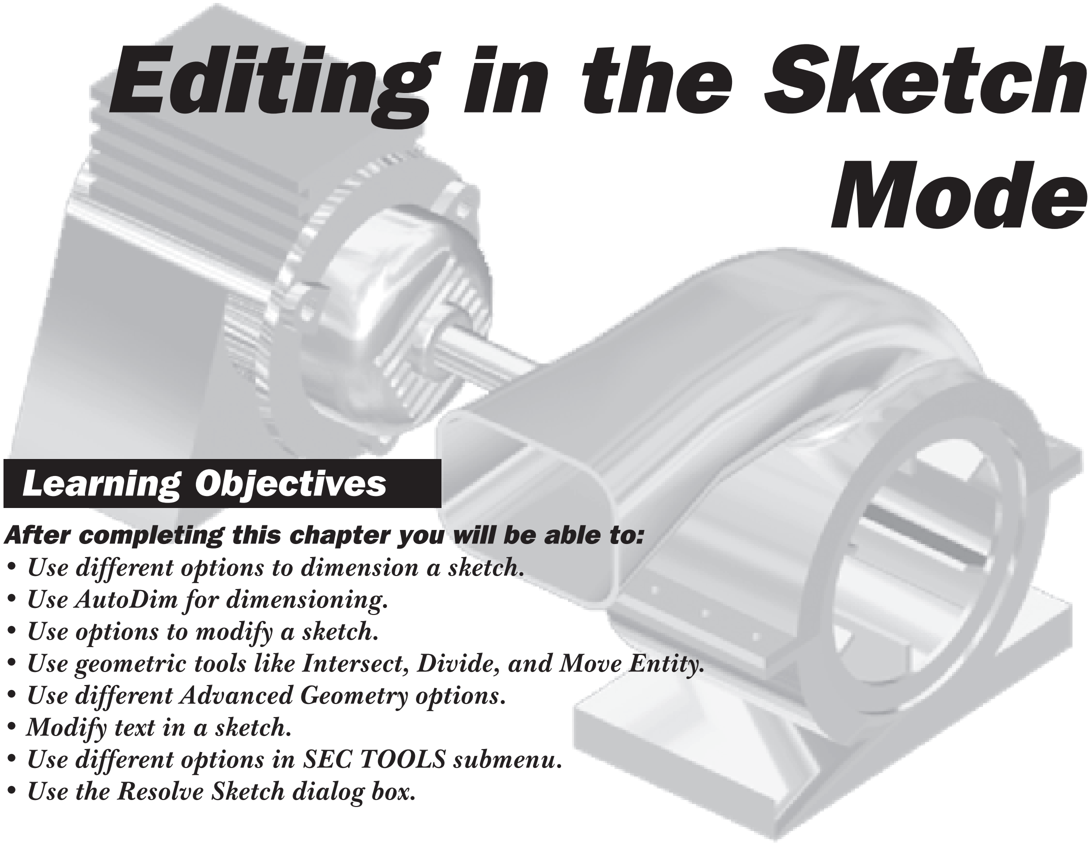
Chapter 2

Editing in the Sketch Mode



Learning Objectives

After completing this chapter you will be able to:

- *Use different options to dimension a sketch.*
- *Use AutoDim for dimensioning.*
- *Use options to modify a sketch.*
- *Use geometric tools like Intersect, Divide, and Move Entity.*
- *Use different Advanced Geometry options.*
- *Modify text in a sketch.*
- *Use different options in SEC TOOLS submenu.*
- *Use the Resolve Sketch dialog box.*

DIMENSIONING THE SKETCH

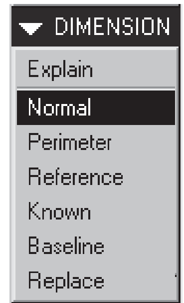


*Figure 2-1 The different options of the **DIMENSION** submenu*

In Chapter 1 you learned how to dimension a sketch using the **Normal** option of the **DIMENSION** menu in the **Menu Manager**. As mentioned in Chapter 1, the **DIMENSION** submenu is available only when the **Intent Manager** is off. In this chapter you will learn to dimension a sketch using the remaining options available under the **DIMENSION** submenu shown in Figure 2-1.

When the **Intent Manager** is on, choose **Sketch** > **Dimension** from the menu bar. The cascading menu is displayed as shown in Figure 2-2. The options that are available to dimension a sketch when the **Intent Manager** is on are shown in the figure.

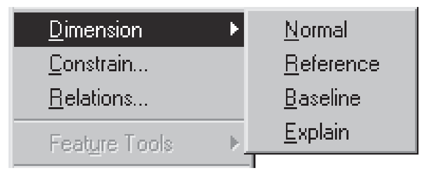


*Figure 2-2 Dimensioning options available when the **Intent Manager** is on*

Dimensioning a Sketch Using the Perimeter Option

The **Perimeter** option specifies the perimeter of the entities or a loop of entities selected. In this type of dimensioning, a dimension is selected from a dimensioned sketch that will vary with the change in the perimeter. If you modify the perimeter value of the sketch the modification is reflected in the dimension of the entity selected to vary and hence the sketch is also modified corresponding to the variable dimension. The following steps explain the procedure to dimension a sketch using the **Perimeter** option:

1. Draw a sketch as shown in Figure 2-3 and dimension it using the **Normal** option. After adding the dimensions, regenerate the sketch.

2. Now, choose **SKETCHER** > **Dimension** > **Perimeter** from the **Menu Manager**. You will be prompted to select the entity that is one end of the desired chain or a part of the desired loop. This will be the first entity of the chain. Using the left mouse button select entity (a) as shown in Figure 2-3. The color of the selected entity changes from cyan to red.

3. You will be prompted to select the last entity of the chain or to choose **Done Sel** to select the complete sketch. Select the last entity (c) for the chain using the left mouse button. The color of all the entities forming a chain (lines (a), (b), and (c) in this case) changes to red and the **CHOOSE** submenu is displayed.

   The **CHOOSE** submenu is displayed because when you select the entities defining the first and the last end of the chain, two chains of entities are created. The first one consists of the entities in the clockwise direction and the second one consists of the entities in the counterclockwise direction. By default, the entities that form a chain in the clockwise direction is highlighted (lines (a), (b), and (c)).

4. Choose the **Accept** option to accept the highlighted chain of entities. You will be prompted to select a dimension that will be driven by the perimeter dimension. Using the left mouse

button select the dimension of line (a) as the variable dimension, see Figure 2-3. Regenerate the sketch.

5. Now, if you modify the perimeter value, the dimension selected to vary will be proportionately modified after regeneration, thus altering the geometry of the sketch. Remember that the dimension selected to vary cannot be modified using the **Modify** option in the **SKETCHER** menu. Figure 2-4 shows the perimeter modified to 100 and the consequent change in the variable dimension.

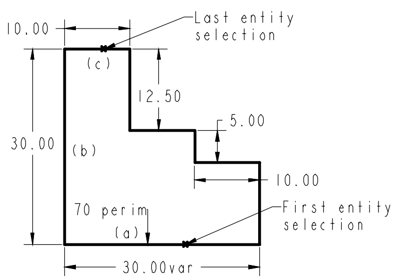


*Figure 2-3 The **Perimeter** option*

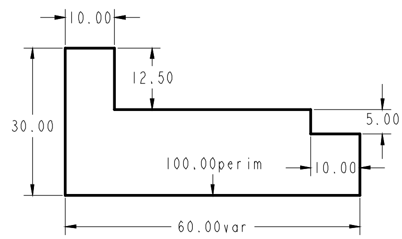


*Figure 2-4 The **Perimeter** option*

Note
If you delete the variable dimension then the perimeter dimension is also deleted.

Dimensioning a Sketch Using the Reference Option

The **Reference** option is used to create reference dimensions. These dimensions are used only for information and are not used in manufacturing the part. These dimensions appear with the suffix "REF". The following steps explain the procedure to create reference dimensions.

1. Choose the **Reference** option from the **DIMENSION** submenu.

2. Select an entity in the sketch to dimension by using the left mouse button. The cyan color of the entity changes to red. Place the dimension using the middle mouse button. The dimension appears in symbolic form as **rsd** with the suffix "REF". If the **Intent Manager** is on, the dimension value appears.

If you select a reference dimension to modify, a message is displayed "**Modifying extra or reference dimension can only affect sketch via relations. Continue?**". Choose the **Yes** button, the value of the reference dimension will be displayed in the **Message Input Window**. Enter a new value of the reference dimension. The reference dimension will be displayed in white color after specifying a new value. However, as you regenerate the sketch, the value of the reference dimension will be restored to its original value. This is because the value of a reference dimension is driven by the normal dimensions and therefore, you cannot modify the value of a reference dimension. Therefore, if you make any change in the normal dimensions of the sketch and regenerate it, the change is automatically reflected in the value of the reference dimension.

Tip: *In a sketch, you can toggle between the symbolic dimensions and dimension values by choosing* ***Info > Switch Dimensions*** *from the menu bar.*

Dimensioning a Sketch Using the Known Option

The **Known** option is used to create dimensions of a sketch by adding relations using the known dimensions of a part. This option is not available when you are sketching in the Sketch mode and is available only in the sketcher environment of the Part mode. These known dimensions are considered as the driving dimensions of the sketch.

