

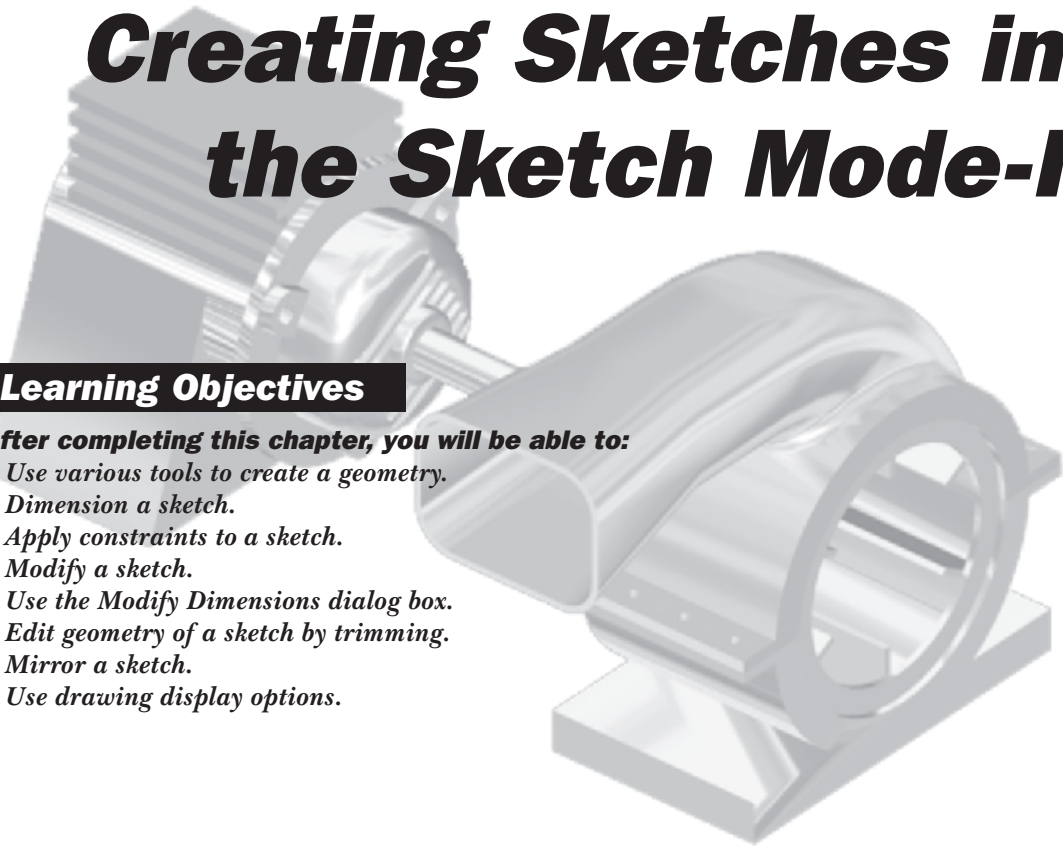
Chapter 1

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 geometry of a sketch by trimming.*
- *Mirror a sketch.*
- *Use drawing display options.*



THE SKETCH MODE

Almost all models designed in Pro/ENGINEER Wildfire 3.0 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. You can draw a 2D sketch of the product and assign the required dimensions and constraints to it. By assigning the dimensions, you can make sure that the 2D sketch of the product or model satisfies 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 3.0, 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, it 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, the sketcher environment is referred to as the environment in Pro/ENGINEER where you can draw 2D geometries. Apart from the **Sketch** mode, the sketcher environment can be accessed in the 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 will be displayed with other Pro/ENGINEER modes. Select the **Sketch** radio button to start a new file in the **Sketch** mode, see Figure 1-1; 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.

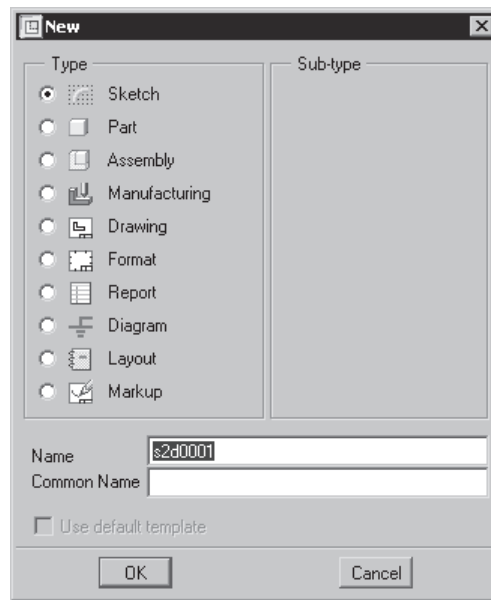


Figure 1-1 The New dialog box

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 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 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 sketches.



