the **Top Toolchest**.

## TUTORIALS

### Tutorial 1

In this Tutorial you will import an existing sketch that you had drawn in Tutorial 3 of Chapter 1. After placing the sketch, draw the keyway, as shown in Figure 2-23.

(Expected time: 15 min)

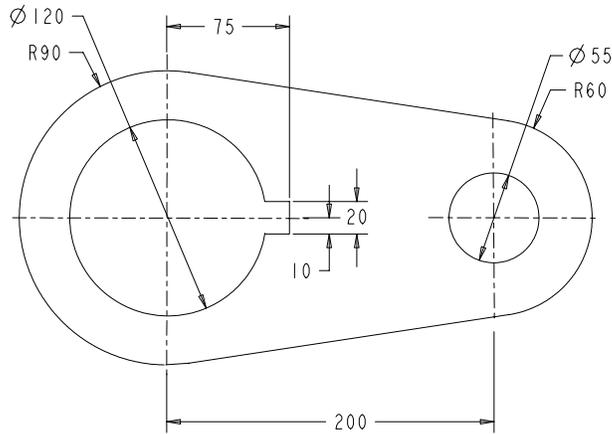


Figure 2-23 Sketch for Tutorial 1

The following steps are required to complete this tutorial:

- Start Pro/ENGINEER Wildfire 2.0.
- Set the working directory and create a new object file.
- Import the section by using the **Data from file** option, refer to Figure 2-24.
- Draw the keyway and dimension it, refer to Figures 2-25 and 2-26.
- Modify the dimensions, refer to Figure 2-27.
- Save the sketch and exit the sketcher environment.

### Starting Pro/ENGINEER

- Start Pro/ENGINEER Wildfire 2.0 by double-clicking on the Pro/ENGINEER icon on the desktop of your computer or by using the **Start** menu.

### Setting the Working Directory

When the Pro/ENGINEER session is started, the first task is to set the working directory. As mentioned earlier, working directory is a directory on your system where you can save the work done in the current session of Pro/ENGINEER. You can set any directory existing on your system as the working directory. Since this is the first tutorial of this chapter, you need to create a folder named **c02** in the *C:\ProE-WF-2.0* folder, if it does not exist.

- Choose the **Set Working Directory** option from the **File** menu. The **Select Working Directory** dialog box is displayed.
- Browse and select *C:\ProE-WF-2.0*.
- Choose the **New Directory** button in the **Select Working Directory** dialog box. The **New Directory** dialog box is displayed.

4. Type **c02** in the **New Directory** edit box. Choose **OK** from the dialog box. You have created a folder named *c02* in *C:\ProE-WF-2.0*.
5. Choose **OK** from the **Select Working Directory** dialog box. You have set the working directory to *C:\ProE-WF-2.0\c02*.

**Starting a New Object File**

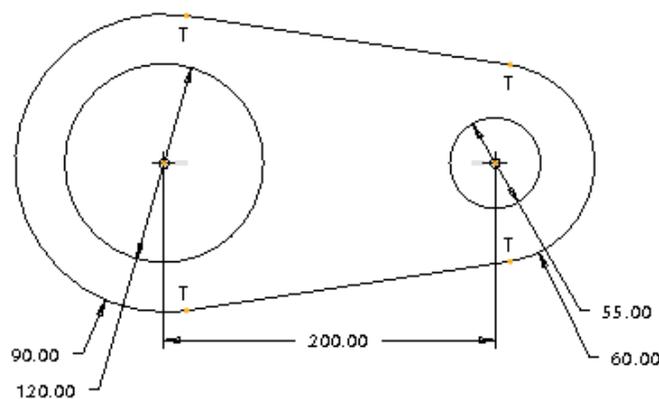
1. Choose the **Create a new object** button from the **File** toolbar; the **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box. A default name of the sketch appears in the **Name** edit box.
2. Enter *c02tut1* in the **Name** edit box. Choose the **OK** button.

You are in the sketcher environment of the **Sketch** mode. When you enter the sketcher environment, the Navigator is displayed to the left in the graphics window. Slide in the Navigator by clicking on the sash present on its right edge. Now, the drawing area is increased.