Remember that when using relations in the Sketch mode, the dimension is denoted by the symbol **sd**, and in Part mode by **d**; the reference dimension in the Sketch mode is denoted by **rsd** and in Part mode by **rd**. You will come across these dimension symbols as you learn various modes available in Pro/ENGINEER. The relations are discussed in Chapter 8. The following steps explain the procedure to create known dimensions:

1. Retrieve a model to create a feature on it.

   Although the sketch of the retrieved model does not have the dimensions displayed on it, yet you can select any entity. The dimension of the selected entity is the known dimension and that act as the driving dimension for the future dimensions to be drawn on the retrieved section of the model. The dimensions of the sketched feature act as the driven dimensions.

2. Sketch the other feature you want to create. Dimension it using the **Normal** option. Now, you can use the known dimensions of the retrieved model to determine the dimensions of the sketched feature.

3. Choose the **Known** option from the **DIMENSION** submenu. Now, select an entity with respect to which you have to dimension the sketched entity. The dimension of the selected entity is displayed as **kd#**, where # is the number associated with the dimension and varies from user to user. Then choose **Relation** from the **SKETCHER** menu. A **RELATIONS** submenu appears on the screen. Choose **Add** from this submenu; the **Message Input Window** appears to enter the relation.

4. Now, enter the relation of the sketch with respect to the known dimensions of the part and press ENTER. Remember to use the appropriate dimension symbols while giving the relations. After you are finished with entering all the relations, press ENTER and then regenerate the sketch. The dimensions of the sketch will be changed according to the relations added.

Dimensioning a Sketch Using the Baseline Option

In Pro/ENGINEER, the **Baseline** option of dimensioning is used to create dimensions in terms of horizontal and vertical location values of an entity with respect to a specified baseline. This type of dimensioning in a drawing is required for writing a CNC program to manufacture a component. This option can be used to dimension lines, conics, arcs, and so on. The following steps explain the procedure to create dimension using the **Baseline** option:

1. Choose **SKETCHER** > **Dimension** > **Baseline** from the **Menu Manager**.

2. Select the entity that will act as the baseline (origin or reference). Using the middle mouse button, place the dimension.

   Depending upon the entity selected to act as the baseline, the horizontal or the vertical dimension value of the location of the entity will be placed. For example, if you select a vertical line, the vertical value of its location will be placed. Similarly, if you select a horizontal line, the horizontal value of its location will be placed. However, for arcs, circles, and splines there are two options to dimension using the **Baseline** option. A **VERT HORIZ** submenu appears when you select a circle center or an arc for baseline dimensioning and you are prompted to select the dimension orientation. The dimension is placed according to the orientation selected. Note that since the location value of the baseline is taken as the origin, the dimension value of the baseline entity will become 0.00 when regenerated. The dimensions values of the other entities dimensioned with reference to the baseline will be measured from this origin.

3. Next, choose the **Normal** option from the **DIMENSION** menu in the **Menu Manager**. Select the baseline dimension that was placed earlier and then select the entity to dimension. Now, using the middle mouse button, place the dimension.

   Depending upon the baseline dimension and the entity selected, the dimension will be placed. Figure 2-5 shows a regenerated sketch dimensioned using the above-mentioned method. In this figure, the two baselines are dimensioned using the **Baseline** option. Therefore, after regeneration, the dimensions of these lines are displayed as 0.00. The remaining lines are dimensioned using the **Normal** option by first selecting the baseline dimension and then the entity to dimension.

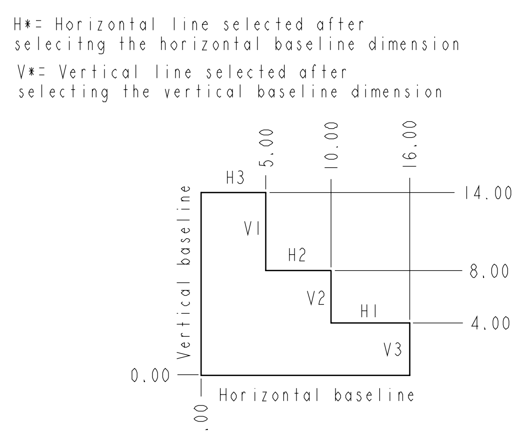


Figure 2-5 *Baseline dimensioning of a sketch*

Replacing the Dimensions of a Sketch Using the Replace Option

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

1. Choose the **Replace** option from the **DIMENSION** submenu. When the **Intent Manager** is on, choose **Edit > Replace** from the menu bar. You will be prompted to select a dimension to be replaced.

2. Select the dimension to be replaced using the left mouse button. The selected dimension is erased. Select the entity using the left mouse button to dimension and place the dimension at the desired place. The previous dimension will be replaced by a new dimension.

AUTODIMENSIONING IN PRO/ENGINEER

Dimensioning a sketch is necessary in order to regenerate the sketch. If the **Intent Manager** is on, the sketch is automatically dimensioned. But, if you are sketching without the **Intent Manager**, you have an option that allows you to create the dimensions automatically. This option is **AutoDim**. This option is available under the **SKETCHER** menu. The **AutoDim** option is not available when the **Intent Manager** is on. The **AutoDim** option uses two reference planes for dimensioning when you are in the Part mode. But, when you are sketching in the Sketch mode and the **AutoDim** option is chosen, the sketch is automatically dimensioned without any references. Also, note that the sketch is automatically regenerated when you select this option.

MODIFYING A SKETCH

When the **Intent Manager** is off, a sketch is modified by using the **MOD SKETCH** submenu. The **MOD SKETCH** submenu, shown in Figure 2-6, is displayed when you choose the **Modify** option from the **SKETCHER** menu in the **Menu Manager**.

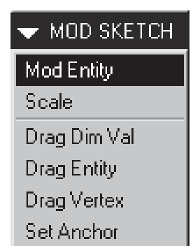


Figure 2-6 *Options in the* ***MOD SKETCH*** *submenu*

The **Mod Entity** option was discussed in Chapter 1 and the rest of the options are discussed next.

Scale

In case the section is complex and involves many dimensions, then the dimensions may conflict when you modify them. Hence, the section will not be regenerated successfully. Using the **Scale** option of the **MOD SKETCH** submenu, the overall scale of the sketch can be changed. This option is available only when the section has been regenerated once. The following steps explain the procedure to use the **Scale** option to modify the dimension values:

1. Choose **SKETCHER > Modify > Scale** from the **Menu Manager**.

2. You will be prompted to select a linear dimension. Select the dimension you want to set as the driving dimension. The color of the selected dimension changes to red and the **Message**

Input Window appears. Enter the dimension in this window and press ENTER.

The dimension selected is called the driving dimension because when the driving dimension is modified all the dimensions in the sketch are scaled proportionately.

3. Now, choose the **Regenerate** option from the **SKETCHER** menu. All the other dimensions in the sketch are scaled to the same scale factor thus modifying the section. Remember that the selected dimension will scale the other dimensions in the sketch proportionately only once. If you again want to scale the dimension of the sketch in relation to a dimension, you need to repeat step 2 and 3.

When the **Intent Manager** is on, you can scale a sketch by using the **Scale Rotate** dialog box. Choose the black arrow on the right of the **Mirror selected entities.** button from the **Right Toolchest** to display the flyout. From this flyout, choose the **Scale and rotate selected entities.** button. The **Scale Rotate** dialog box is displayed. This button is available only when an entity or a complete sketch is selected. Select an entity or a complete sketch and then choose this button, the **Scale Rotate** dialog box is displayed as shown in Figure 2-7. You can dynamically scale or rotate the sketch or the selected entity on the graphics screen. You can also use the **Scale Rotate** dialog box to enter a value for rotating and scaling in the **Scale** and **Rotate** edit boxes.

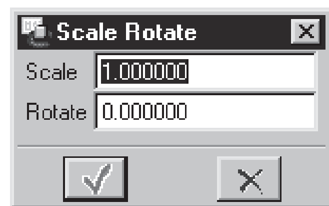


*Figure 2-7 **Scale Rotate** dialog box*

Drag Dim Val

Using the **Drag Dim Val** option you can make minute changes in the value of the selected dimension by varying the dimension and dynamically view the modification in the section geometry. You can select up to five dimensions at a time to make alterations. This option is available only when the sketch is regenerated once. The following steps explain the procedure to use the **Drag Dim** Val option to modify the dimension values:

1. Choose **SKETCHER** > **Modify** > **Drag Dim Val** from the **Menu Manager**.

2. You will be prompted to select the dimensions to modify. Select the dimensions you want to modify. You can select a maximum of five dimensions at a time. Choose **Done Sel** from the **GET SELECT** submenu or press the middle mouse button. A thermotool appears with all the dimensions selected to modify.

3. You can set sensitivity for modification in the dimension. At any time you can right-click on the thermotool to automatically reset to the original values of the selected dimension. Move the red pointer on the scale below each dimension in the thermotool to modify the selected dimension. The changes in the dimension values will be automatically reflected in the sketch.

4. To exit the thermotool, press the middle mouse button.

Figure 2-8 shows the section sketch and Figure 2-9 shows the thermotool for modifying all the dimensions of the sketch.