Note

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

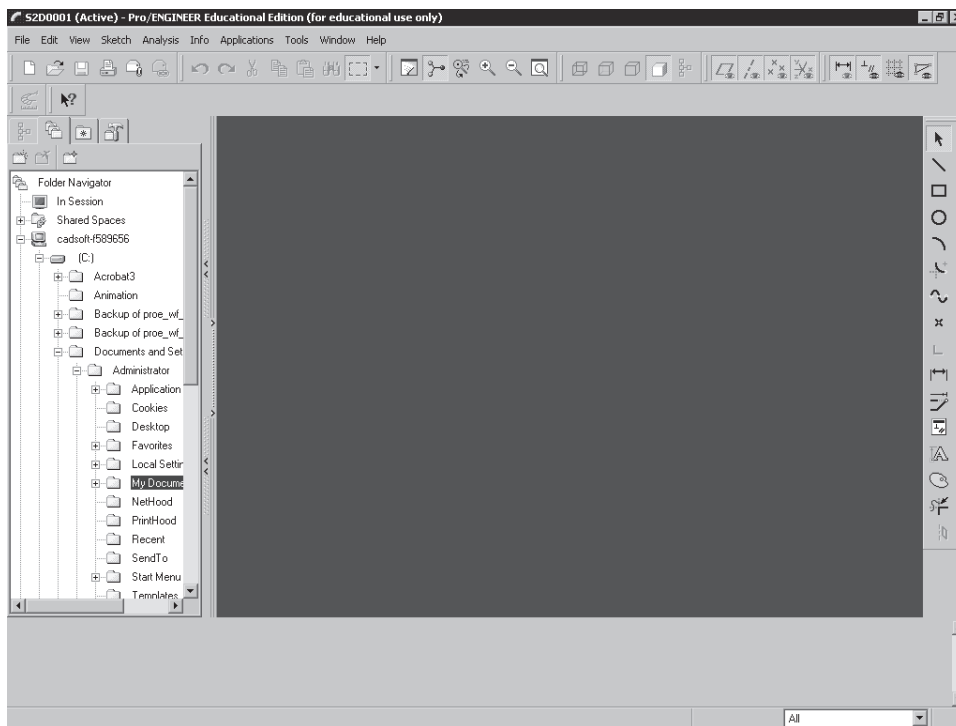


Figure 1-2 Initial screen appearance in the Sketch mode

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



Note

The functions of the navigator are discussed in 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.

You can create a simple sketch by using the options 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 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 on the **Right Toolchest**, contains the tools to draw, dimension, and modify a sketch. In this section, you will learn to draw sketched entities using the tools 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 the 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 drawing area.



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 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 with three tool buttons appears. The first button is **Create 2 point lines**. This button is used to create a line by selecting two points in the drawing area. 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 drawing area. 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 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 in detail later in this chapter.

2. The system prompts you to specify the endpoint. Move the cursor in the drawing area to a desired location and click to specify the endpoint of the line; a yellow line appears. 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 drawing area. After drawing a line, weak dimensions are applied to it and these are displayed in gray color. 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 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 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 will be 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 creating revolved features. It can also be used for mirroring, aligning, and dimensioning the sketcher 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 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 sketch a rectangle using the **Create rectangle** tool:

1. Choose the **Create rectangle** button from the **Sketcher Tools** toolbar; you will be prompted to select two points to indicate the diagonal of the 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.

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 with five buttons appears. The procedures to create a circle and an ellipse using 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 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 the circle creation, press the middle mouse button at any stage of its creation.

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 a 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.

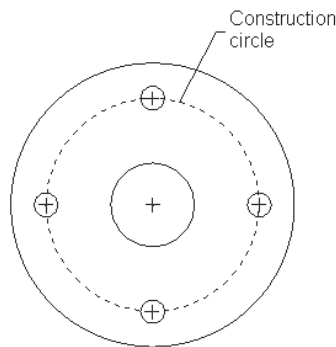


Figure 1-3 Sketch of a flange

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. A yellow circle appears 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:

1. Choose the **Create concentric circle** 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 clicking and then pressing the middle mouse button.

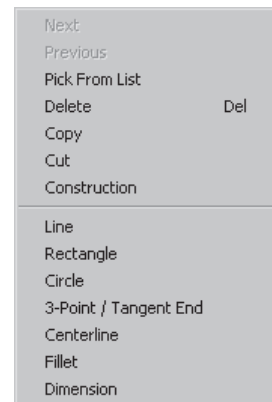


Figure 1-4 The Construction option in the shortcut menu



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.

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 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 again prompted 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 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 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 an 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.

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

3. To end the circle creation, 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 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 default dimensions for the major axis and minor axis will be displayed in gray color in the drawing area.

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 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 **Create an arc by 3 points or tangent to an entity at its endpoint** 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 its endpoint. 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 if you move the cursor in a direction perpendicular to the line, 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 of the **Target** symbol in the direction tangent to the line, the arc is drawn tangent to the line at the specified endpoint, as shown in Figure 1-6.

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.