**Importing the Section**

1. Choose **Sketch > Data from file** from the menu bar; the **Open** dialog box is displayed with the working directory as the current directory.
2. Choose the **Up One Level** button and then open the *c01* directory. Make sure that the **sketch (\*.sec)** option is selected in the **Type** drop-down list. Select *c01tut3.sec* and choose the **Open** button from the **Open** dialog box. 

The sketch is placed in the graphics window, as shown in Figure 2-24.



*Figure 2-24 Sketch imported and placed in the current file*

## Drawing the Keyway

To create the keyway, you will sketch the keyway and then the portion of the circle that lies between the horizontal lines will be removed.

1. Choose the **Create 2 points line** button from the **Sketcher Tools** toolbar. 
2. Draw the keyway, as shown in Figure 2-25; the weak dimensions and constraints are automatically applied to the sketch of the keyway.

The horizontal lines of the keyway and the circle intersect at the points where the lines meet the circle. The intersection points are displayed in orange color. The portion of the circle that lies between the two horizontal lines of the keyway needs to be deleted from the circle.

3. Choose the **Zoom In** button from the **View** toolbar in the **Top Toolchest**; the cursor is converted into a magnifying glass symbol. 
4. Draw a window around the keyway to zoom in. Now, the display of the keyway is enlarged.
5. Choose the **Dynamically trim section entities** button from the **Sketcher Tools** toolbar. 
6. Click to select the part of the circle that lies between the two intersection points. The part of the circle that was between the two horizontal lines is deleted.
7. Choose the **Refit object to fully display it on the screen** button from the **View** toolbar to view the full sketch. 

## Dimensioning the Keyway

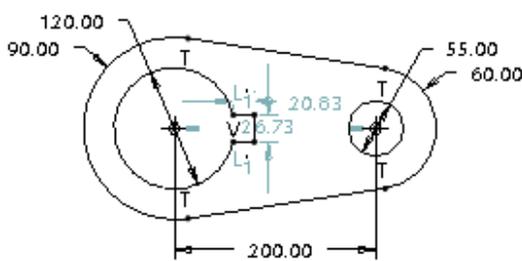
The dimensions that will be applied to the keyway are shown in Figure 2-25.

1. Choose the **Create defining dimensions** button from the **Sketcher Tools** toolbar. 
2. Dimension the keyway, as shown in Figure 2-26.

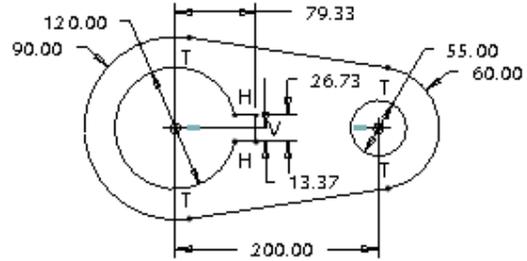
## Modifying the Dimensions

The dimensions of the keyway will be modified to the given dimension values.

1. Select the three dimensions of the keyway by using CTRL+left mouse button.
2. Choose the **Modify the values of dimensions, geometry of splines, or text entities** button from the **Sketcher Tools** toolbar. The **Modify Dimensions** dialog box is displayed. 
3. Clear the **Regenerate** check box and then modify the values of the dimensions. When you clear the check box, the sketch does not regenerate as you modify the dimensions.



**Figure 2-25** Sketch of the keyway with weak dimensions and constraints

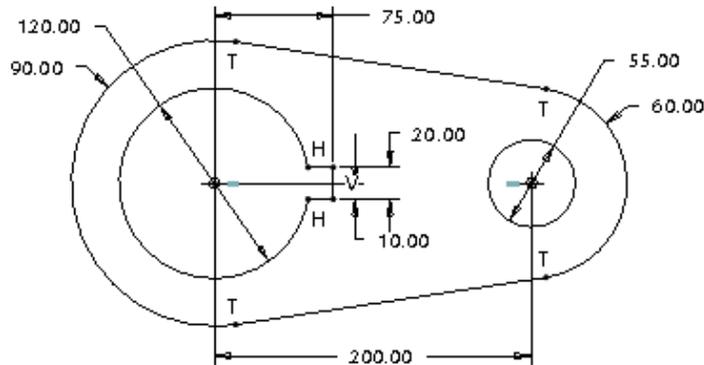


**Figure 2-26** Sketch after dimensioning the keyway

The dimension that you edit in the **Modify Dimensions** dialog box is enclosed in a yellow box in the sketch.