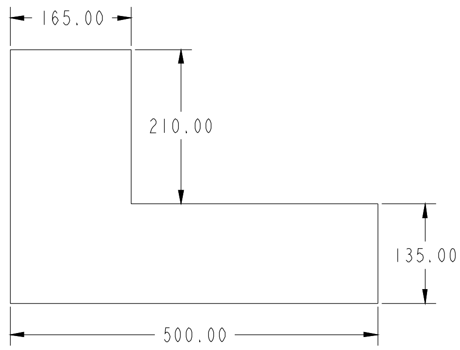


***Figure 2-8** The section sketch*

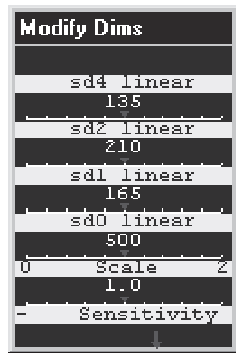


***Figure 2-9 Modify Dims** thermotool*

Drag Entity

The **Drag Entity** option is used to modify a selected dimension to change the shape of the section sketch by selecting the dimension and moving the corresponding entity. Remember that if the selected entity is constrained then you cannot modify it. You can only modify it by first disabling the constraint. This option is available only when the section is regenerated. The following steps explain the procedure to use the **Drag Entity** option to modify the dimensions:

1. Choose **SKETCHER** > **Modify** > **Drag Entity** from the **Menu Manager**.

2. You will be prompted to select the dimension to modify. Select the dimension you want to modify using the left mouse button.

3. You will now be prompted to select an entity for dragging or to select a new dimension. Select an entity corresponding to the selected dimension and move it to modify the dimension value accordingly. Note that you can move only one entity at a time to change the dimension of the entity.

Drag Vertex

The **Drag Vertex** option is used to modify the section sketch by dragging the vertices. The following steps explain the procedure to modify a sketch by using the **Drag Vertex** option.

1. Choose **SKETCHER** > **Modify** > **Drag Vertex** from the **Menu Manager**. You will be prompted to select two dimensions to modify.

2. Select the two dimensions to modify and select a corresponding vertex to drag. The dimensions are modified as you move the cursor and accordingly the sketch is also modified.

3. When you get the desired geometry of the sketch, accept the new values of the selected dimensions using the left mouse button. You can abort the new value of the dimension and restore the original one by using the middle mouse button.

Set Anchor

The **Set Anchor** is available only when you have dimensioned the sketch and regenerated it. When you select this option the anchor point is stored with the sketch. The following steps explain the procedure to modify the sketch using the **Set Anchor** option:

1. Choose **Set Anchor** from the **MOD SKETCH** submenu in the **Menu Manager**. You will be prompted to select a vertex that should remain fixed while using the other options of the **MOD SKETCH** submenu.

2. Select a vertex in the sketch that you want to be fixed using the left mouse button. Now, you can modify the other sketch dimensions by using the other options of the **MOD SKETCH** submenu. When you regenerate the sketch you will see that the sketch has been modified without altering the position of the vertex selected as the anchor point.

GEOM TOOLS

The tools that help a designer to complete the section sketches are available in the **GEOM TOOLS** submenu in the **Menu Manager**. This submenu is available only when the **Intent Manager** is off. Choose **Geom Tools** from the **SKETCHER** menu. The **GEOM TOOLS** submenu appears with different options as shown in Figure 2-10. Some of the options are not displayed in this submenu because they are not needed in the Sketch mode. The **Trim** and the **Mirror** options were discussed in Chapter 1. The **Intersect**, **Divide**, and the **Move Entity** options are discussed next.

***Figure 2-10** Figure showing different **GEOM TOOLS** option*

Intersect

The **Intersect** option is used to intersect two entities and convert them into separate entities at the point of intersection. You can delete the portion of the entity that is not required. The following steps explain the procedure to use the **Intersect** option. Refer Figure 2-11 and Figure 2-12:

1. Choose **Intersect** from the **GEOM TOOLS** submenu. You will be prompted to select two entities.

2. Select any two entities individually using the left mouse button. The two selected entities are broken at the intersecting point and are converted into four entities.

3. Continue step 2 until you have intersected all the desired entities. Figure 2-11 shows a sketch with three lines extending beyond the intersection point and Figure 2-12 show the lines after intersecting and deleting the extended lines.

Note

You can use the center lines and planes to intersect but they themselves are not converted into two entities.

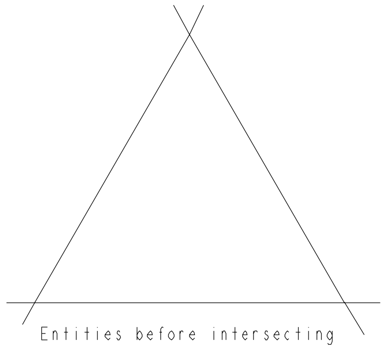


Figure 2-11 Three entities that split into nine entities after intersecting

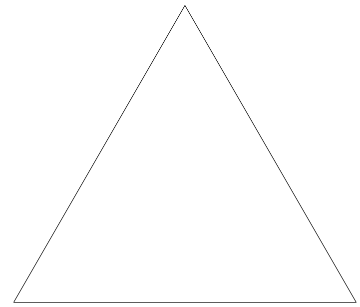


Figure 2-12 The required sketch

Divide

The **Divide** option is used to divide an entity into any number of parts or entities by specifying the points on the entity. Note that if you select a dimensioned entity to divide, you will be prompted to first delete the dimension and then divide it.

When the **Intent Manager** is on, choose the **Divide an entity at the point of selection.** button from the **Right Toolchest**. This button is available on the flyout that is displayed when you choose the black arrow that is on the right side of the **Dynamically trim section entities** button.

The following steps explain the procedure to divide an entity:

1. Choose **Divide** from the **GEOM TOOLS** submenu in the **Menu Manager**. You will be prompted to select an entity to divide.

2. Using the left mouse button 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 different entities.

3. Repeat step 2 until you divide the entities in the required number of points.

Move Entity

The **Move Entity** option is used to move the sketched entities and the dimensions. Choose **Move Entity** from the **GEOM TOOLS** submenu in the **Menu Manager**. The **MOVE ENTITY** submenu appears with the options to move the sketcher entities and dimensions. Figure 2-13 shows the options available in the **MOVE ENTITY** submenu. The options in this submenu are discussed next.

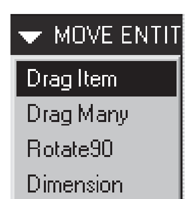


***Figure 2-13** Options in the **MOVE ENTITY** submenu*

Drag Item

The **Drag Item** option is used to drag individual entities and dimensions of a sketch. The following steps explain the procedure to use the **Drag Item** option:

1. Choose **Drag Item** from the **MOVE ENTITY** submenu in the **Menu Manager**. You are prompted to select an entity or one of its vertices to drag.

2. Select the entity you want to drag using the left mouse button. The color of the entity changes to red and the cursor is attached to the entity. You are prompted to select a new location to place the entity or the vertex selected. Note that if you select a vertex of an entity, it will not be moved, it will be stretched. Similarly, if you select an arc and move it to a new location, the center of the arc will not change but its radius will be changed.

   You can abort this option by using the middle mouse button at any time.

3. Move the cursor and place the selected entity using the left mouse button.

4. Repeat steps 2 and 3 until you drag all the required entities or dimensions. Use the middle mouse button to abort the **Drag Item** option.

Drag Many

The **Drag Many** option is used to move many entities at a time to a new location. You cannot drag the dimensions using this option. The following steps explain the procedure to drag many entities at a time.

1. Choose the **Drag Many** option from the **MOVE ENTITY** submenu in the **Menu Manager**. You will be prompted to select the entities to translate.

2. Select the entities you want to move. The color of the entities becomes red. Choose **Done Sel** from the **GET SELECT** submenu to complete your selection.

3. You will be prompted to select a start point for dragging. Select a point on the graphics screen. The selected entities will move with the cursor. Move the cursor at the desired place on the graphics screen and using the left mouse button, select the new location of the entities.

4. Repeat step 2 and 3 until you move all the required entities. Use the middle mouse button to end this option.

Rotate 90

The **Rotate 90** option is used to rotate a single or multiple entities in 90-degree increments. The following steps explain the procedure to rotate entities.

1. Choose **Rotate 90** from the **MOVE ENTITY** submenu in the **Menu Manager**. You will be prompted to select the entities to rotate:

2. Select the entities you want to rotate. The color of the entities to be rotated turns red. Choose **Done Sel** from the **GET SELECT** submenu to complete the selection. The **Message Input Window** is displayed and you will be prompted to enter counterclockwise rotation

angle in 90-degree increments in the **Message Input Window**.

3. Enter the required value in the multiples of 90-degree that will rotate the selected entity, and then press **ENTER**.

4. You will now be prompted to select the center point of rotation about which the selected entities will rotate. Specify the center point of rotation at the desired location on the graphics screen. The selected entities are rotated about the specified center point.

5. Repeat step 2, 3, and 4 until you rotate all the required entities. Use the middle mouse button to end this option.

Dimension

The **Dimension** option is used to move the dimensions of a sketch. Note that you can move only one dimension at a time. The following steps explain the procedure to move dimensions of a sketch:

1. Choose **Dimension** from the **MOVE ENTITY** submenu in the **Menu Manager**. You will be prompted to select the dimension to be moved.

2. Select the dimension to be moved using the left mouse button. The dimension text is replaced by a red colored box and is attached to the cursor. You are prompted to select a new location to place the dimension.

3. Move the cursor on the graphics screen to move the dimension to the desired location. Place it by using the left mouse button. The dimension text is redisplayed.

4. Repeat step 2 and 3 until you move all the dimensions at the desired place. Use the middle mouse button to end this option.