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

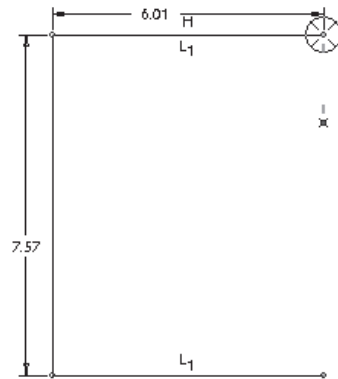


Figure 1-5 Cursor moved out of the **Target** symbol perpendicular to the endpoint

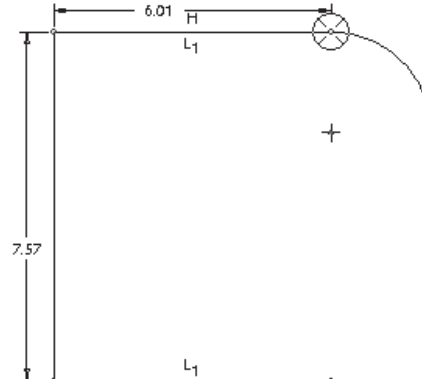


Figure 1-6 Cursor moved out of the **Target** symbol along the tangent direction.

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 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 dashed 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 the 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 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 dashed circle appears and is attached to the cursor.
3. Select the start point of the arc on the circumference of the dashed 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 its endpoint. 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 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 entities selected.
4. You can continue drawing arcs or press the middle mouse button to abort the 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 will be prompted to specify the first endpoint of the conic entity.
2. Specify a 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 **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.

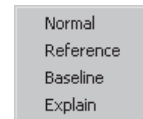


Figure 1-7 The dimensioning options

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



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.

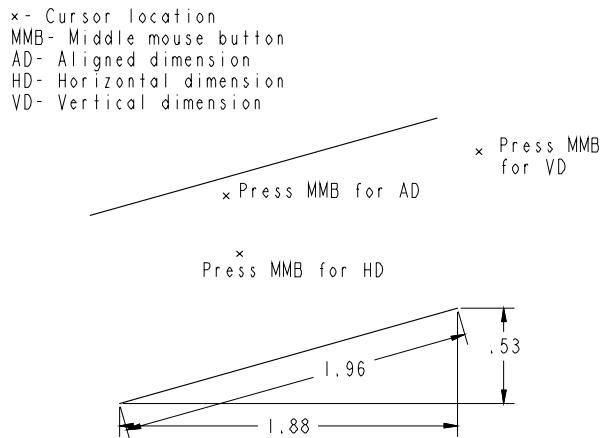


Figure 1-8 Approximate locations of the cursor to achieve different dimensions

Angular Dimensioning of an Arc

To add angular dimension to an arc, select its ends 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 the tools that are discussed later in this chapter.

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 1-10. The same diameter dimensioning technique can also be used for arcs.

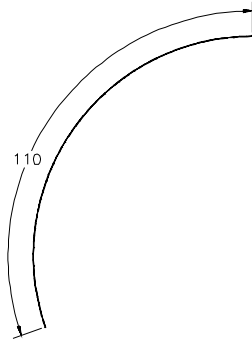


Figure 1-9 Angular dimensioning

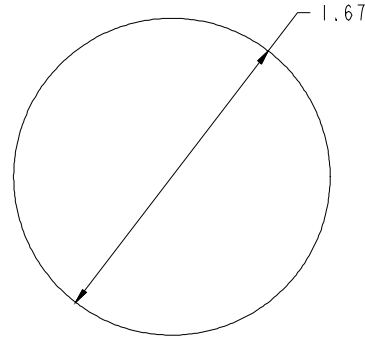


Figure 1-10 Diameter dimensioning

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 will be 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 will be displayed, as shown in Figure 1-12. This dimension represents the diameter of a revolved section.

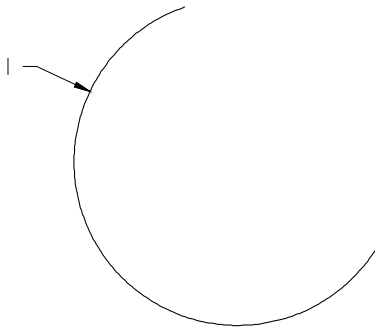


Figure 1-11 Radial dimensioning

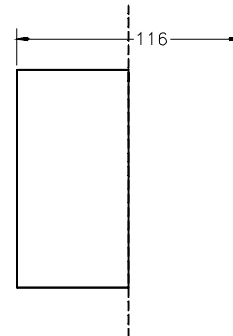


Figure 1-12 Dimensioning for revolved sections



Tip: To add a 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 it.

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 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 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, 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, a red 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 drawing area. 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.

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

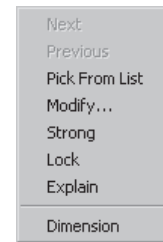


Figure 1-14 Shortcut menu to convert the weak dimensions to strong

MODIFYING 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 will be displayed, as shown in Figure 1-15.