4. Modify all dimensions. Refer to Figure 2-23 for dimension values.
5. After the dimensions are modified, choose the **Regenerate the section and close the dialog** button from the **Modify Dimensions** dialog box. The message **Dimension modifications successfully completed** is displayed in the **Message Area**.

The sketch after modifying the dimension values of the sketch is shown in Figure 2-27.



**Figure 2-27** Sketch after modifying the dimensions

## Saving the Sketch

You have to save the sketch because you may need the sketch later.

1. Choose the **Save the active object** button from the **File** 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 **Window > Close** from the menu bar to exit the **Sketch** mode.

## Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 2-28. The sketch is shown in Figure 2-29. **(Expected time: 30 min)**

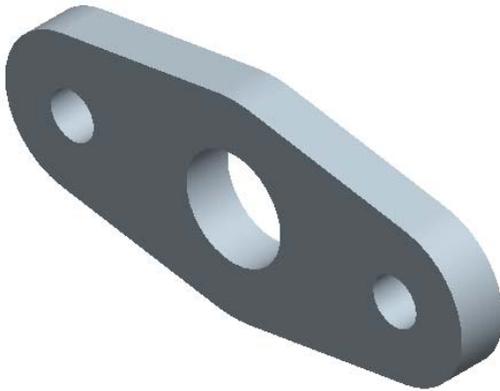


Figure 2-28 Model for Tutorial 2

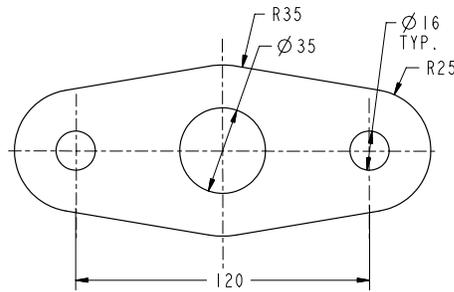


Figure 2-29 Sketch of the model

The following steps are required to complete this tutorial:

- a. Set the working directory and create a new object file.
- b. Draw the sketch using sketcher tools, refer to Figures 2-30 through 2-33.
- c. Apply the required constraints and dimensions to the sketched entities, refer to Figure 2-35.
- d. Modify the dimensions of the sketch and then save the sketch, refer to Figure 2-36.
- e. 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 Pro/Engineer is started, then you have to again set the working directory by following the steps given next.

1. Open the Navigator by clicking the top arrows on the left edge of the Pro/ENGINEER main window; the Navigator slides out.

2. Click on the plus symbol adjacent to the *ProE-WF-2.0* folder in the Navigator; the contents of the *ProE-WF-2.0* folder are displayed.
3. Now right-click on the *c02* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option. The working directory is set to *c02*.
4. Close the Navigator by clicking the sash on the right edge of the Navigator. The Navigator slides in.

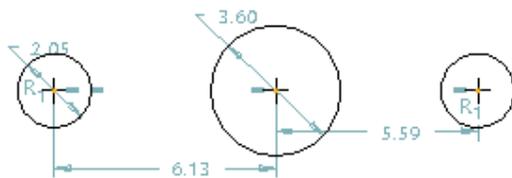
### Starting a New Object File

1. Start a new object file in the **Sketch** mode. Name the file as *c02tut2*.

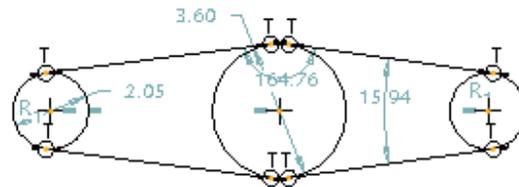
### Drawing the Sketch

To draw the outer loop, you need to draw three circles and then draw lines tangent to the three circles.