When the **Intent Manager** is on, a dimension can be moved by pressing and holding the left mouse button on the dimension and moving it. The dimension text is replaced by a red colored box and the cursor is replaced by a hand. You can drag the dimension to the desired location on the graphics screen and release the left mouse button to place the dimension at that point.

SKETCHING ADVANCED GEOMETRY

When the **Intent Manager** is off, you can sketch different advanced geometries in the Sketch mode using the **Adv Geometry** option . Choose **Sketch** from the **SKETCHER** menu and **Adv Geometry** from the **GEOMETRY** submenu in the **Menu Manager**. The **ADV GEOMETRY** submenu appears as shown in Figure 2-14 with different options available to create advanced geometries.

The different options in the **ADV GEOMETRY** submenu are discussed next.

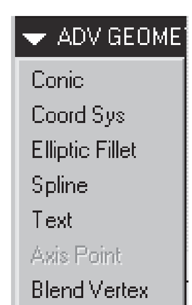


***Figure 2-14** Options in the **ADV GEOMETRY** submenu*

Creating Conic Splines Using the Conic Option

The **Conic** option is used to create conic splines. The following steps explain the procedure to create a conic spline using the **Conic** option:

1. Choose **SKETCHER > Sketch** > **Adv Geometry** > **Conic** from the **Menu Manager**. You will be prompted to select the first endpoint of the conic entity. Using the left mouse button select a point on the graphics screen as the first endpoint of the conic entity.

2. You are now prompted to select the second endpoint of the conic entity. A red rubber-band line appears with the cursor attached to one end. Move the mouse to select the second endpoint of the conic entity at the desired place on the graphics screen using the left mouse button.

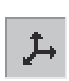
3. A red rubber-band conic spline appears with the cursor attached to it. You are prompted to select the shoulder point of the conic. Move the mouse to size the conic spline. Select the shoulder point using the left mouse button at the desired point. A conic spline is created. The red color of the conic spline is now changed to cyan.

4. Repeat step 1, 2 and 3 until you create all the conic splines required. To abort conic spline creation before the spline is completed, press the middle mouse button.

Creating Coordinate System Using the Coord Sys Option

The **Coord Sys** option is used to create a coordinate system and add it to the sketch you create. Unlike other CAD packages, Pro/ENGINEER does not use the Cartesian coordinate system. In Pro/ENGINEER the user has to define a coordinate system where required. The coordinate system acts as reference for dimensioning. You can dimension the splines using the coordinate system. Thus, it provides you a flexibility to modify the spline points by specifying the different coordinates with respect to the coordinate system. The coordinate system is used in blend features to align different sections in a blend. It is also used in the Assembly mode and the Manufacturing mode of Pro/ENGINEER. The following steps explain the procedure to create a coordinate system:

1. Choose **Adv Geometry** from the **GEOMETRY** submenu and **Coord Sys** from the **ADV GEOMETRY** submenu in the **Menu Manager**. You are prompted to select the location for the coordinate system. Place the coordinate system at the desired points on the screen using the left mouse button. The coordinate system will be placed at as many places as you select on the graphics screen. You can end coordinate system creation by using the middle mouse button.



When the **Intent Manager** is on, choose the **Create reference coordinate system.** button from the **Right Toolchest**. This button is available on the flyout that is displayed when you choose the black arrow on the right of the **Create points.** button. When you choose this button, the reference coordinate system is attached to the cursor. You can place the coordinate system at the desired points on the screen.

Note

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

Creating Elliptical Fillet Using the Elliptic Fillet Option

The **Elliptic Fillet** option is used to create elliptical fillets between any two entities. Elliptical fillets are always created in elliptical shapes, that is, they have the properties of an ellipse.

When the **Intent Manager** is on, choose the **Create an elliptical fillet between two entities.** button. This button is available on the flyout that is displayed when you choose the black arrow on the right of the **Create a circular fillet between two entities.** button.

The following steps explain the procedure to create elliptical fillets:

1. Choose **SKETCHER** > **Sketch** > **Adv Geometry** > **Elliptic Fillet**. You will be prompted to select two entities.

2. Select the first entity using the left mouse button. The color of the entity changes to red. Select the second entity. As soon as you select the second entity, the elliptical fillet is created. The shape of the elliptical fillet depends upon the specified points. After the current elliptical fillet is created, you will again be prompted to select two entities for elliptical fillet.

3. Repeat step 2 until you have created all the fillets.

Creating Splines Using the Spline Option

Splines are curved entities that can pass through an infinite number of intermediate points. Splines can be created in many different ways.

Choose **SKETCHER** > **Sketch** > **Adv Geometry** > **Spline**. The **SPLINE MODE** menu and **TYPE** submenu is displayed with different options available to create the spline. The **SPLINE MODE** menu is modal in nature, that is, each button remains selected until you select another button.

Figure 2-15 shows you different **SPLINE MODE** menu and **TYPE** submenu options available to create splines. The options shown in Figure 2-15 are explained next.

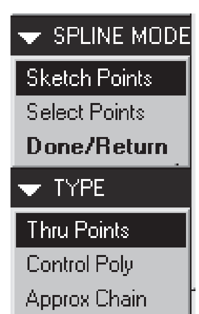


***Figure 2-15** SPLINE MODE submenu and **TYPE** submenu*

Spline created using the Sketch Points option

The **Sketch Points** option creates a spline by picking the points on the screen. The spline passes through all the selected points.

Spline created using the Select Points option

Using the **Select Points** option, the spline is created by selecting the predefined sketcher points. These points are selected for the spline to pass through.

The options under the **TYPE** submenu are discussed next.

Thru Points

The **Thru Points** option is selected by default in the **TYPE** submenu. The following steps explain the procedure to create a spline using this option:

1. When you select this option, you will be prompted to create points for the spline to pass through. Select a point on the graphics screen as the start point of the spline.

2. A red rubber-band line appears from the start point. Select different points on the graphics screen for the spline to pass through. After selecting the points use the middle mouse button to complete spline creation. You can abort spline creation using the right mouse button.

The modification of the spline is possible by using the options of the **MOD SPLINE** submenu. The **MOD SPLINE** submenu shown in Figure 2-16 can be invoked by choosing the **Modify** option in the **SKETCHER** menu and then selecting a spline.

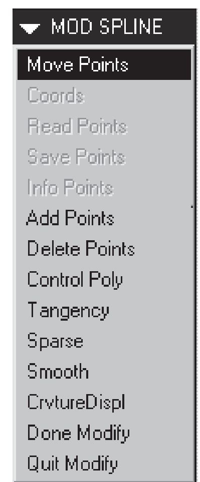


Figure 2-16 *The options in the* ***MOD SPLINE*** *submenu*

Control Poly

The **Control Poly** option is used to create a spline that is enclosed in a polygon such that the endpoints of the spline are tangent to the control polygon. The control polygon of the spline is considered as the parent of the spline. You can dimension the lines of the control polygon as normal sketched entities to modify the shape of the spline. Note that if you delete the control polygon, the spline is not deleted. You can use the **MOD SPLINE** submenu to make some modifications in the shape of this spline.

Approx Chain

The **Approx Chain** option is used to create a spline approximately along the selected chain of entities such as lines, arcs, and so on. The following steps explain the procedure to create splines using the **Approx Chain** option:

1. Choose **Approx Chain** from the **TYPE** submenu. You will be prompted to select section entities forming single continuous chain.

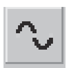
2. Select the entities such as lines, arcs, and so on using the left mouse button. The color of the selected entities changes to red. A spline is created along the selected entities with a control polygon when the middle mouse button is pressed.

Points to remember while deleting the **Approx Chain** splines.

1. When you delete the spline created by the **Approx Chain** option, the chained entities used to create the spline are restored. You may need to use the **Query Bin** to delete the spline.

2. If you delete any entity of the chain, the entire chain will be deleted completely and the spline will be left. This spline would behave like any other spline in the Sketch mode.

3. If you restore the deleted chain by choosing the **Undelete Last** option, the entire chain is restored. If the spline is modified before you restore the change, Pro/ENGINEER ignores the modifications and the original chain as well as the spline is obtained.



When the **Intent Manager** is on, the splines are created using the **Create spline curve through several points.** button from the **Right Toolchest**.

Creating Text Using the Text Option

You can use the **Text** option to enter text in your sketch. The following steps explain the procedure to create text in the sketch:

1. Choose **SKETCHER** > **Sketch** > **Adv Geometry** > **Text**. The **Message Input Window** appears in the **Message** area.

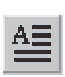
2. Enter text in the **Message Input Window** and press ENTER or choose the green check mark button on the window. You will be prompted to specify a box on the graphics screen to place the text. Specify a point on the graphics screen from where you want the text to start. A red rubber-band rectangle appears as you move the cursor from the point selected. Move it to the desired position and press the left mouse button. The text entered in the **Message Input Window** will be placed at this point on the graphics screen. The **Message Input Window** will appear again to place the other desired text. Remember, you can enter a maximum of 79 characters in a line.

3. Repeat step 2 until you write the required text for your sketch. To end text insertion, press ENTER or select the green button. Now, you have exited the **Message Input Window**. If you want to abort text placement, press the middle mouse button.