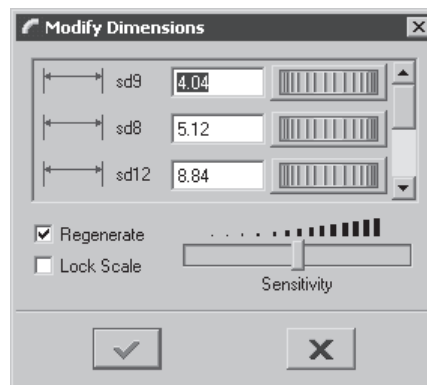


Figure 1-15 The *Modify Dimensions* dialog box

To modify dimensions using this dialog box, you can 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 will be selected and any modifications in the dimensions will be 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 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, the selection box will be displayed; you can now 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 will be selected. Therefore, the sketch will be regenerated dynamically as you modify the dimension.

Modifying Dimensions 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 1-16 and the constraints or dimensions that 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 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 will be displayed in the message area.

DELETING THE SKETCHED 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 it. 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.

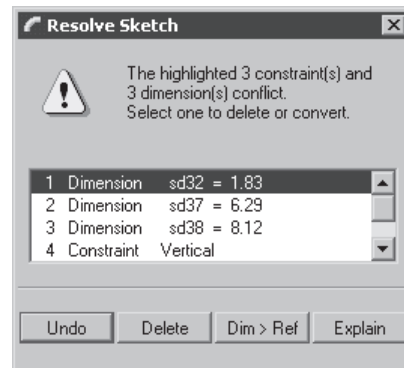


Figure 1-16 The **Resolve Sketch** dialog box

**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 SKETCHED 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 will be 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.

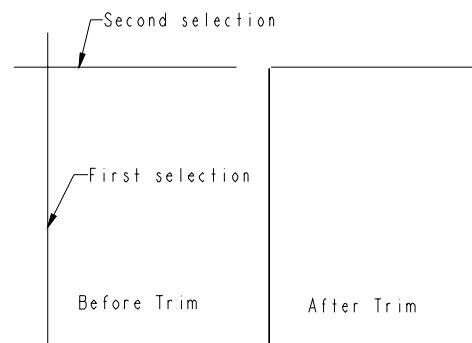


Figure 1-17 Trimming the lines

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 will be 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 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 smaller entities.

MIRRORING THE SKETCHED ENTITIES



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

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 will be prompted to select the centerline about which you need to mirror; select the centerline. The selected entities are mirrored about the centerline.



Tip: In case of symmetrical parts, you can save the time required 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.

INSERTING FOREIGN DATA INTO THE SKETCH



The **Insert foreign data from Palette into active object** tool helps you to 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 procedure to insert a foreign entity in the **Sketch** mode is discussed next.

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

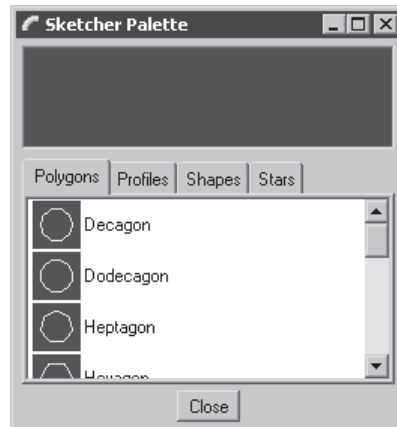


Figure 1-18 The **Sketcher Palette** dialog box



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 enabled in the **Sketcher Palette** dialog box. Also, this tab is selected by default. If the working directory contains other types of files such as .prt and so on, the same tab will be displayed at the last. In this case, the **Polygons** tab will be selected by default.

2. For inserting a sketch from the **Polygons** 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 1-19.

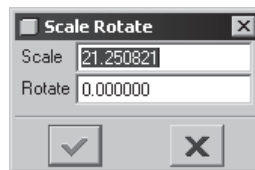


Figure 1-19 The **Scale Rotate** dialog box

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 steps 2-4 to continue inserting more sketches in the drawing area.

Similarly, you can add the sketches from the **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 need 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 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 the zoom tool, right-click in the drawing area.

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 drawing area 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 fit 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 drawing area 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 the 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 1-20. The sketch is shown in Figure 1-21. **(Expected time: 30 min)**



Figure 1-20 Model for Tutorial 1

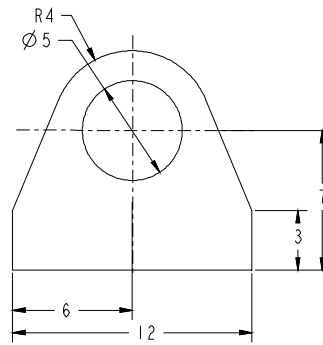


Figure 1-21 Sketch of the model

The following steps are required to complete this tutorial:

- Start Pro/ENGINEER Wildfire 3.0.
- Set the working directory and create a new sketch file.
- Draw lines using the line tool, refer to Figures 1-22 and 1-23.
- Draw an arc and a circle, refer to Figures 1-24 and 1-25.
- Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 1-26
- Save the sketch and close the file.


Starting Pro/ENGINEER Wildfire

- Start Pro/ENGINEER Wildfire 3.0 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-3.0*. If this folder does not exist, then create it prior to setting the working directory.