1. Choose the **Create circle by picking the center and a point on the circle** button. Draw the three circles, as shown in Figure 2-30. 
2. Choose the black arrow on the right of the **Create 2 points line** button to display a flyout. Choose the **Create lines tangent to 2 entities** button from this flyout. 
3. Select the left circle at the top and then select the middle circle at the top. A tangent is drawn from the top of the left circle to the top of the middle circle.
4. Select the right circle at the top and the middle circle at the top. A tangent is drawn from the top of the right circle to the top of the middle circle.
5. Similarly, using the **Create lines tangent to 2 entities** button, draw the other tangents from the bottom of the circles, as shown in Figure 2-31.



**Figure 2-30** Three circles with weak dimensions and constraints



**Figure 2-31** Lines joining the three circles with weak dimensions

### Trimming the Circles

The inner portions of the circles are not required. Therefore, you need to trim them.

1. Choose the **Dynamically trim section entities** button from the **Sketcher Tools** toolbar. 
2. Bring the cursor close to the right portion of the left circle. The right part of the circle turns cyan in color. Click on the right portion of the circle to delete it.
3. Similarly, trim the parts of the middle and the right circle that are not required. The sketch after trimming the circles is shown in Figure 2-32.

### Drawing the Circles

1. Choose the black arrow on the right of the **Create circle by picking the center and a point on the circle** button to display the flyout. From this flyout, choose the **Create concentric circle** button. You are prompted to select an arc.



**Tip:** While drawing a concentric circle, sometimes the circle snaps to the other circle or arc and it becomes difficult to draw a circle of the size you need. The snapping of the circle to the other circle or arc can be disabled. While drawing the circle when the circle snaps to the other circle, use **TAB+right-click** to disable the snapping or to disable the equal radii constraint that the system tends to apply.

2. Click on the left arc. Notice that when you move the mouse, a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button.



#### Note

You may need to zoom in to select the top arc in the next step.

3. Click on the top arc. Notice that when you move the mouse a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button to end circle creation.
4. Click on the right arc. Notice that when you move the mouse a circle appears. When  $R_2$  symbol appears on the circle, click to complete the circle. Press the middle mouse button to end circle creation.

The  $R_2$  symbol appears on both the left and the right circle indicating that their radii are same. The constraint symbols like  $R_1$  or  $R_2$  vary from sketch to sketch.

The sketch after drawing the three circles is shown in Figure 2-33.

### Applying the Constraints

1. Choose the **Impose sketcher constraints on the section** button from the **Sketcher Tools** toolbar. The **Constraints** dialog box is displayed. 

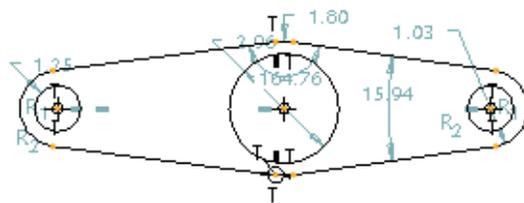
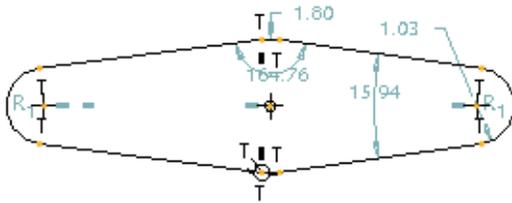


Figure 2-32 Sketch after trimming the circles

Figure 2-33 Sketch after drawing the three circles

2. Choose the **Create Equal Lengths, Equal Radii, or Same Curvature constraint** button from the **Constraints** dialog box. =
3. Select the left arc and then select the right arc to apply the equal radius constraint.
4. Select the left circle and the right circle to apply the equal radius constraint.
5. Use the CTRL+middle mouse button to zoom in and now select the top arc and the bottom arc to apply the equal radius constraint. If the constraint is already applied, the **Resolve Sketch** dialog box is displayed, as shown in Figure 2-34.