4. The text has to be dimensioned with reference to the sketch.

Note

*By default, the font used in the text is **font 3d**. You can also modify the text by choosing **Modify** from the **SKETCHER** menu and selecting the text using the left mouse button. The different options to modify the text will be discussed in the next section of this chapter.*



When the **Intent Manager** is on, you can choose the **Create text as a part of a section.** button to create text. When you choose this button, the system prompts you to select a start point to determine the text height. Using the left mouse button, specify the height of the text by drawing a line. The **Text** dialog box is displayed. The default font of the text is **font 3d**. You can change the font by using the **Text** dialog box.

Using the Blend Vertex Option

The **Blend Vertex** option is used to create blended features having unequal number of segments in their sections. This option is discussed in detail in Chapter 7.

MODIFYING THE TEXT

The text that was written using the **Text** option can be modified and edited. To modify a text, choose **SKETCHER** > **Modify** from the **Menu Manager** and then select the text to modify. When you select a text to modify, the text changes to red color and the **MOD SEC TEXT** submenu is displayed as shown in Figure 2-17. The options in this submenu are discussed next.

***Figure 2-17 MOD SEC TEXT** submenu*

Text Line

When you choose the **Text Line** option, the **Message Input Window** is displayed containing the text selected to modify. In this window you can change the text string or can edit it. After editing the text choose the green button on the window or press ENTER.

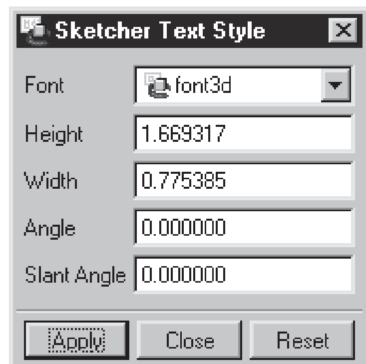


***Figure 2-18 Sketcher Text Style** dialog box*

Text Style

When you choose the **Text Style** option, the **Sketcher Text Style** dialog box is displayed as shown in Figure 2-18. In the **Font** drop-down list there are various fonts available to choose from. The default font is **font 3D**. The different edit boxes available in this dialog box are used to change the properties of the text. After modifying the value in the edit box, choose the **Apply** button to reflect the modifications on the selected text.

SEC TOOLS

The **Sec Tools** (Section Tools) option is used to perform the operations such as place a section, set sketcher environment options, toggle a section, and so on. When you choose **SKETCHER** > **Sec Tools**, the **SEC TOOLS** submenu is displayed. The options under the **SEC TOOLS** submenu are shown in Figure 2-19. Some of these options are discussed next.

***Figure 2-19 SEC TOOLS** submenu*

Copy Draw

The **Copy Draw** option copies the draft entities sketched in the Drawing mode of Pro/ENGINEER to the current sketch. When you choose this option, the **Open** dialog box is displayed. Select the drawing file from where you want to copy the draft entities. The selected drawing is opened in a sub-window. If no entity exists on the graphics screen of the current sketch window, the **SCALE** submenu is displayed that is used to select the options for modifying the size of the draft entities. After setting the size option, you will be prompted to select the entities to copy from the sub-window. After selecting the entities, choose **Done Sel** from the **GET SELECT** menu. If there are no entities in the current file, the section will be automatically placed in the current file. However, if some entity exists on the graphics screen then you need to specify the rotating angle and scale factor. The cursor is attached to the section after you specify the scale factor. To place a section or an entity use the left mouse button.

When the **Intent Manager** is on, the draft entities from a drawing file can be copied by choosing **Sketch > Data from File** from the menu bar.

Place Section

The **Place Section** option places a section saved in a sketch file (.sec) in the same working directory or in a different directory. This option saves time in drawing the same or similar section again.

When you choose this option the **Open** dialog box is displayed. Select the section you want to place in the current sketch from this dialog box. Choose the **Open** button to place the section. If there are no entities in the sketcher environment, the selected sketch will be automatically placed. If there are some entities, the sketch will be opened in a sub-window. The procedure to place a section using this option is the same as discussed in the **Copy Draw** option.

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

Sec Environ

The display of vertices of the section, the display of dimensions, and the display of constraints can be turned on or off from the **SEC ENVIRON** submenu. This submenu is displayed when you choose the **Sec Environ** option from the **SEC TOOLS** submenu. The accuracy of the sketch, the number of decimal places dimensions will have, and the grid parameters can be set from this submenu.

REGENERATION FAILED

As mentioned earlier, if the **Intent Manager** is off you need to regenerate the sketch after you finish sketching. This is done because you need to check whether the design you have made is correct and satisfies all constraints. Sometimes, the sketch may not regenerate successfully. Some of the reasons for the sketch to not regenerate in the sketch mode are listed below.

1. The section is not fully dimensioned.

2. The section is overdimensioned. In this case, you are prompted to delete a dimension or dimensions that appear in red. After deleting the extra dimensions you need to regenerate the sketch again for successful regeneration.

3. Incompatible dimension values. This message is displayed when the dimensions assigned are incompatible with the geometry of the sketch.

When the **Incompatible dimension values** message appears the **SECTION FAIL** submenu is displayed as shown in Figure 2-20. This submenu is displayed as soon as you regenerate the sketch. The options available under this submenu are discussed next.

***Figure 2-20** The **SECTION FAIL** menu*

RestoreDim

The **RestoreDim** option is selected when you want to restore the original dimension values individually. As you select the dimensions to restore, the selected dimension value is changed to the value that was before regeneration.

RestoreAll

The **RestoreAll** option restores the state of the section sketch that was before regeneration. When this option is selected, all the dimensions in the section sketch are restored to their original values that were before regeneration.

RESOLVE SKETCH DIALOG BOX

When the **Intent Manager** is on, while applying constraints or dimensions, the system may sometimes prompt you to delete one or more than one highlighted dimensions or constraints. This is because while adding dimensions or constraints some strong dimensions or constraints conflict with the added dimensions or constraint. As soon as the conflict occurs the **Resolve Sketch** dialog box is displayed as shown in Figure 2-21 and the constraint or the dimension under conflict are displayed in red. When you select a dimension or constraint from the **Resolve Sketch** dialog box, the dimension or constraint selected is enclosed in a yellow box. The buttons available in the **Resolve Sketch** dialog box are discussed next.

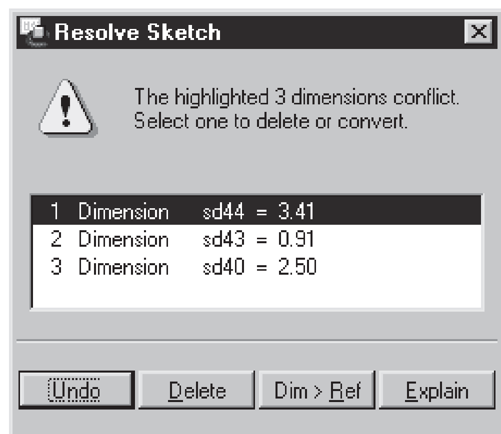


***Figure 2-21** **Resolve Sketch** dialog box*

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 a yellow box. Select the highlighted dimension or the constraint to delete using the left mouse button.

Dim > Ref

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

Explain

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

TUTORIALS

Tutorial 1

In this Tutorial you will place an existing sketch that you had drawn in Tutorial 3 of Chapter 1. After placing the sketch, draw the keyway as shown in Figure 2-22. Do not use the **Intent Manager** in this tutorial. **(Expected time: 15 min)**

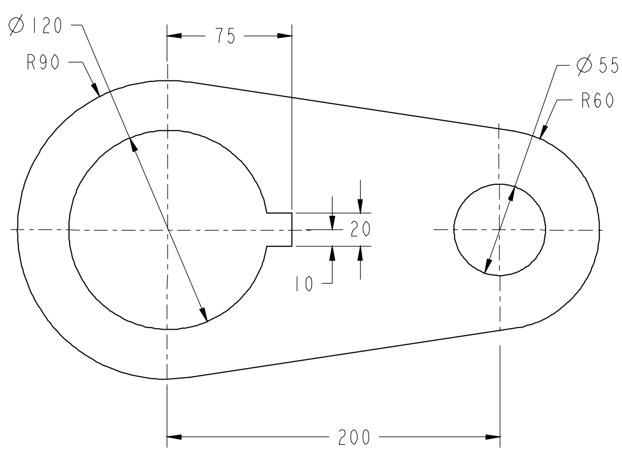


Figure 2-22 *Sketch for Tutorial 1*

The following steps outline the procedure for completing this tutorial.

a. Start Pro/ENGINEER.

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

c. Exit the **Intent Manager** and place the section by using the **Place Section** option.

d. Draw the keyway and dimension it.

e. Modify the dimensions.

f. Regenerate the sketch and then save the sketch.

Starting Pro/ENGINEER

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

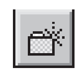
Setting the Working Directory

When Pro/ENGINEER session is started, the first task is to set the working directory. As mentioned earlier, working directory is a directory on your system where you can save the work done in the current session of Pro/ENGINEER. You can set any directory existing on your system as the working directory. Since this is the first tutorial of this chapter, you need to create a folder named **c02** in **C:\ProE** folder, 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. Browse and select **C:\ProE**.

3. Choose the **New Directory** button in the **Select Working Directory** dialog box. The **New Directory** dialog box is displayed.



4. Type **c02** in the **New Directory** edit box. Choose **OK** from the dialog box. You have created a folder named c02 at **C:\ProE**.

5. Choose **OK** from the **Select Working Directory** dialog box. You have set the working directory to **C:\ProE\c02**.

