

Chapter 2

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 the Modify Dimensions dialog box.
- Edit the geometry of a sketch by trimming.
- Mirror a sketch.
- Use the drawing display options.

THE SKETCH MODE

Almost all models designed in Pro/ENGINEER Wildfire 4.0 consist of datums, sketched features, and placed features. For creating datums and placed features, you do not need to draw sketches. However, to create a 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; 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 4.0, 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 **New** button from the **File** toolbar; the **New** dialog box will be displayed with other Pro/ENGINEER modes. Select the **Sketch** radio button to start a new file in the **Sketch** mode, see Figure 2-1; 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.

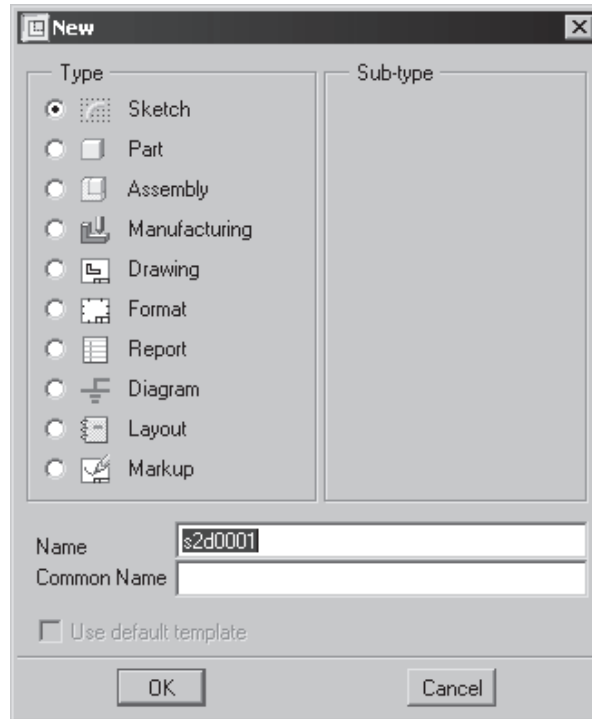


Figure 2-1 The New dialog box

THE SKETCHER ENVIRONMENT

When you invoke the **Sketch** mode, the initial screen appearance is similar to the one shown in Figure 2-2. This figure also shows the **Sketcher Tools** toolbar that will be displayed on the right side of the drawing area. The buttons 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 turned 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 drawing area. The options in these shortcut menus vary depending on the item selected. These shortcut menus also contain the tools to draw the sketches.



Note

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

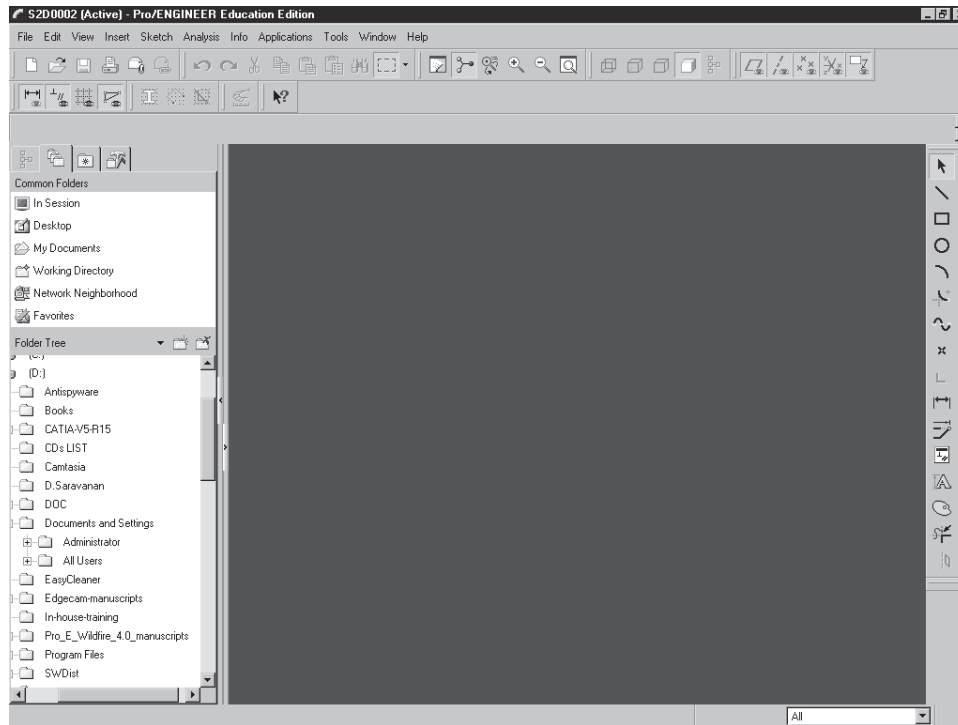


Figure 2-2 Initial screen appearance in the **Sketch** mode

The navigator is displayed on the left of the drawing area. In navigator, the **Folder Browser** tab is activated by default. It covers a part of the drawing area and therefore, the drawing area is decreased. You can increase the drawing area by clicking on the navigator sash, which is present on the right edge of the navigator.



Note

The **Folder Browser** tab is divided into two sections, **Common Folders** and **Folder Tree**. The functions of the **Common Folders** and the **Folder Tree** sections have already been discussed in Chapter 1.

WORKING WITH THE SKETCH IN THE SKETCH MODE

When you invoke the sketcher environment, the **One-by-One** button is chosen by default. The **One-by-One** button is available in the **Sketcher Tools** toolbar. 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.

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 drawing area. Note that, once the shortcut menu will be 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, available on the **Right Toolchest**, contains the buttons to draw a sketch, dimension it, and modify the dimensions. In this section, you will learn how to draw sketched entities using the button 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 **Point** 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 drawing area.



Note

To increase the number of visible command prompt lines in the message area, select the bottom sash of the message area using the left mouse button and drag it downward, 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 buttons available in the **Sketcher Tools** toolbar. To view these buttons, choose the black arrow on the right of the **Line** button; the flyout appears with three buttons. The first button is the **Line** button. This button is used to create a line by selecting two points in the drawing area. The second button on the flyout is the **Line Tangent** button. This button is used to create a tangent between two entities. The third button is the **Centerline** button. This button is used to create a centerline by selecting two points in the drawing area. The centerline is used for creating revolved features, mirroring, and so on.