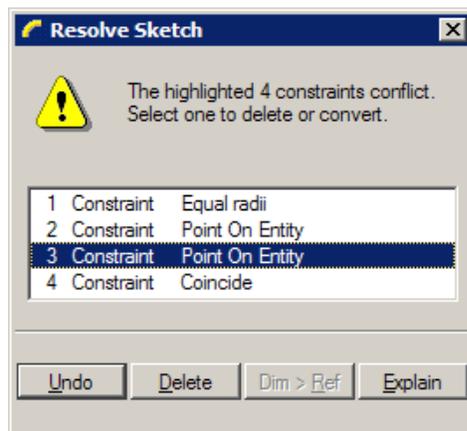


Figure 2-34 The **Resolve Sketch** dialog box

6. Select the third constraint in the **Resolve Sketch** dialog box and choose the **Delete** button. The constraint that was conflicting is deleted.
7. Choose the **Make two lines parallel** constraint button from the **Constraints** dialog box. //

8. Click to select the tangent line that connects the left circle and the middle circle at the top.
9. Click to select the tangent line that connects the right circle and the middle circle at the bottom.

It is evident from the parallel constraint symbol that the parallel constraint is applied to the two tangent lines in the sketch.

10. Similarly, apply the parallel constraint to the other set of tangent lines. The sketch after applying the constraints is shown in Figure 2-35.
11. Choose the **Refit object to fully display it on the screen** button from the **View** toolbar to view the full sketch in the graphics window. 

### Dimensioning the Sketch

Pro/ENGINEER applies the weak dimensions to the sketch automatically. These dimensions are not the dimensions that are needed because these dimensions will not help to machine the model. Therefore, you will dimension the sketch with the dimensions that will be used to machine the model.

1. Choose the **Create defining dimensions** button from the **Sketch Tool** toolbar. 
2. Select the center of right and left circles; the centers of the circles turn red in color. Now, using the middle mouse button, place the dimension below the sketch.

The rest of the weak dimensions are needed and therefore they will be modified.

### Modifying the Dimensions

All constraints and dimensions are applied to the sketch and now dimensions will be modified.

1. Select all dimensions using CTRL+ALT+A.
2. Choose the **Modify the values of dimensions, geometry of splines, or text entities** button. The **Modify Dimensions** dialog box is displayed. 
3. Clear the **Regenerate** check box and then modify the values of the dimensions. When you clear the check box, the sketch does not regenerate as you modify the dimensions.

The dimension that you edit in the **Modify Dimensions** dialog box is enclosed in a yellow box in the sketch.

4. Modify all dimensions. Refer to Figure 2-29 for dimension values.
5. After the dimensions are modified, choose the **Regenerate the section and close the dialog** button from the **Modify Dimensions** dialog box. The message **Dimension modifications successfully completed** is displayed in the **Message Area**.

The sketch after modifying the dimension values is shown in Figure 2-36.

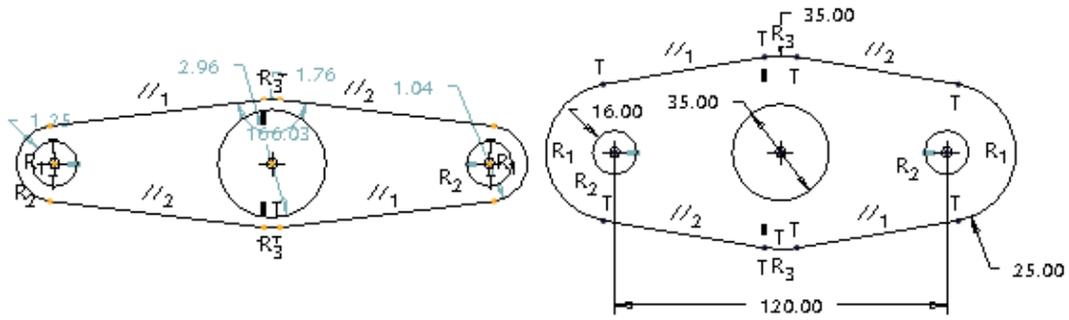


Figure 2-35 Sketch with constraints

Figure 2-36 Sketch after modifying the dimensions

### Saving the Sketch

1. Choose the **Save the active object** button from the **File** toolbar and save the sketch.

### Exiting the Sketch Mode

1. After saving the sketch, choose **Window > Close** from the menu bar to exit the **Sketch** mode.

## Tutorial 3

In this tutorial, you will draw the sketch for the model shown in Figure 2-37. The sketch is shown in Figure 2-38. **(Expected time: 30 min)**