3. Choose the **New Directory** button in the **Select Working Directory** dialog box; the **New Directory** dialog box is displayed. 
4. 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-3.0*.
5. Choose **OK** from the **Select Working Directory** dialog box. You have set the working directory to *C:\ProE-WF-3.0\c01*. A message is displayed in the message area that the directory successfully changed to *C:\ProE-WF-3.0\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 drawing area.

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 drawing area 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 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 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 it.

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 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, **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. Notice that the **L₁** constraint is displayed in white color, as shown in Figure 1-20. This suggests that it is a strong 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.

8. Move the cursor to size the line and specify the endpoint of the left inclined line, see Figure 1-22.
9. Press the middle mouse button to end the 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 the system deletes weak dimensions, it does not confirm their deletion.
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 to end the line creation.

Figure 1-23 shows the lines that you have drawn. Now, the arc and the circle will be drawn.

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.

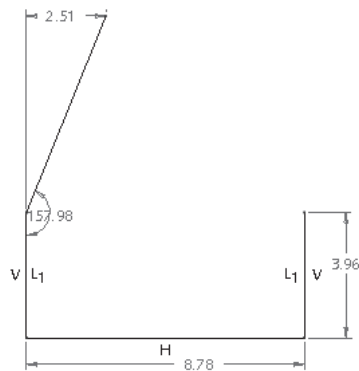


Figure 1-22 Lines with weak dimensions

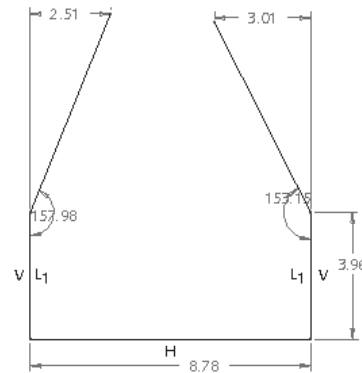


Figure 1-23 Partial sketch with weak dimensions

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 1-24. 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 1-23 there are some weak dimensions that are not displayed in Figure 1-24. 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 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 1-25.

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.

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 1-25.

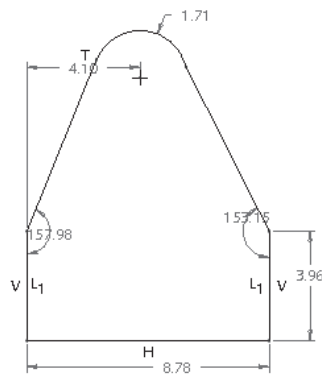


Figure 1-24 Sketch with arc

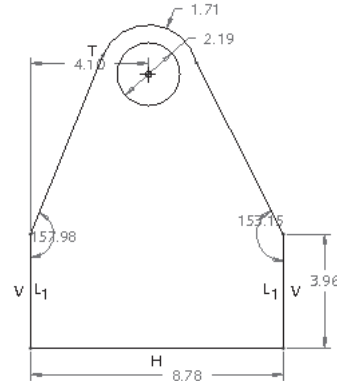



Figure 1-25 Sketch with all entities, weak dimensions, and weak constraints

Modifying the Dimensions

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

1. Choose the **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.

- 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.

- 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.

- 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 1-26.

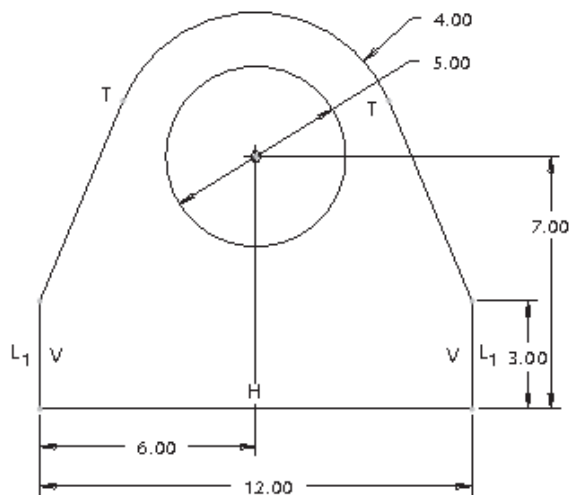


Figure 1-26 The complete sketch with dimensions and constraints



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 it 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 1-27. The sketch is shown in Figure 1-28. For your reference, all entities in the sketch are labeled alphabetically.

(Expected time: 30 min)