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

Drawing a Line Using the Line Button



The following steps explain the procedure to create a line using the **Line** button:

1. Choose the **Line** button. Click in the drawing area 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 later in this chapter.
2. After specifying the start point for the line, move the cursor in the drawing area 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 the line creation, use the middle mouse button.

**Note**

If you draw a single line, the color of the line drawn 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 drawing area. After drawing a line, weak dimensions are applied to the sketch and they appear in gray color. These weak dimensions are applied automatically to the sketched entities as you draw them. The concept of weak dimensions is discussed later in this chapter.

Drawing a Line Using the Line Tangent Button



The **Line Tangent** 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 button:

1. Choose the black arrow on the right of the **Line** button and then choose the **Line Tangent** button from the **Sketcher Tools** toolbar; you will be prompted to select the start location on an 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 will be prompted to select the end location on an 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 will be prompted to select an entity in the sketcher environment, the **Select** message box will be displayed. You can ignore this dialog box because it appears automatically and disappears without any confirmation.*

Drawing a Line Using the Centerline Button



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

The steps discussed next explain the procedure to draw a centerline:

1. In the **Sketcher Tools** toolbar, choose the black arrow on the right of the **Line** button and then choose the **Centerline** button; you will be prompted to select the start point.
2. Click in the drawing area to specify the start point. Now, you will be prompted to select the end point.
3. Click in the drawing area to specify the endpoint. A centerline of infinite length is drawn.

Drawing a Rectangle



The following steps explain the procedure to create a rectangle using the **Rectangle** button:

1. Choose the **Rectangle** button from the **Sketcher Tools** toolbar; you will be 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 drawing area 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 buttons to draw a circle and one button to draw an ellipse. To view the buttons available to draw circles and ellipses, choose the black arrow on the right of the **Center and Point** button. The flyout appears with five buttons. The procedures to create a circle and an ellipse using various buttons are discussed next.

Drawing a Circle Using the Center and Point Button



As the name suggests, the **Center and Point** button 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 button.

1. Choose the **Center and Point** button; you will be prompted to select the center of a circle.
2. Click in the drawing area to specify the center point of the circle; you will be 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 2-3 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.

To create a construction circle, draw a circle or select a previously drawn circle. Then, hold the right mouse button to invoke the shortcut menu, as shown in Figure 2-4. Choose the **Construction** option from the shortcut menu. The circle appears in yellow color with a dashed line style, indicating that it is a construction circle.

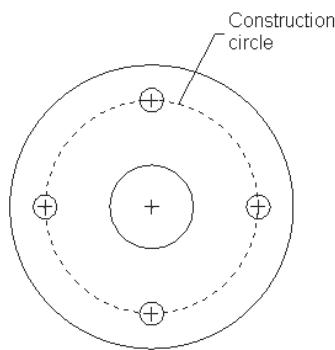


Figure 2-3 Sketch of a flange

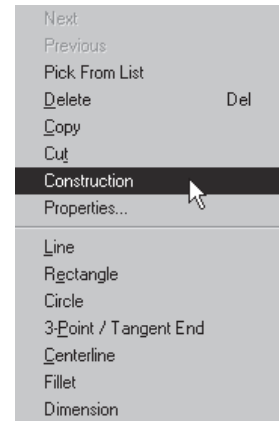


Figure 2-4 The **Construction** option in the shortcut menu

Drawing a Circle Using the Concentric Button



The following steps explain the procedure to draw a concentric circle using the **Concentric** button:

1. Choose the **Concentric** button from the flyout in the **Sketcher Tools** toolbar; you will be prompted to select an arc to determine the center. You can select an arc or a circle to specify the center point.
2. Click on an arc or a circle to determine the concentricity of the circle to be drawn. Move the mouse to size the circle.
3. After sizing the circle, finish the circle creation by pressing the middle mouse button.



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.

Drawing a Circle Using the 3 Point Button



The following steps explain the procedure to draw a circle using the **3 Point** button:

1. Choose the **3 Point** button from the flyout in the **Sketcher Tools** toolbar; you will be prompted to specify the first point on the circle.
2. Click for the first point at the desired location in the drawing area; you will be prompted to select the second point on the circle. Move the cursor and click to select the second point in the drawing area.
3. As soon as you select the second point, a yellow rubber-band circle appears with the cursor attached to it. You will be 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 prompted again to select the first point on the circle to draw the next circle.
4. You can press the middle mouse button to finish the creation of the circle. Also, you can press the middle mouse button at any stage in the circle creation to abort it.

Drawing a Circle Using the 3 Tangent Button



The **3 Tangent** button is used to draw a circle that is tangent to three existing entities. This button references other entities to draw a circle. The circle created using this button is drawn irrespective of the points selected on the entities. The following steps explain the procedure to draw a circle using the **3 Tangent** button:

1. Choose the **3 Tangent** button from the flyout in the **Sketcher Tools** toolbar; you will be 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 will be prompted to select the end location on an arc, circle, or line. Select the second tangent entity. Next, you will be prompted to select the third location on an arc, circle, or line. Select the third tangent entity; 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 Ellipse Button



The following steps explain the procedure to draw an ellipse using the **Ellipse** button:

1. Choose the **Ellipse** button from the flyout in the **Sketcher Tools** toolbar; you will be prompted to specify the center of the ellipse.
2. Click at the desired location in the drawing area to specify the center point; a yellow rubber-band ellipse appears with the cursor attached to the ellipse. Move the cursor in the drawing area to size the ellipse.
3. An ellipse is drawn when you click to specify the second point and the dimension for the major radius and the minor radius will be displayed in gray color. The gray color indicates that it is weak dimension.

Drawing an Arc

To draw an arc there are five buttons in the **Sketcher Tools** toolbar. To view them, choose the black arrow on the right of the **3-Point / Tangent End** button; the flyout appears with five buttons. The procedures to draw arcs using various buttons in the flyout are discussed next.

Drawing an Arc Using the 3-Point / Tangent End Button



The **3-Point / Tangent End** button is used to draw arcs that are tangent from the endpoint of an existing entity, or by defining three points in the drawing area.