Creating New Object File

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

2. Enter **c02tut1** in the **Name** edit box. Choose the **OK** button.

   You will enter the sketcher environment of the Sketch mode. When you enter the sketcher environment, the **Intent Manager** is on by default. This is evident from the buttons available on the **Right Toolchest**. You can reconfirm this from the check mark that is displayed on the left of the **Intent Manager** option in the **Sketch** menu in the menu bar. Since you need to complete this tutorial without the **Intent Manager**, you need to exit it.

3. To exit the **Intent Manager**, choose **Sketch** > **Intent Manager** from the menu bar to clear the check mark. The **Right Toolchest** is no more displayed and the **Menu Manager** is displayed on the right of the window. The **Menu Manager** contains all the sketcher options.

Placing the Section

1. Choose the **Sec Tools** option from the **SKETCHER** menu in the **Menu Manager**. The **SEC TOOLS** submenu is displayed.

2. Choose the **Place Section** option from this submenu. The **Open** dialog box is displayed with the working directory as the current directory.

3. Choose the **Up One Level** button and then open the **c01** directory. Select **c01tut3-1.sec** and choose the **Open** button from the **Open** dialog box.

   The sketch will be placed on the graphics screen as shown in Figure 2-23.

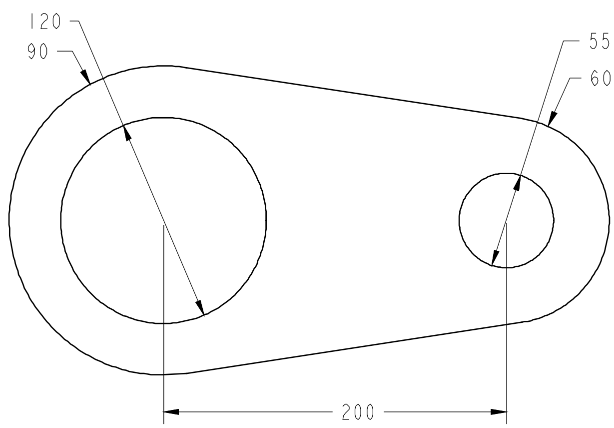


Figure 2-23 *Sketched placed in the current file*

Drawing the Keyway

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

1. Choose **Sketch** > **Line** from the **SKETCHER** menu in the **Menu Manager**. The **LINE TYPE** submenu is displayed.

2. Select the **Horizontal** option from the **LINE TYPE** submenu and draw the keyway as shown in Figure 2-24.

3. Choose the **Geom Tools** option from the **SKETCHER** menu in the **Menu Manager**. The **GEOM TOOLS** submenu is displayed.

4. Choose the **Trim** option from this submenu. The **DRAFT TRIM** submenu is displayed.

5. Choose the **Bound** option from this submenu. You are prompted to select the bounding entity.

6. Select the circle as the boundary. The circle turns red in color. Select the top horizontal line of the keyway and then select the bottom horizontal line of the keyway. This is done to make sure that the lines touch the circle.

7. Choose the **Geom Tools** option from the **SKETCHER** menu in the **Menu Manager**. The **GEOM TOOLS** submenu is displayed.

8. Choose the **Intersect** option from this submenu.

9. Select the circle close to the top horizontal line of the keyway and then select the top horizontal line of the keyway. Similarly, select the circle close to the bottom horizontal line and then select the bottom horizontal line of the keyway.

   The lines and the circle intersect at the points where the lines meet the circle. The intersection points are displayed in yellow color. The portion of the circle that lies between the two horizontal lines of the keyway is separated from the circle.

10. Choose the **Delete** option from the **SKETCHER** menu and delete the part of the circle that lies between the two intersection points.

Dimensioning the Keyway

The dimensions that will be applied to the keyway are shown in Figure 2-22.

1. Choose the **Dimension** option from the **SKETCHER** menu in the **Menu Manager**. The **DIMENSION** submenu is displayed. The **Normal** option is chosen by default.

2. Dimension the keyway as shown in Figure 2-25.

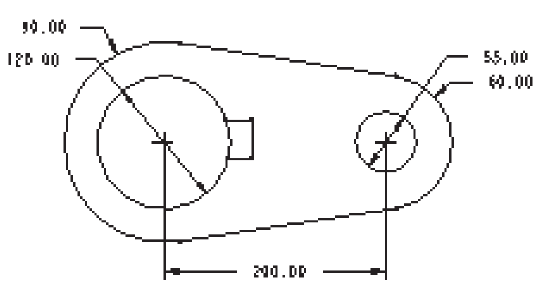


Figure 2-24 *Sketch of the keyway*

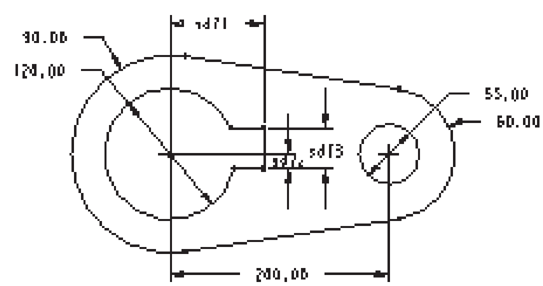


Figure 2-25 *Sketch after dimensioning the keyway*

Modifying the Dimensions

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

1. Choose the **Modify** option from the **SKETCHER** menu in the **Menu Manager**. The **MOD SKETCH** submenu is displayed and the **Mod Entity** option is selected by default.

2. Modify the dimensions of the keyway. For dimension values refer to Figure 2-22.

Regenerating the Sketch

You need to regenerate the sketch because the sketch is completed only when Pro/ENGINEER accepts the sketch. If the sketch drawn is complete and fully dimensioned, and you choose the **Regenerate** option from the **SKETCHER** menu, the **Section regenerated successfully.** message is displayed in the **Message Area**.

1. Choose the **Regenerate** option from the **SKETCHER** menu in the **Menu Manager** to regenerate the sketch. The sketch after regeneration is shown in Figure 2-26.

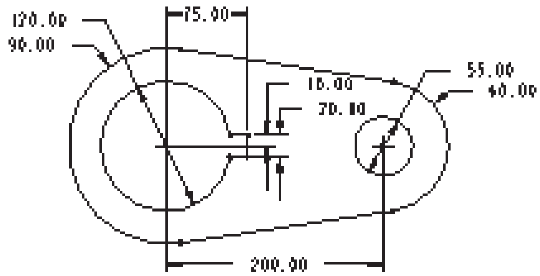


Figure 2-26 *Sketch after regeneration*

Saving the Sketch

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

1. Choose the **Save the active object** button from the **File** toolbar. The **Message Input Window** is displayed with the name of the sketch that you had entered earlier.

2. Press ENTER or choose the check mark button on the window. The sketch is saved.

3. Choose the **Done** option from the **SKETCHER** menu in the **Menu Manager** to exit the Sketch mode.

Tutorial 2

Draw the sketch shown in Figure 2-27 using the **Intent Manager**. **(Expected time: 30 min)**

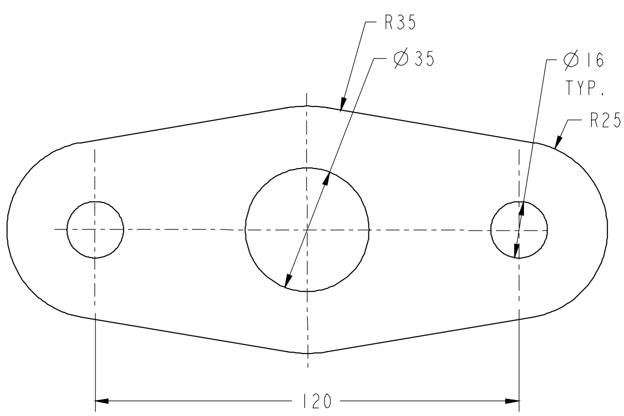


Figure 2-27 *Sketch for Tutorial 3*

The following steps outline the procedure for creating this sketch using the **Intent Manager**:

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

b. Draw the sketch using sketcher tool buttons.

c. Apply the required constraints and dimensions to the entities in the sketch.

d. Modify the dimensions of the sketch and then save the sketch.

Setting the Working Directory

The working directory was selected in Tutorial 1, therefore there is no need to select the working directory again. But if you still want to select the working directory, choose the **Set Working Directory** option from the **File** menu. The **Select Working Directory** dialog box is displayed. Set the working directory to **C:\ProE\c02**.

Creating New Object File

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

Drawing the Sketch

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

1. Choose the **Create circle by picking the center and a point on the circle.** button. Draw the three circles as shown in Figure 2-28.

2. Choose the **Create lines.** button from the **Right Toolchest**. Draw the lines connecting the three circles as shown in Figure 2-29. Later you will apply the tangent constraint on these lines.

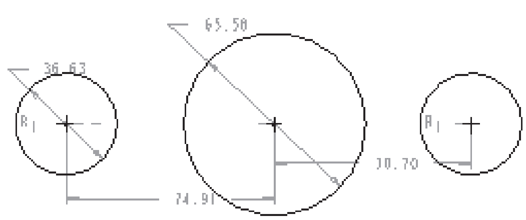


Figure 2-28 Three circles with weak dimensions and constraints

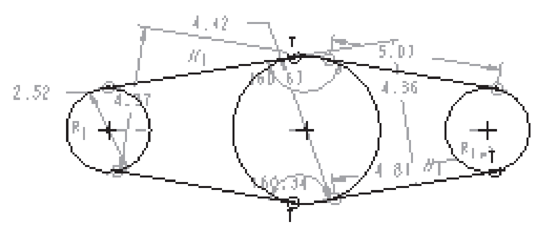
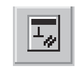


Figure 2-29 Lines joining the three circles with weak dimensions

Applying Constraints

The lines in the sketch are tangent to the circles, therefore you need to apply the tangent constraint to the lines and the circles.