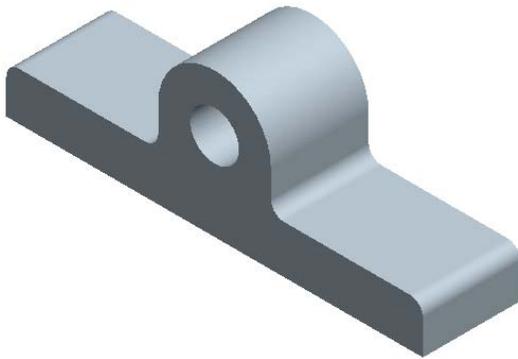


Figure 2-37 Model for Tutorial 3

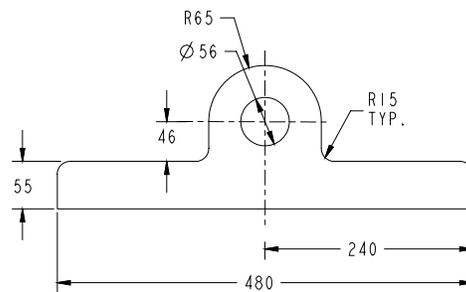


Figure 2-38 Sketch of the model

The following steps are required to complete this tutorial:

- a. Set the working directory and create a new object file.

- b. Draw the sketch using sketcher tools, refer to Figures 2-39 and 2-40.
- c. Apply fillets at two corners of the sketch, refer to Figures 2-41 and 2-42.
- d. Dimension the sketch, refer to Figure 2-43.
- e. Modify dimensions of the sketch, refer to Figure 2-44.
- f. 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 Pro/Engineer is started, then you have to again set the working directory by following the steps given next.

1. Open the Navigator by clicking the top arrows on the left edge of the Pro/ENGINEER main window; the Navigator slides out.
2. Click on the plus symbol adjacent to the *ProE-WF-2.0* folder in the Navigator. The contents of the *ProE-WF-2.0* folder are displayed.
3. Now right-click on the *c02* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; the working directory is set to *c02*.
4. Close the Navigator by clicking the top arrows on the right edge of the Navigator; the Navigator slides in.

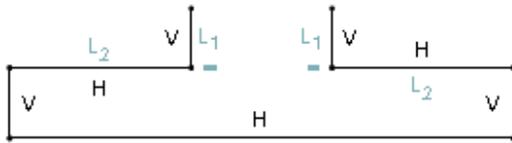
### Starting a New Object File

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

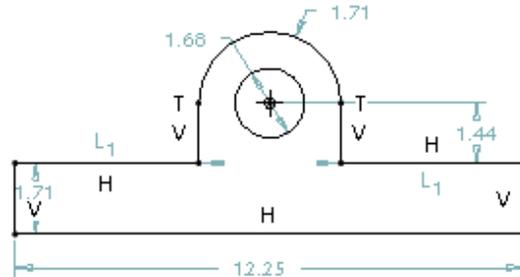
### Drawing the Sketch

1. Choose the **Create 2 points line** button from the **Sketcher Tools** toolbar.
2. Draw the lines, as shown in Figure 2-39.
3. Choose the **Create an arc by 3 points or tangent to an entity at its endpoint** button from the **Sketcher Tools** toolbar. 
4. Select the endpoint of the left vertical line as the start point of the arc. Complete the arc at the endpoint of the right vertical line.
5. Choose the black arrow on the right of the **Create circle by picking the center and a point on the circle** button to display the flyout. From this flyout, choose the **Create concentric circle** button; you are prompted to select an arc. 
6. Click on the arc; a red rubber-band circle appears. Size the circle by moving the cursor and click to complete it.

The sketch after drawing the circle is shown in Figure 2-40.



**Figure 2-39** Lines in the sketch with the dimensions turned off for clarity



**Figure 2-40** Sketch after drawing the arc and the circle



**Note**

Choose the **Toggle display of dimensions on/off** button from the **Sketcher** toolbar in the **Top Toolchest** to turn the dimensions on or off.