When you choose this button to draw an arc from an endpoint, the **Target** symbol will be displayed as soon as you select the endpoint. This **Target** symbol is a green colored 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 **3-Point / Tangent End** button from the **Sketcher Tools** toolbar; you will be prompted to select the start point of the arc.
2. Specify three points in the drawing area 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 2-5.

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 2-6.

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

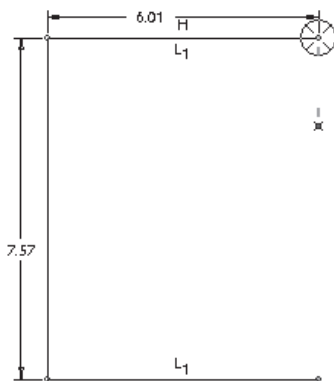


Figure 2-5 Cursor moved out of **Target** symbol perpendicular to the endpoint

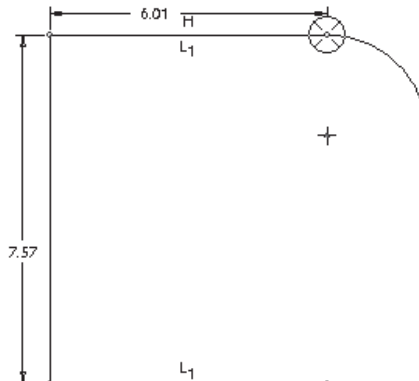


Figure 2-6 Cursor moved out of the **Target** symbol along the tangent direction



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

Drawing an Arc Using the Concentric Button



The **Concentric** button 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 button:

1. Choose the black arrow on the right of the **3-Point / Tangent End** button, the flyout appear with five button. Choose the **Concentric** button from the flyout; you will be 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 will be 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 will be 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 Center and Ends Button



The following steps explain the procedure to draw an arc using the **Center and Ends** button:

1. Choose the **Center and Ends** button from the flyout in the **Sketcher Tools** toolbar; you will be prompted to select the center of the arc.
2. Click to specify a center point for the arc in the drawing area; a yellow colored center mark appears at that point. Now, you will be 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 will be 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 3 Tangent Button



The **3 Tangent** button 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 button:

1. Choose the black arrow on the right of the **3-Point / Tangent End** button and then choose the **3 Tangent** button from the flyout in the **Sketcher Tools** toolbar; you will be 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 will be prompted to select the end location on an arc, circle, or line.
3. After doing so, you will be prompted to select a third location on an arc, circle, or line. Select a third entity. An arc is drawn tangent to the three selected entities.
4. You can continue drawing arcs or press the middle mouse button to abort arc creation.

Drawing an Arc Using the Conic Button



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

1. Choose the **Conic** button from the flyout in the **Sketcher Tools** toolbar; you will be prompted to specify the first endpoint of the conic entity.
2. Specify the first point in the drawing area; you will be 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 will be 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 drawing area, 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 **Normal** 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 2-7. This menu contains different dimensioning options.

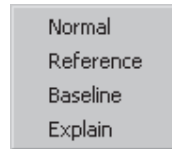


Figure 2-7 Dimensioning options



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 **Sketch** menu.

Converting a Weak Dimension into a Strong Dimension

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 from the drawing area. The selected dimension will be highlighted in red. Press and hold the right mouse button to invoke the shortcut menu, as shown in Figure 2-8. Choose the **Strong** option from the shortcut menu.

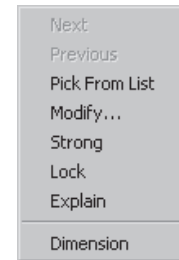


Figure 2-8 Shortcut menu to convert the weak dimensions to strong

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 dimension is made permanent.

Dimensioning a Sketch Using the Normal Button or the Normal Option



The **Normal** button or the **Normal** option is used for normal dimensioning of the sketch. The following steps explain the procedure to dimension a sketch using this option:

1. Choose the **Normal** 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 3.

DIMENSIONING THE BASIC SKETCHED ENTITIES

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

Linear Dimensioning of 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 will be displayed on the screen.

Figure 2-9 explains the three possible orientations of dimension that can be displayed when you dimension a line.

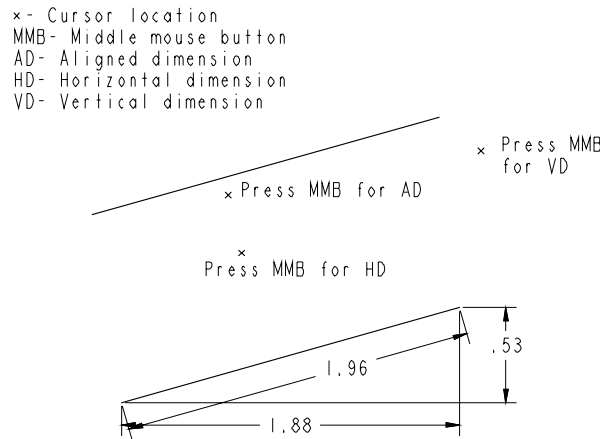


Figure 2-9 Approximate locations of the cursor to achieve different dimensions



Note

It is not possible to dimension a line in three orientations simultaneously in the sketcher environment. The dimensions in Figure 2-9 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 2-10. 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 will be displayed, as shown in Figure 2-11. The same diameter dimensioning technique can also be used for arcs.

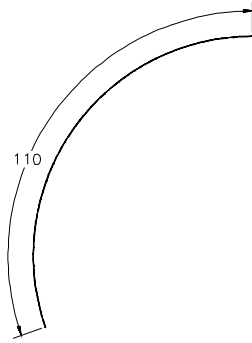


Figure 2-10 Angular dimensioning

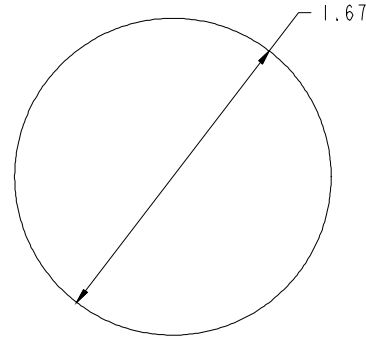


Figure 2-11 Diameter dimensioning

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 will be displayed, as shown in Figure 2-12.