1. Choose the **Impose sketcher constraints on the section.** button from the **Right Toolchest**. The **Constraints** dialog box is displayed.



2. Choose the **Make two entities tangent** button from the **Constraints** dialog box.

3. Select the top left line and the left circle. You will notice that the tangent constraint symbol **T** is applied. This indicates that the selected line is tangent to the selected circle. Similarly, one by one select all the circles and lines to make them tangent. Figure 2-30 shows the sketch after the tangent constraint is applied.

Trimming the Circles

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

1. Choose the **Dynamically trim section entities.** button from the **Right Toolchest**.

2. Bring the cursor close to the right portion of the left circle. The right part of the circle turns magenta in color. Select the right portion of the circle using the left mouse button to remove it.

3. Similarly, trim the part of the middle circle and the right circle. The sketch after trimming the circles is shown in Figure 2-31.

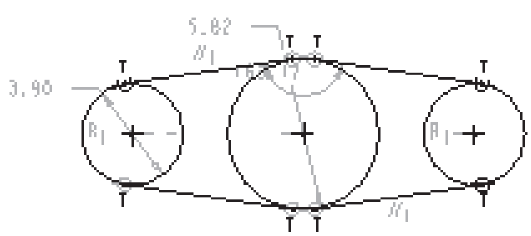


Figure 2-30 *Sketch with tangent constraints*

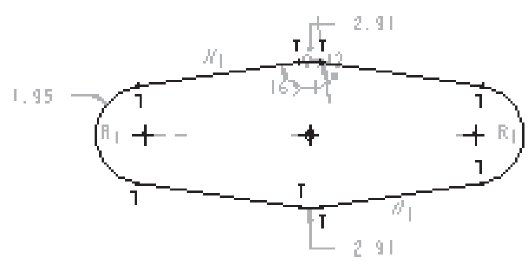
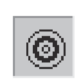


Figure 2-31 *Sketch after trimming the circles*

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 will be prompted to select an arc.



2. Using the left mouse button, select the left arc. You will notice that when you move the

mouse, a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button.

3. Using the left mouse button, select the top arc. You will notice that when you move the mouse a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button to end circle creation.

4. Using the left mouse button, select the right arc. You will notice that when you move the mouse a circle appears. When $\mathbf{R_2}$ symbol appears on the circle, use the left mouse button to complete the circle. Press the middle mouse button to end circle creation.

   The $\mathbf{R_2}$ symbol appears on both the left and the right circle indicating that their radius is same. The constraint symbols like R_1 or R_2 varies from sketch to sketch.

Applying the Equal Radii Constraint

1. Choose the **Impose sketcher constraints on the section.** button from the **Right Toolchest**. The **Constraints** dialog box is displayed.

2. Choose the **Create Equal Lengths, Equal Radii, or Same Curvature constraint** button from the **Constraints** dialog box.

3. Select the left arc and then select the right arc to apply the equal radius constraint.

4. Select the left circle and the right circle to apply the equal radius constraint.

5. Use the CTRL+left mouse button to zoom in and now select the top arc and the bottom arc to apply the equal radius constraint. If the constraint is already applied, the **Resolve Sketch** dialog box is displayed. Choose the **Undo** button from this dialog box.

Dimensioning the Sketch

1. Choose **Create defining dimensions.** button from the **Right Toolchest**.

2. Select the center of right and left circles. The center of the circles turns red in color. Now, using the middle mouse button, place the dimension below the sketch.

Modifying the Dimensions

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

1. Select all the dimensions using CTRL+ALT+A.

2. Choose the **Modify the values of dimensions, geometry of splines, or text entities.** button. The **Modify Dimensions** dialog box is displayed.

3. Clear the **Regenerate** check box and then modify the values of the dimensions. When you clear the check box, the sketch does not regenerate as you modify the dimensions.

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

4. Modify all the dimensions. Refer to Figure 2-27 or Figure 2-32 for dimension values.

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

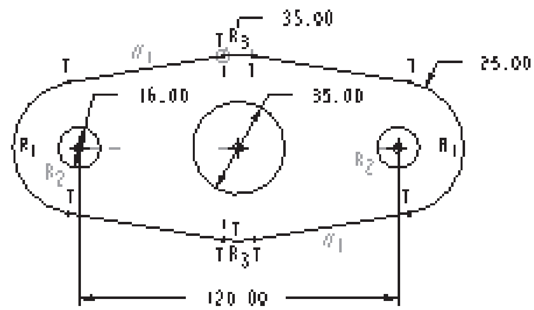


Figure 2-32 The required sketch with constraints and dimensions

Saving the Sketch

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

Exiting the Sketch Mode

1. Choose the **Continue with the current section.** button and exit the Sketch mode.

Tutorial 3

Draw the sketch shown in Figure 2-33 without using the **Intent Manager**.

(Expected time: 30 min)

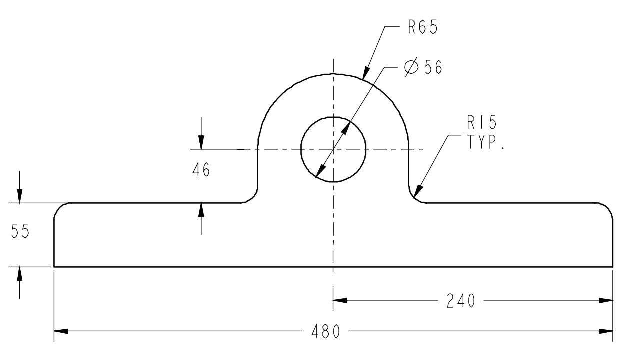


Figure 2-33 Sketch for Tutorial 3

The following steps outline the procedure for creating this sketch without using the **Intent Manager**:

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

b. Exit the **Intent Manager** and draw the sketch using sketcher options.

c. Autodimension the sketch.

d. Modify dimensions of the sketch and regenerate the sketch.

Setting the Working Directory

The working directory was selected in Tutorial 1, therefore there is no need to select the working directory again. But if you still want to select the working directory, choose the **Set Working Directory** option from the **File** menu. The **Select Working Directory** dialog box is displayed. Set the working directory to **C:\ProE\c02**.

Creating New Object File

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

2. Exit the **Intent Manager** by choosing the **Intent Manager** option from the **Sketch** menu in the menu bar.

   The **Right Toolchest** is no more displayed and the **Menu Manager** is displayed.

Drawing a Rectangle

1. Choose the **Rectangle** option from the **GEOMETRY** submenu in the **Menu Manager**. Draw a rectangle as shown in Figure 2-34.

Dividing the Line

The rectangle drawn will be divided at two points on the top horizontal line. As mentioned earlier, when an entity is divided, it is converted into separate entities.

1. Choose the **Geom Tools** option from the **SKETCHER** menu in the **Menu Manager**. The **GEOM TOOLS** submenu is displayed.

2. Choose the **Divide** option from the **GEOM TOOLS** submenu. You are prompted to select an entity to divide.

3. Select points on the top line as shown in Figure 2-34 to divide.

Deleting the Line

The line that is divided is now converted into three separate entities.

1. Choose **SKETCHER > Delete** from the **Menu Manager**. Select the line between the two division points to delete the line segment as shown in Figure 2-35.

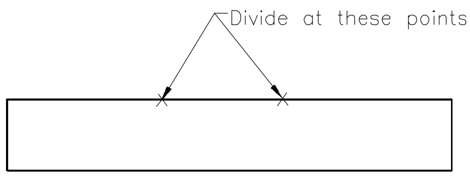


Figure 2-34 *Points to be selected to divide*

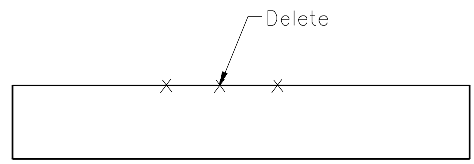


Figure 2-35 *The part of the line to be deleted*

Drawing the Upper Portion of the Sketch

The bottom portion of the sketch is drawn and now the top portion will be drawn. The top portion of the sketch consists of two vertical lines, a tangent arc, and a concentric circle.

1. Choose **SKETCHER** > **Sketch** > **Line** > **Geometry** > **Vertical** from the **Menu Manager**.

   Draw two vertical lines starting from the points where the line was divided as shown in Figure 2-36.

2. Choose the **Arc** option from the **GEOMETRY** submenu. The **ARC TYPE** submenu is displayed.

   The **Tangent End** option is selected by default in the **ARC TYPE** submenu.

3. Select the endpoint of the left vertical line as the start point of the arc. Complete the arc at the endpoint of the right vertical line.

4. Choose the **Circle** option from the **GEOMETRY** submenu in the **Menu Manager**. The **CIRCLE TYPE** submenu is displayed. Choose the **Concentric** option. You will be prompted to select an arc to which the circle to be drawn will be concentric.

5. Select the arc using the left mouse button. Now, using the left mouse button specify a point on the graphics screen. A red rubber-band circle appears. Size the circle by moving the cursor and use the left mouse button to complete it.

Note

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

Filleting

1. Choose **SKETCHER** > **Sketch** > **Arc** > **Fillet** from the **Menu Manager**. You will be prompted to select the two entities to fillet. The corners that you need to fillet are shown in Figure 2-37.