*Pro/ENGINEER does not have options like midpoint, endpoint, or center of. However, while drawing a sketch, these options are applied in the form of weak constraints as you sketch. For example, endpoint of any entity snaps the cursor when a new entity is to be drawn. The middle point constraint appears when you approximately bring the cursor near to the middle point of the line to draw a line.*

**Filleting the Corners**

1. Choose the **Create a circular fillet between two entities** button from the **Sketcher Tools** toolbar. You are prompted to select the two entities to be filleted. The corners that you need to fillet are shown in Figure 2-41.
2. Select the two entities one by one using the left mouse button. The corner of the selected lines are filleted. Fillet all required corners shown in Figure 2-41.



The sketch after filleting is shown in Figure 2-42.

**Applying the Constraints**

1. Choose the **Impose sketcher constraints on the section** button from the **Sketcher Tools** toolbar. The **Constraints** dialog box is displayed.
2. Choose the **Create Equal Lengths, Equal Radii, or Same Curvature constraint** button from the **Constraints** dialog box.
3. Click to select the fillets and apply the equal constraint to all fillets.



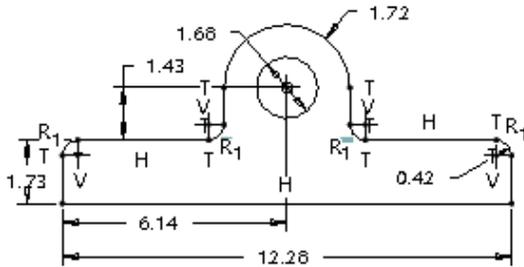


Figure 2-43 Sketch after dimensioning

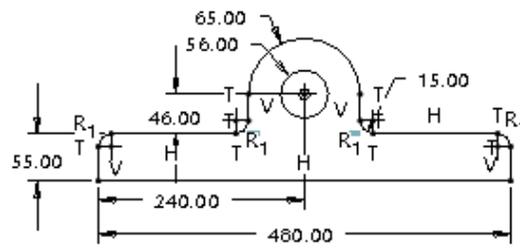


Figure 2-44 Sketch after modifying the dimensions

### Saving the Sketch

1. Choose the **Save the active object** button from the **File** toolbar and save the sketch.

### Exiting the Sketch Mode

1. Choose **Window > Close** from the menu bar to exit the **Sketch** mode.

## Self-Evaluation Test

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

1. You can increase the length of the spline by adding points before the start point and after the endpoint of the spline. (T/F)
2. While copying sketched entities, the **Scale Rotate** dialog box is also displayed. (T/F)
3. When you modify a weak dimension it becomes strong. (T/F)
4. You can dimension the length of a centerline. (T/F)
5. The font of the text written in the sketcher environment is fixed and cannot be modified. (T/F)
6. The \_\_\_\_\_ of a spline can be dynamically modified.
7. The \_\_\_\_\_ dialog box is used to modify dimensions.
8. The display of dimensions and constraints can be turned on/off by choosing the \_\_\_\_\_ and \_\_\_\_\_ buttons respectively from the **Sketcher** toolbar.
9. The \_\_\_\_\_ button is used to rotate selected entities.

10. You can delete the entities by selecting them and then using the \_\_\_\_\_ key on the keyboard.

### Review Questions

Answer the following questions:

- How many handles are displayed in the graphics window while scaling and rotating entities?
  - one
  - two
  - three
  - four
- Which of the following mouse buttons is used to place the dimension?
  - left
  - middle
  - right
  - mouse is not used for dimensioning
- Which of the following is the default font for the text in the sketcher environment?
  - font
  - filled
  - font3d
  - isofont
- Which of the following toolbar is used to toggle the display of dimensions and constraints in the sketcher environment?
  - Sketcher Tools**
  - Sketcher**
  - File**
  - Edit**
- In which type of dimensioning the **Dim Orientation** dialog box is displayed while dimensioning the arcs and circles?
  - Normal**
  - Perimeter**
  - Baseline**
  - None of the above
- While placing a section in a new sketch, use the right mouse button to place the section. (T/F)
- Elliptical fillets can be created in Pro/Engineer . (T/F)
- While creating text in the sketcher environment, you need to draw a construction line that will define the height of the text. (T/F)
- You can modify the dimensions dynamically. (T/F)
- The modification of a spline is possible by moving its interpolation points. (T/F)