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 will be displayed, as shown in Figure 2-13. This dimension represents the diameter of a revolved section.

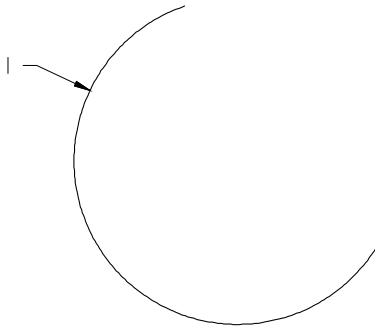


Figure 2-12 Radial dimensioning

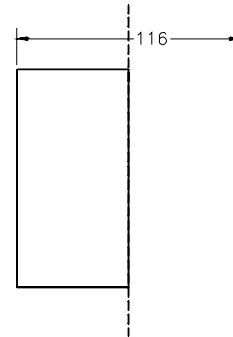


Figure 2-13 Dimensioning for revolved sections



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 **Constrain** button from the **Sketcher Tools** toolbar to display the **Constraints** dialog box. This dialog box is shown in Figure 2-14.

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 in the **Constraints** dialog box are discussed next.

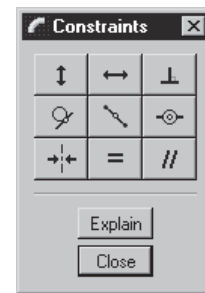


Figure 2-14 The Constraints dialog box

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 will be 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 will be prompted to select two lines to make their lengths equal, or you will be 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 will be 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 will be 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.

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 Button



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 keys or define a window to select the dimensions in the sketch. Choose the **Modify** button from the **Sketcher Tools** toolbar to modify the dimensions; the **Modify Dimensions** dialog box will be displayed, as shown in Figure 2-15.

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.

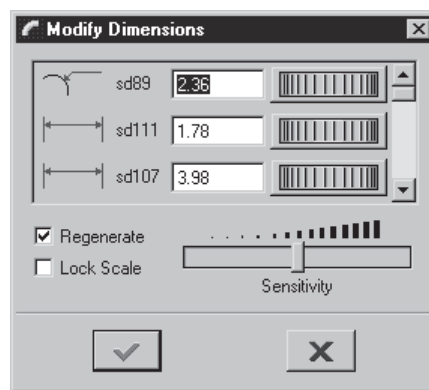


Figure 2-15 The *Modify Dimensions* dialog box



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

will be 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 occurs, the **Resolve Sketch** dialog box will be displayed, as shown in Figure 2-16 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 drawing area is enclosed in a yellow box. The buttons available in the **Resolve Sketch** dialog box are discussed next.

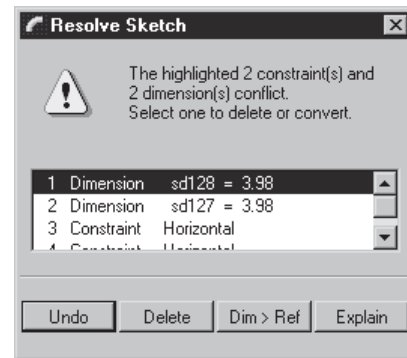


Figure 2-16 The *Resolve Sketch* dialog box

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 the feature creation.

Explain

When you choose the **Explain** button, the system provides you with the information about the selected constraint or dimension. The information will be 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 drawing area 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 drawing area, 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.

**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

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

Delete Segment Button

This button deletes the selected entities. After choosing the **Delete Segment** 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 button also trims entities that extend beyond the point of intersection.

Corner Button



The **Corner** 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 button:

1. Choose the black arrow on the right of the **Delete Segment** button to display the flyout. From this flyout, choose the **Corner** button; you will be prompted to select two entities to be trimmed.
2. Click to select the two entities on the sides that you want to keep after trimming, see Figure 2-17. These two entities must be intersecting entities. The entities are trimmed from the point of intersection.

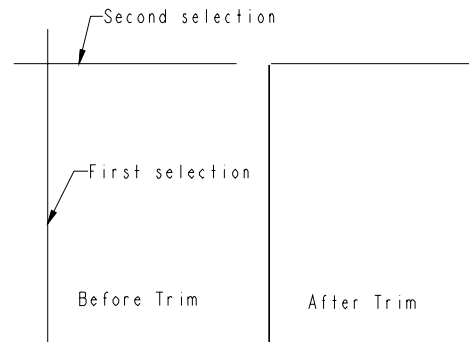


Figure 2-17 Trimming the lines

Divide Button



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

When you choose the black arrow on the right of the **Delete Segment** button, a flyout is displayed. You can choose the **Divide** button from this flyout.

The following steps explain the procedure to divide an entity:

1. Choose the **Divide** button from the flyout; you will be 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 small entities.

MIRRORING THE SKETCHER ENTITIES



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

The following steps explain the procedure to mirror the 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** button from the **Sketcher Tools** toolbar. You will be 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.

PALETTE BUTTON



This button helps you insert certain standard or user defined features such as polygons, profiles, shapes, stars, and other previously created sketches in the **Sketch** mode, thus minimizing the time for repetitive sketching.

The steps that explain the procedure to insert a foreign entity in the **Sketch** mode are given next.

1. Choose the **Palette** button from the **Sketcher Tools** toolbar; the **Sketcher Palette** dialog box will be displayed, as shown in the Figure 2-18. The options in this dialog box are used to insert a previously created sketch.



Figure 2-18 The **Sketcher Palette** dialog box



Note

If you have selected a working directory which contains only .sec files, then a tab with the name of the working directory will be available in the **Sketcher Palette** dialog box. Also, this tab is chosen by default. On the other hand, if you have selected a working directory which contains other types of files, then the same tab will be displayed in the end. In such cases, the **Polygon** tab will be chosen by default.

- 2.. For inserting a sketch from the **working directory** tab, double click on the sketch and click anywhere in the drawing area; the **Scale Rotate** dialog box will be displayed, as shown in Figure 2-19.