2. Select the two entities one by one using the left mouse button. The corner of the selected lines will be filleted. Fillet all the required corners shown in Figure 2-37.

   The sketch after filleting is shown in Figure 2-38.

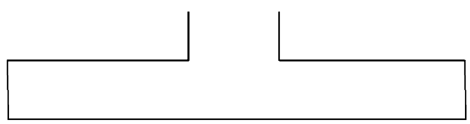


Figure 2-36 *Lines drawn using the* ***Vertical*** *option*

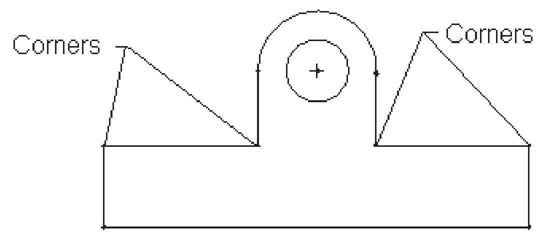


Figure 2-37 *Fillet corners*

Autodimensioning the Sketch

When a sketch is autodimensioned, Pro/ENGINEER assigns some default dimension values to the sketch. You can then modify these dimension values.

1. Choose the **AutoDim** option from the **SKETCHER** menu in the **Menu Manager**.

   When you use the **AutoDim** option, the sketch is automatically regenerated and strong constraints and dimensions are applied to the sketch, see Figure 2-39. Retain only the dimensions shown in Figure 2-33 and delete the remaining dimensions. If some of the

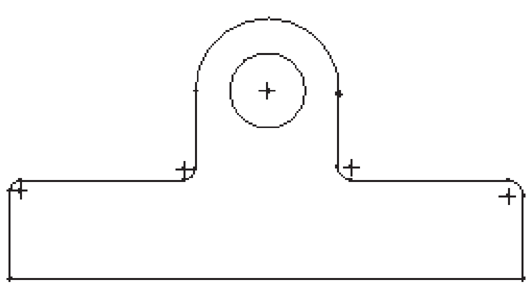


Figure 2-38 *Sketch after filleting the corners*

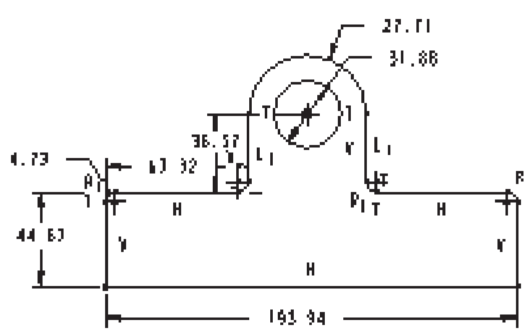


Figure 2-39 *Sketch after autodimensioning*

dimensions shown in Figure 2-33 are not displayed, create them manually by using the **Dimension** option from the **SKETCHER** menu in the **Menu Manager**.

Tip: *In Figure 2-39 the dimensions are moved so that the clarity of the sketch is improved. You can move the dimensions by choosing the **Geom Tools** option from the **SKETCHER** menu and from the **GEOM TOOLS** submenu that appears, choose the **Move Entity** option. You are prompted to select an entity. Select the dimension text, the selected dimension turns red in color. Now, move the cursor and using the left mouse button, place the dimension at the desired location on the graphics screen.*

Modifying the Dimensions

1. Choose **SKETCHER** > **Modify** > **Mod Entity** from the **Menu Manager**. Select the dimension to modify using the left mouse button. The selected dimension changes to red color and the **Message Input Window** is displayed in the **Message Area**.

2. Enter the required dimension in this window. Similarly, modify all the dimensions as required. See Figure 2-33 for dimension values.

Regenerating the Sketch

1. Choose the **Regenerate** option from the **SKETCHER** menu in the **Menu Manager**. The sketch is regenerated successfully, see Figure 2-40.

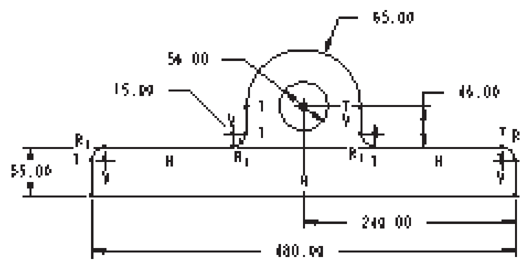


Figure 2-40 Sketch after regeneration

Saving the Sketch

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

Exiting the Sketch Mode

1. Choose the **Done** option from the **SKETCHER** menu from the **Menu Manager** to exit the Sketch mode.

Self-Evaluation Test

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

1. The variable dimension created using the **Perimeter** option can be modified directly by selecting it using the left mouse button. (T/F)

2. If the variable dimension is deleted then the perimeter dimension is also deleted. (T/F)

3. When you modify a weak dimension it becomes strong. (T/F)

4. You can dimension the length of a center line. (T/F)

5. Center lines and planes split when intersected. (T/F)

6. The splines created by using the __________ option can be dimensioned to the coordinate system.

7. When you are using the **Intent Manager**, the __________ dialog box is used to modify dimensions.

8. The display of dimensions and constraints can be turned on/off by choosing the __________ and __________ button respectively from the **Sketcher** toolbar.

9. The __________ button is used to rotate selected entities when the **Intent Manager** is on.

10. When the **Intent Manager** is on, you can delete the entities by selecting them and then using the __________ key.

Review Questions

Answer the following questions:

1. Which of the following symbol represents the normal dimension in the Sketch mode?

   (a) d (b) sd
   (c) ref (d) kd

2. Using which submenu the display of vertices of the section, the display of dimensions, and display of constraints can be made off or on?

   (a) **DIMENSION** (b) **SEC ENVIRON**
   (c) **MOD SPLINE** (d) **GEOMETRY**

3. When the **Intent Manager** is off, a sketch is modified by using the __________ submenu?

   (a) **GEOM TOOLS** (b) **GEOMETRY**
   (c) **MOD SKETCH** (d) None

4. Which of the following submenu has the **Intersect** option?

   (a) **GEOMETRY** (b) **DIMENSION**
   (c) **MOD SKETCH** (d) **GEOM TOOLS**

5. Which type of dimensioning uses relations?

   (a) **Normal** (b) **Perimeter**
   (c) **Baseline** (d) **Known**

6. While placing a section in a new sketch, use the right mouse button to place the section. (T/F)

7. There is only one method to make the grids on or off, that is, using the **Sketcher** toolbar button. (T/F)

8. The reference dimensions appear with the suffix REF. (T/F)

9. While using the **Intent Manager**, you can modify the dimensions dynamically. (T/F)

10. The modification of a spline is possible by using the **Move Points** and **Control Poly** options of the **MOD SPLINE** menu. (T/F)

Exercises

Exercise 1

Create the sketch shown in Figure 2-41 and dimension it. Draw the sketch using the **Intent Manager**. **(Expected time: 30 min)**

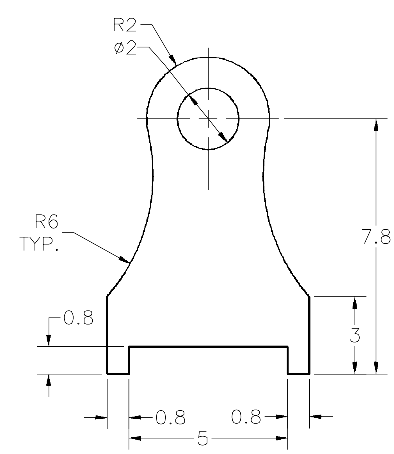


Figure 2-41 *Sketch for Exercise 1*

Exercise 2

Draw the sketch shown in Figure 2-42. Use the **AutoDim** option for dimensioning the sketch. **(Expected time: 15 min)**

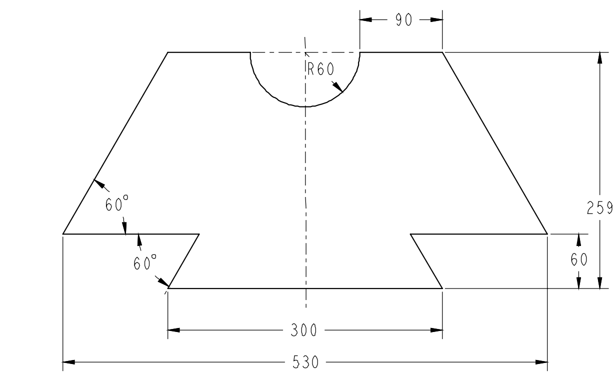


Figure 2-42 *Sketch for Exercise 2*

Exercise 3

Draw the sketch shown in Figure 2-43. Use the **AutoDim** option to dimension the sketch.
(Expected time: 30 min)

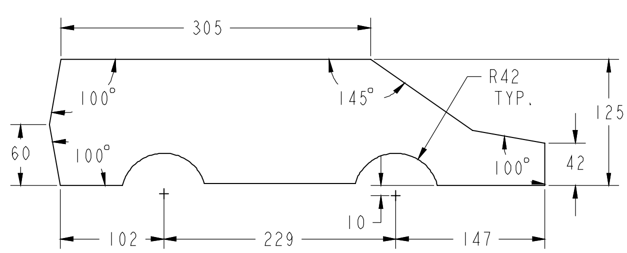


Figure 2-43 *Sketch for Exercise 3*

Answers to the Self-Evaluation Test

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