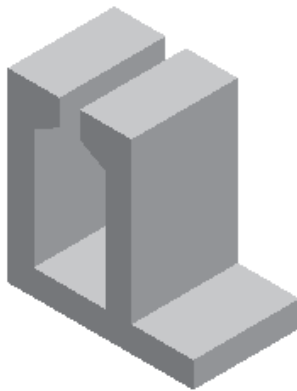


Figure 1-27 Model for Tutorial 2

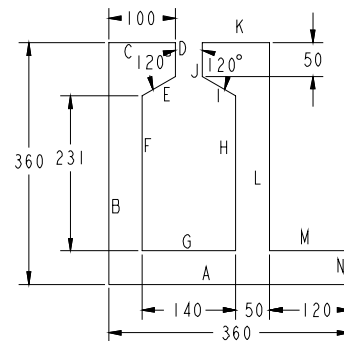


Figure 1-28 Sketch of the model

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 1-29.
- c. Dimension the required entities and then modify the dimensions of the sketch, refer to Figures 1-30 and 1-31.
- 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-3.0* folder in the navigator; the contents of the *ProE-WF-3.0* 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 the 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 1-28 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 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 it. 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 downwards 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 upwards and click to specify the endpoint of line H.
9. Now, continue drawing the remaining lines that are shown in Figure 1-28. 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 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. Now, select lines J and N; the equal length constraint is applied to both the lines.
4. Select lines A and B. The equal length constraint is applied to both the lines.
5. Choose the **Make line or two vertices horizontal** button from the **Constraints** dialog box; you are prompted to select a line or two vertices. 
6. Select the vertex that joins lines L and M and the vertex that joins lines G and H. Both the vertices are aligned horizontally, as shown in Figure 1-29.
7. Select the vertex that joins lines C and D and the vertex that joins lines J and K. Both the vertices are aligned horizontally, as shown in Figure 1-30.

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 1-30 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.

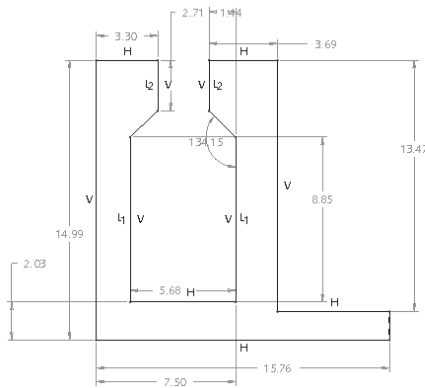


Figure 1-29 Sketch with weak dimensions and constraints

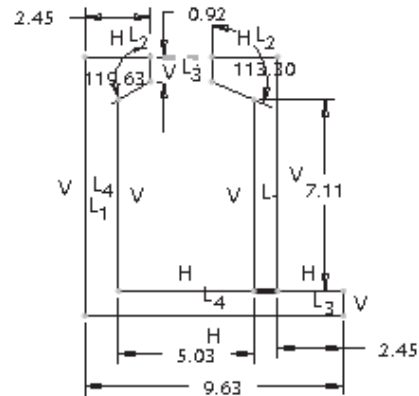




Figure 1-30 Sketch after dimensioning

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. 
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 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, see Figure 1-31.
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 there is a minor change in the dimension value or when only one dimension is required to be modified.

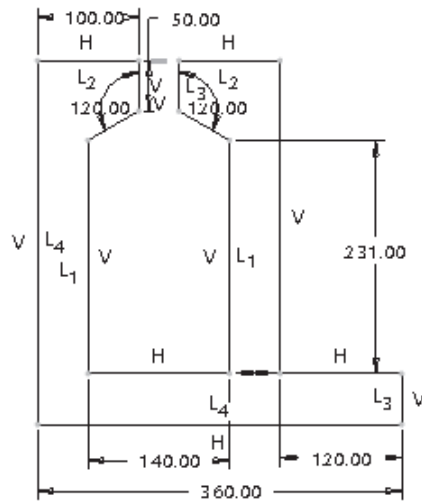


Figure 1-31 Complete sketch with dimensions and constraints

Tutorial 3

In this tutorial, you will draw the sketch for the model shown in Figure 1-32. The sketch is shown in Figure 1-33. For your reference, all entities in the sketch are labeled alphabetically. Also, print the sketch. **(Expected time: 30 min)**

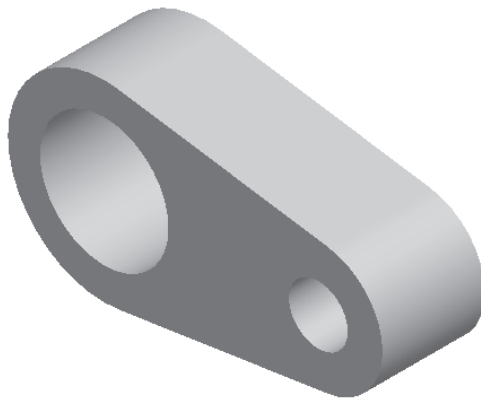


Figure 1-32 Model for Tutorial 3

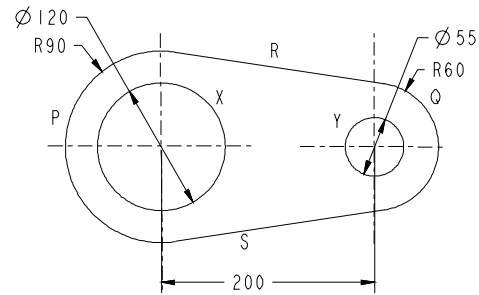


Figure 1-33 Sketch of the model

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 1-34 through 1-37.
- Dimension the sketch and then modify the dimensions of the sketch, refer to Figure 1-38
- 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-3.0* folder in the navigator; the contents of the *ProE-WF-3.0* 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 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 *c01tut3* 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 it.
3. Draw another circle whose center is collinear with the center of the previous circle.