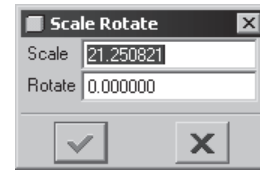


Figure 2-19 The **Scale Rotate** dialog box

3. Enter the scale value in the **Scale** edit box and the rotational angle value in the **Rotate** edit box. Next, choose the **Accept changes and close the dialog** button to exit the **Scale Rotate** dialog box.
4. Choose the **Close** button from the **Sketcher Palette** dialog box to accept the sketch inserted. Otherwise, repeat the steps 2-4 to continue to insert more sketches in the drawing area.

Similarly, you can add the sketches from the **Polygon, Profiles, Shapes, and Stars** tab.

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 the later chapters.

Zoom In



This button 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 drawing area. Note that when you enlarge the view of the drawing, the original size of the entities is not changed. To exit zoom tool right-click in the drawing area.

Zoom Out



This button 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 drawing area is reduced.

Refit



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.

Repaint



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 drawing area using the **Repaint** button. This option is extensively used while 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 2-20. The sketch is shown in Figure 2-21. **(Expected time: 30 min)**



Figure 2-20 Model for Tutorial 1

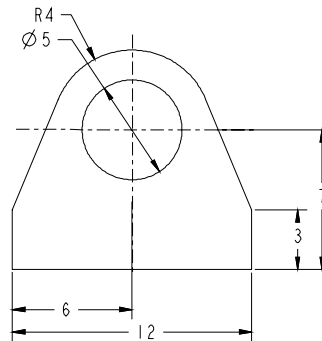


Figure 2-21 Sketch of the model

The following steps are required to complete this tutorial:

- Start Pro/ENGINEER Wildfire 4.0 session.
- Set the working directory and create a new sketch file.
- Draw lines using the line tool, refer to Figures 2-23 and 2-24.

- d. Draw an arc and a circle, refer to Figures 2-25 and 2-26.
- e. Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 2-27
- f. Save the sketch and close the file.

Starting Pro/ENGINEER

1. Start Pro/ENGINEER Wildfire 4.0 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 existing directory on your system as the working directory. Because this is the first tutorial of this chapter, you need to create a folder named *c02*, if it does not exist.

1. Choose the **Set Working Directory** option from the **File** menu; the **Select Working Directory** dialog box is displayed.
2. Now, you need to select *C:\ProE-WF-4.0*. If this folder is not existing, then create it prior to setting the working directory.

For selecting *C:\ProE-WF-4.0*, first click on the double arrow at the upper left corner of the **Select Working Directory** dialog box to display the flyout. Next, select the **C:** option from the flyout, as shown in Figure 2-22; the contents of the **C** drive folder is displayed. Now, select the *ProE-WF-4.0* folder from it.

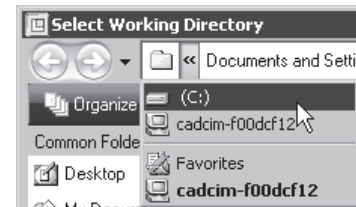


Figure 2-22 The flyout displayed by clicking on the double arrow

3. Choose the **Organize** button from the **Select Working Directory** dialog box to display the flyout. From the flyout, choose the **New Folder** option; the **New Folder** dialog box is displayed.
4. Enter *c02* in the **New Directory** edit box and choose **OK** from the **New Folder** dialog box. You have created a folder named *c02* in *C:\ProE-WF-4.0*.
5. Choose **OK** from the **Select Working Directory** dialog box. You have set the working directory to *C:\ProE-WF-4.0\c02*. A message **Successfully changed to C:\ProE-WF-4.0\c02 directory** is displayed in the message area.

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 **New** button from the **File** toolbar or press CTRL+N; the **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box. The default name of the sketch appears in the **Name** edit box.



2. Enter *c02tut1* 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 on the left in the drawing area.

3. Slide 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 **Line** button from the **Sketcher Tools** toolbar.
2. Specify the start point by clicking to the right in the drawing area. 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 drawing area 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, but it is not visible in the drawing area until the line creation is active.

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.

Notice that when the cursor is moved horizontally toward the left, a red-colored constraint, **H** appears in the drawing area next to the line. This indicates that if you draw a line now, a horizontal constraint will be applied to the line.

5. After you get the desired size of the line, click to end the line. The horizontal constraint **H** is applied to the line, but it is not visible in the drawing area until the line creation is active.
6. Move the cursor upward in the drawing area; 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, an **L₁** symbol is displayed on both the vertical lines. This symbol suggests that the equal length constraint is applied to the two vertical lines.
7. When the **L₁** constraint appears on the vertical line, click to specify the endpoint of the vertical line. 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.

8. Move the cursor to size the line and specify the endpoint of the left inclined line, as shown in Figure 2-23.
9. Press the middle mouse button to end the line creation.
10. 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.
11. Size the inclined line and specify the endpoint of the right inclined line. Press the middle mouse button twice to end the line creation.

Figure 2-24 shows the lines that you have drawn. Notice that when you end the line creation by pressing the middle mouse button twice, all the constraints that you have drawn become visible, as shown in Figure 2-24. Now, you need to draw the arc and the circle.

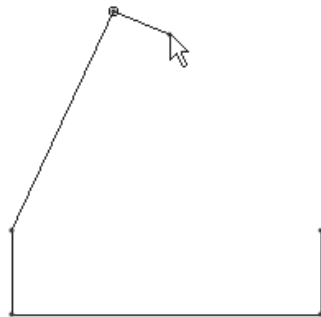


Figure 2-23 Partial sketch with left inclined line

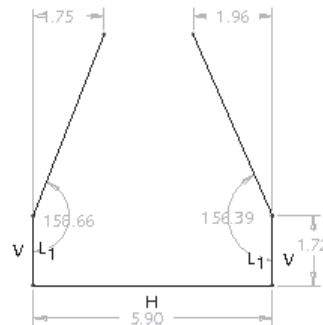


Figure 2-24 Partial sketch with weak dimensions



Note

The horizontal constraint **H** and the vertical constraint **V** appear in yellow color. The yellow color of the constraint indicates that this constraint is strong. This means you cannot change the orientation of the line until you delete the constraint that is applied on the line. The constraints displayed in grey color indicates that they are weak constraints, refer to Figure 2-24.