**Exercises**

**Exercise 1**

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

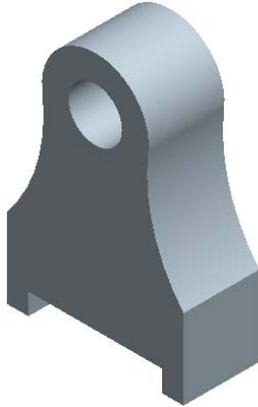


Figure 2-45 Solid model for Exercise 1

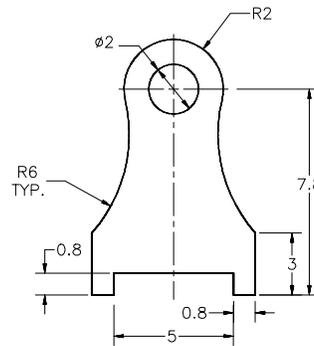


Figure 2-46 Sketch of the model

**Exercise 2**

In this exercise, you will draw the sketch for the model shown in Figure 2-47. The sketch is shown in Figure 2-48. **(Expected time: 15 min)**

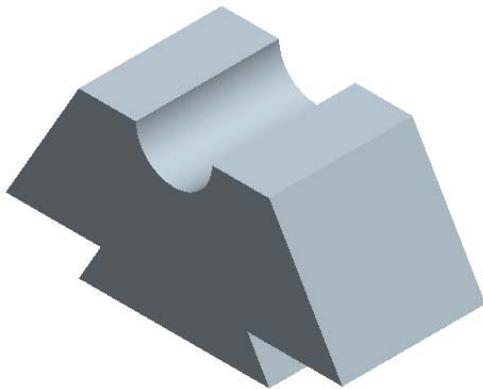


Figure 2-47 Solid model for Exercise 2

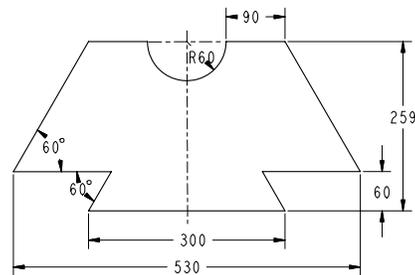


Figure 2-48 Sketch of the model

**Exercise 3**

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

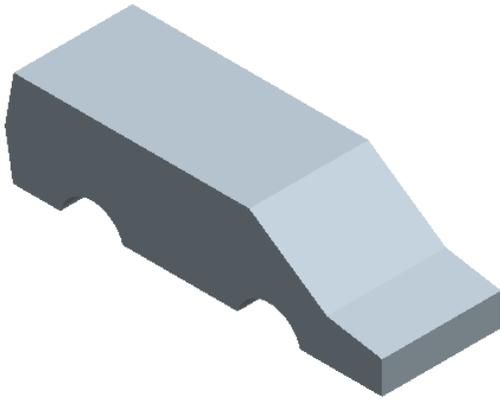


Figure 2-49 Solid model for Exercise 3

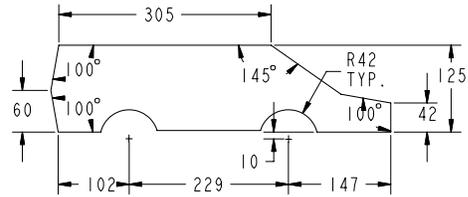


Figure 2-50 Sketch for the model

Evaluation chapter. Logon to [www.cadcam.com](http://www.cadcam.com) for more details

#### Answers to the Self-Evaluation Test

1 - F, 2 - T, 3 - T, 4 - F, 5 - F, 6 - shape, 7 - Modify Dimensions, 8 - Toggle display of dimensions on/off, Toggle display of constraints on/off, 9 - Scale and rotate selected entities, 10 - DELETE.