Figure 1-34 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 1-35 shows the sketch after drawing the tangent lines.

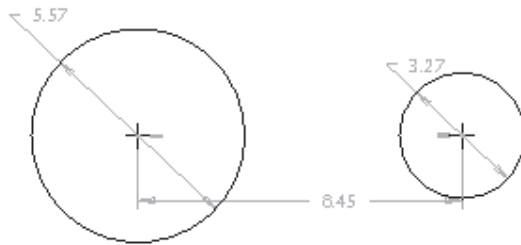


Figure 1-34 The two circles with weak dimensions and constraints

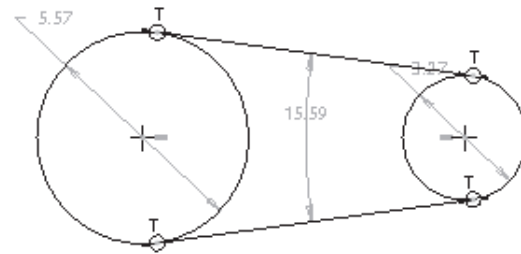


Figure 1-35 Circles joined by lines and the tangent constraint applied to them

Trimming the Circles

As evident from Figure 1-35, 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 1-36. Figure 1-37 shows the two circles after deleting the unwanted portion of the circle.

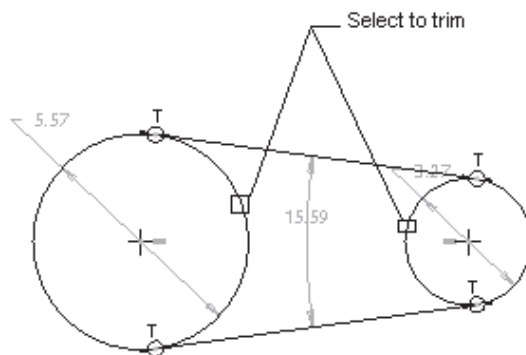


Figure 1-36 Locations to trim



Figure 1-37 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 on the keyboard to select all the 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 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 1-38.

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

Printing the Sketch Using the Plot Option

1. Choose the **Print the active object** button from the **File** toolbar; the **Print** dialog box is displayed, as shown in Figure 1-39 and the **MS Printer Manager** option is selected by default.



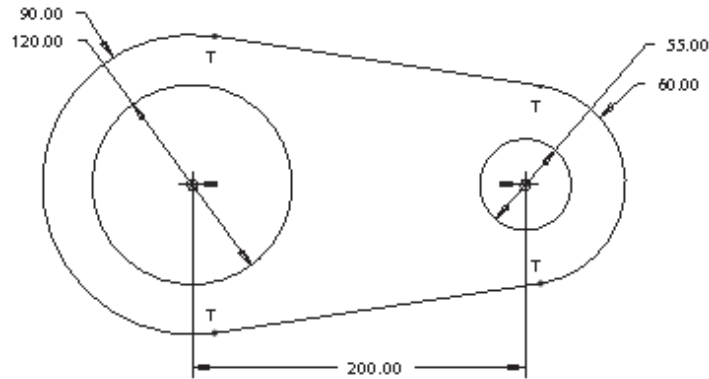


Figure 1-38 The complete sketch with dimensions and constraints

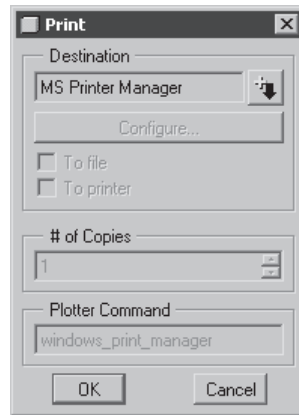


Figure 1-39 The **Print** dialog box



Note

The **MS Printer Manager** option is used to print the active object on the printer installed as the default printer on your computer. It is recommended that you use a laser printer to print the sketch (.sec) files

2. Now, choose the **OK** button from the **Print** dialog box; a new **Print** dialog box is displayed with the default printer installed on your computer selected by default. Select a laser printer from the **Name** drop-down list in the new **Print** dialog box.
3. Choose the **OK** button from the new **Print** dialog box to print the part model you have created.

Closing the Current Window

The given model is completed and is also saved. Now, you can close the current window.

1. Choose **File > Close Window** from the menu bar.
-

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 files in the **Sketch** mode are saved with a _____ file extension.