Drawing the Arc

1. Choose the **3-Point / Tangent End** 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 the arc creation. You will notice that the tangent constraint with a symbol **T** appears at the endpoints of the arc, as shown in Figure 2-25. Press the middle mouse button to end the 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 2-24, there are some weak dimensions that are not displayed in Figure 2-25. This is because the weak dimensions get 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 **Constrain** button from the **Sketcher Tools** toolbar (see page 2-16 for more information).*

Drawing the Circle

1. Choose the black arrow on the right of the **Center and Point** button to display the flyout. From this flyout, choose the **Concentric** button; you are prompted to select an arc. 
2. Select the arc 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 the circle creation. The sketch is complete and appears similar to that shown in Figure 2-26.

Dimensioning the Sketch

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

1. Choose the **Normal** 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 2-26.

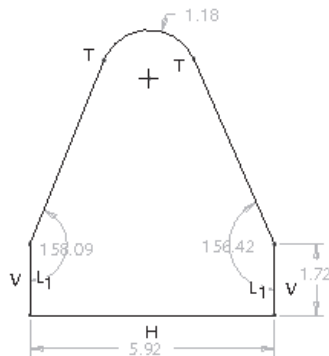


Figure 2-25 Sketch with arc

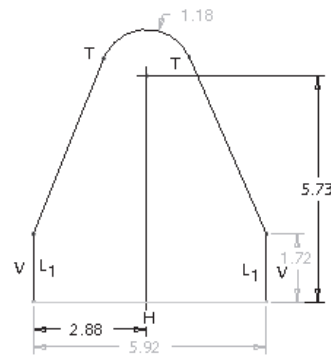


Figure 2-26 Sketch with all entities, weak dimensions, and weak constraints

Modifying the Dimensions

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

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



Note

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

3. When all dimensions turn red in color, choose the **Modify** 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 drawing area.

5. Modify all dimensions according to the dimensions shown in Figure 2-21. 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 completed and is shown in Figure 2-27.

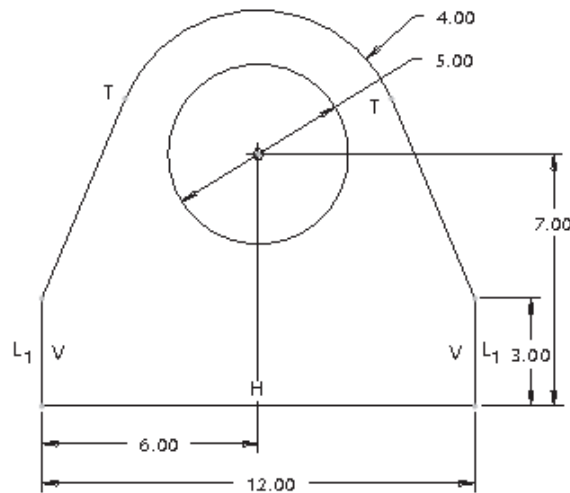


Figure 2-27 The complete sketch with dimensions and constraints



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

Now, the sketch needs to be saved because you may need the sketch later in the **Part** mode to create a 3D model.

1. Choose the **Save** 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 2-28. The sketch is shown in Figure 2-29. For your reference, all entities in the sketch are labeled alphabetically.

(Expected time: 30 min)



Figure 2-28 Model for Tutorial 2

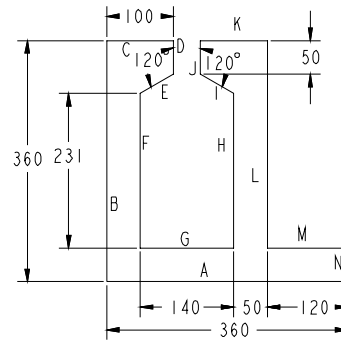


Figure 2-29 Sketch of the model

The following steps are required to complete this tutorial:

- Set the Working directory and create a new object file.
- Draw the sketch using the line tool, refer to Figure 2-30.
- Dimension the required entities and then modify the dimensions of the sketch, refer to Figures 2-31 and 2-32.
- 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 set the working directory again by following the steps given next.

- Open the Navigator (if it is in collapsed state) by clicking the top arrows on the left edge of the Pro/ENGINEER main window; the Navigator slides out. In the Navigator, the **Folder Tree** is displayed at the bottom. Click on the black arrow available on the right of the **Folder Tree**; the **Folder Tree** expands.
- Click on the plus sign adjacent to the *ProE-WF-4.0* folder in the Navigator; the contents of the *ProE-WF-4.0* folder are displayed.
- 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*.
- Close the Navigator by clicking on the sash located at the right edge of Navigator. The Navigator slides in.

Starting a New Object File


- Choose the **New** 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. The default name of the sketch appears in the **Name** edit box.



2. Enter *c02tut2* 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 2-29 consists of only lines. For ease of understanding, all lines in the sketch are labelled alphabetically.

1. Choose the **Line** button from the **Sketcher Tools** toolbar. Select a point close to the lower right corner of the drawing area 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 drawing area 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 downward and click to specify the endpoint of line F.
7. Now, to draw line G, move the cursor horizontally toward the right and click to specify the endpoint of line G.
8. Move the cursor vertically upward and click to specify the endpoint of line H.
9. Now, continue drawing the remaining lines that are shown in Figure 2-29. When the sketch is complete, end the 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 **Constrain** 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 **L₂** is applied to both the lines. The constraint labels such as **L₂** or **L₃** vary from sketch to sketch. 

3. Now, select lines J and N; the equal length constraint is applied to both the lines.
4. Select lines C and K; 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 2-30.
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 2-31.

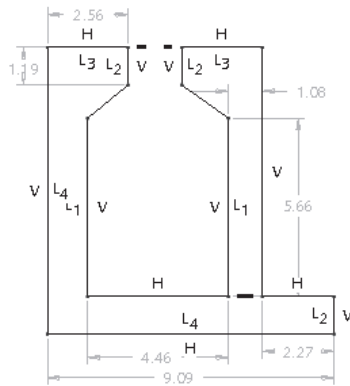


Figure 2-30 Sketch with weak dimensions and constraints

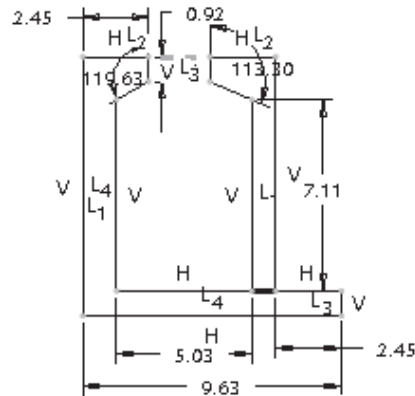


Figure 2-31 Sketch after dimensioning

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 **Normal** button from the **Sketcher Tools** toolbar. 
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 2-31 shows the sketch after applying dimensions. If your sketch does not have all dimensions shown in this figure, apply them using the **Normal** button.

Modifying the Dimensions

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

1. Choose the **One-by-One** button and then select all dimensions by specifying a window around them. 
2. When the dimensions turn red in color, choose the **Modify** 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 drawing area.
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 2-32.

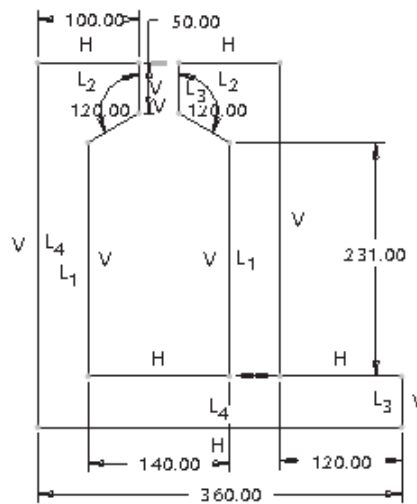


Figure 2-32 Complete sketch with dimensions and constraints

5. 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 2-33. The sketch is shown in Figure 2-34. For your reference, all entities in the sketch are labeled alphabetically. Also, print the sketch. **(Expected time: 30 min)**

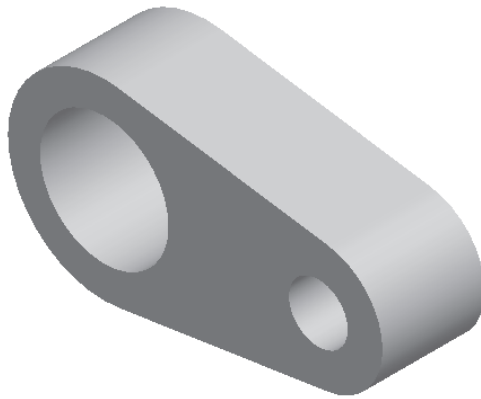


Figure 2-33 Model for Tutorial 3

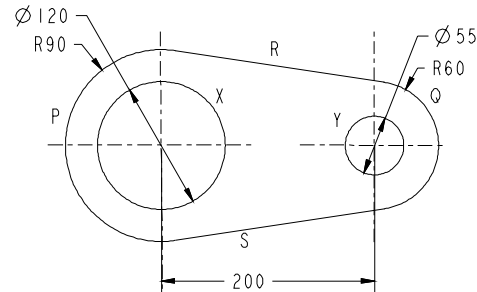


Figure 2-34 Sketch of the model

The following steps are required to complete this tutorial:

- Set the working directory and create a new object file.
- Draw the sketch using the sketcher tools, refer to Figures 2-35 through 2-38.
- Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 2-39.
- 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 set the working directory again by following the steps given next.

- Open the Navigator by sliding it out. In the Navigator, the **Folder Tree** is displayed at the bottom. Click on the black arrow, which is available on the right of the **Folder Tree**; the **Folder Tree** expands. Click on the plus sign adjacent to the *ProE-WF-4.0* folder in the Navigator; the contents of the *ProE-WF-4.0* folder are displayed.
- 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*. Close the Navigator.

Starting New Object File

- Choose the **New** 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. The default name of the sketch appears in the **Name** edit box.



2. Enter *c02tut3* in the **Name** edit box. Choose the **OK** button to enter the sketcher environment of the **Sketch** mode.

Drawing the Circles

1. Choose the **Center and Point** 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 2-35 shows the two collinear circles drawn using the **Center and Point** button from the **Sketcher Tools** toolbar.

Drawing the Tangent Lines

1. Choose the **Line Tangent** button from the flyout in the **Sketcher Tools** toolbar. You are prompted to select the start location on the arc or the 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 2-36 shows the sketch after drawing the tangent lines.

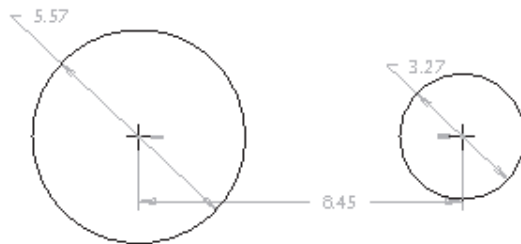


Figure 2-35 Two circles with weak dimensions and constraints

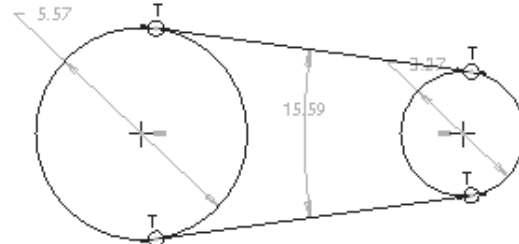


Figure 2-36 Circles joined by lines and the tangent constraint applied to them

Trimming the Circles

As evident from Figure 2-36, 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 **Delete Segment** button from the **Sketcher Tools** toolbar.



2. Select the two circles individually to trim them at the locations shown in Figure 2-37. Figure 2-38 shows the two circles after deleting the unwanted portion of the circle.

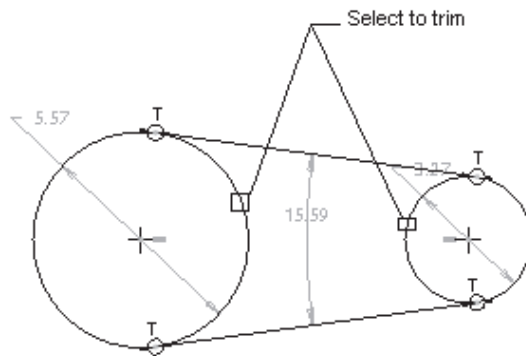



Figure 2-37 Locations to trim



Figure 2-38 Sketch after trimming


Drawing the Circles

1. Choose the black arrow on the right of the **Center and Point** button to display the flyout. From this flyout, choose the **Concentric** 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 **Normal** 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 **One-by-One** button. 
2. Select all dimensions by defining a window.