Review Questions

Answer the following questions:

1. What is the need of the **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 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 1-40. The sketch is shown in Figure 1-41. **(Expected time: 30 min)**

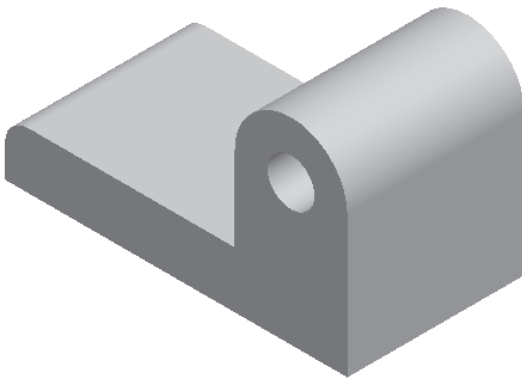


Figure 1-40 Solid model for Exercise 1

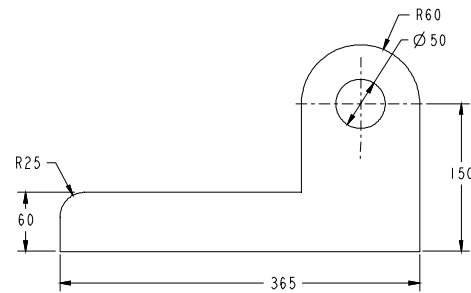


Figure 1-41 Sketch of the model

Exercise 2

In this exercise, you will draw the sketch for the model shown in Figure 1-42. The sketch is shown in Figure 1-43. **(Expected time: 30 min)**



Figure 1-42 Solid model for Exercise 2

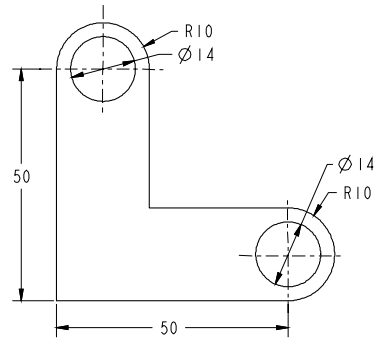


Figure 1-43 Sketch of the model

Exercise 3

In this exercise, you will draw the sketch for the model shown in Figure 1-44. The sketch is shown in Figure 1-45. **(Expected time: 30 min)**

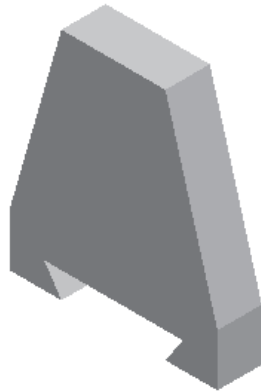


Figure 1-44 Solid model for Exercise 3

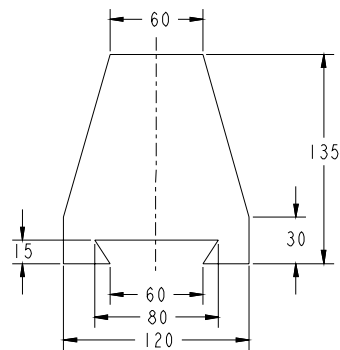


Figure 1-45 Sketch of the model

Exercise 4

In this exercise, you will draw the sketch for the model shown in Figure 1-46. The sketch is shown in Figure 1-47. **(Expected time: 30 min)**

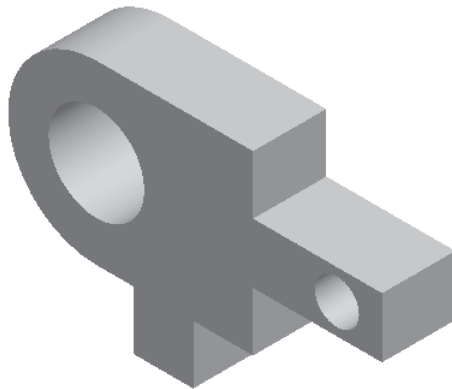


Figure 1-46 Solid model for Exercise 4

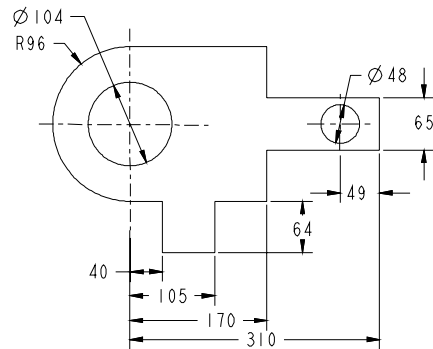


Figure 1-47 Sketch of the model

Exercise 5

In this exercise, you will draw the sketch for the model shown in Figure 1-48. The sketch is shown in Figure 1-49. **(Expected time: 30 min)**

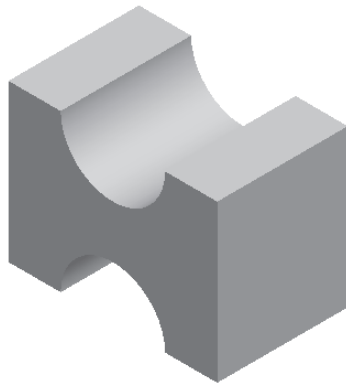


Figure 1-48 Solid Model for Exercise 5

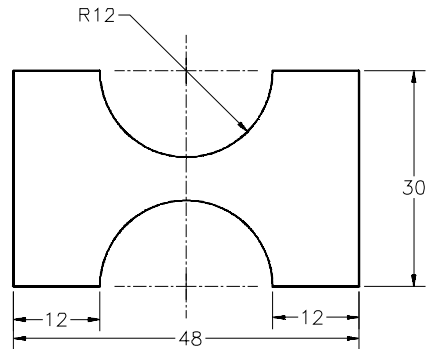


Figure 1-49 Sketch of the model

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