Note

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

3. When all dimensions turn red in color, choose the **Modify** 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 drawing area.
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 2-39.

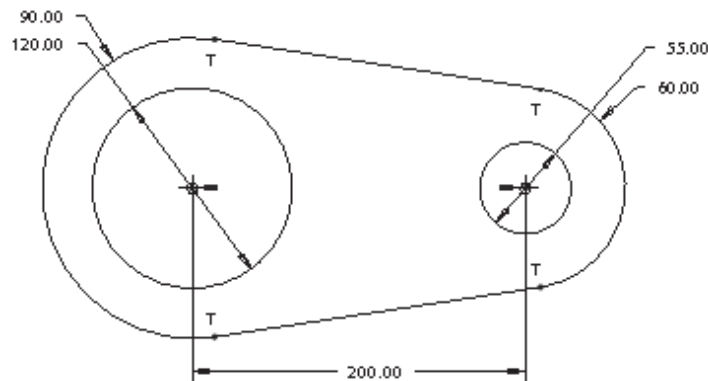


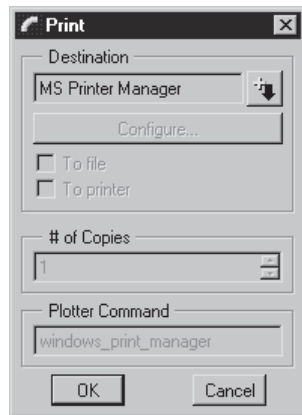
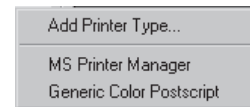


Figure 2-39 Complete sketch with dimensions and constraints

6. Save the sketch as discussed earlier. After saving the sketch, you need to print the sketch.

Printing the Sketch Using the Plot Option

1. Choose the **Print** button from the **File** toolbar or press CTRL+P; the **Print** dialog box is displayed, as shown in Figure 2-40. 
2. Choose the **Commands and Settings** button in this dialog box; a shortcut menu is displayed, as shown in Figure 2-41. 
3. Choose the **Add Printer Type** option from the shortcut menu; the **Add Printer Type** dialog box is displayed.
4. From the printers listed in the **Add Printer Type** dialog box, select the printer which is installed on your system and choose the **OK** button.
5. From the **Print** dialog box, choose the **Configure** button; the **Shaded Image Configuration** dialog box is displayed. This dialog box allows you to set the paper size.
6. Select the **A** option from the **Size** drop-down list; note that the dimensions of the sheet are set by default. Also, select the image resolution and the image depth from the dialog box.

Figure 2-40 The **Print** dialog boxFigure 2-41 The **Commands and Settings** shortcut menu

7. Choose the **OK** button from the **Shaded Image Configuration** dialog box.
8. Now, choose the **OK** button from the **Print** dialog box to complete printing.

Self-Evaluation Test

Answer the following questions and then compare them to 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. The **Intent Manager** is _____ by default when you enter the **Sketch** mode. (on/off)
9. In the **Sketch** mode, the tangent constraint is represented by a _____ 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 required 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 **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 2-42. The sketch is shown in Figure 2-43. **(Expected time: 30 min)**

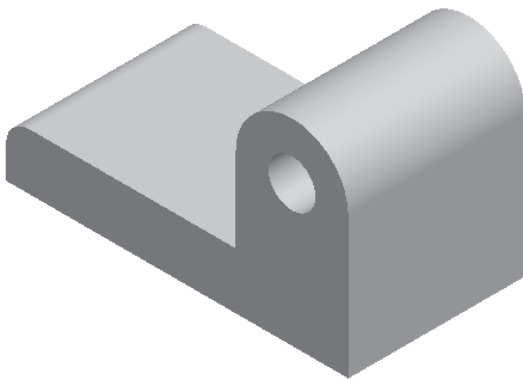


Figure 2-42 Solid model for Exercise 1

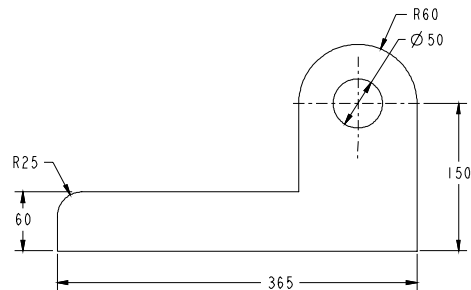


Figure 2-43 Sketch of the model

Exercise 2

In this exercise, you will draw the sketch for the model shown in Figure 2-44. The sketch is shown in Figure 2-45.
(Expected time: 30 min)



Figure 2-44 Solid model for Exercise 2

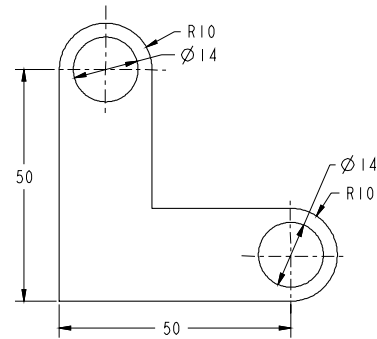


Figure 2-45 Sketch of the model

Exercise 3

In this exercise, you will draw the sketch for the model shown in Figure 2-46. The sketch is shown in Figure 2-47.
(Expected time: 30 min)

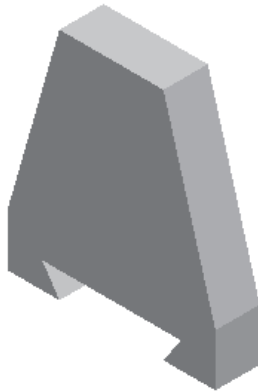


Figure 2-46 Solid model for Exercise 3

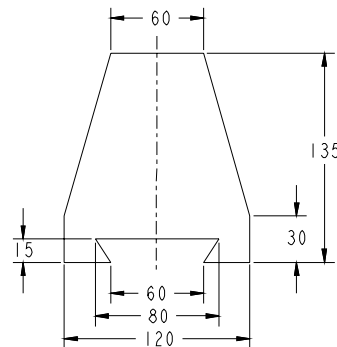


Figure 2-47 Sketch of the model

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