Chapter 3

Creating Sketches in the Sketch Mode-II

Learning Objectives

After completing this chapter, you will be able to:

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

DIMENSIONING THE SKETCH

In Chapter 2, you learned dimensioning a sketch using the **Normal** tool from the **Dimension** group. In this chapter, you will learn the use of the **Baseline** tool for dimensioning a sketch.

Dimensioning a Sketch Using the Baseline Tool

Ribbon: Sketch > Dimension > Baseline

In PTC Creo Parametric, the **Baseline** tool is used to create dimensions in terms of horizontal and vertical distance values of an entity with respect to a specified baseline. This type of dimensioning in a drawing makes writing a CNC program for manufacturing a component easy.

The **Baseline** tool is available in the **Dimension** group. Using this tool, you can dimension a line, arc, conic, and so on. The following steps explain the procedure to create dimensions using the **Baseline** tool:

1. Choose the **Baseline** tool from the **Dimension** group, refer to Figure 3-1.

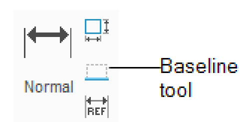


*Figure 3-1 The **Dimension** group*

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

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

   You can dimension arcs, circles, and splines by using the two options that are displayed on invoking the **Baseline** tool. If you select the center of a circle or an arc for baseline dimensioning and press the middle mouse button, the **Dim Orientation** dialog box will be displayed, as shown in Figure 3-2. Also, you will be prompted to select the orientation. Select the required radio button from this dialog box and choose the **Accept** button; the dimension will be placed based on the orientation selected.

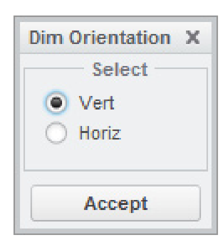


*Figure 3-2 The **Dim Orientation** dialog box*

3. Next, choose the **Normal** tool from the **Dimension** group. Select the baseline dimension that was placed earlier and then select the entity to dimension. Now, press the middle mouse button to place the dimension.

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

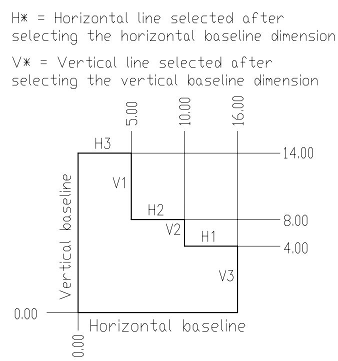


Figure 3-3 *Baseline dimensioning of a sketch*

Replacing the Dimensions of a Sketch Using the Replace Tool

Ribbon: Sketch > Operations > Replace

The **Replace** tool is used to replace a dimension with a new dimension in 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** tool:

1. Choose the **Replace** tool from the menu displayed on clicking the down arrow adjacent to the **Operations** group; you will be prompted to select the dimension to be replaced.

2. Select the dimension to be replaced; the selected dimension will be deleted and you will be prompted to create a replacement dimension. Select the entities between which you want to create a new dimension; the previous dimension will be replaced by a new dimension.

CREATING FILLETS

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

1. Circular fillets
2. Elliptical fillets

Creating Circular Fillets

Ribbon: Sketch > Sketching > Fillet > Circular

A circular fillet is the arc formed at the intersection of two lines, a line and an arc, or two arcs. This type of fillet is controlled by the radius or diameter dimension of the fillet. The resulting fillet will depend on the location where the elements are selected.

Figure 3-4 shows two non-parallel lines and Figure 3-5 shows the circular fillet created between them. The circular fillet thus created is an arc with its endpoints tangent to the two lines. This is evident from the **T** symbol that is automatically applied to the endpoints of the fillet arc.

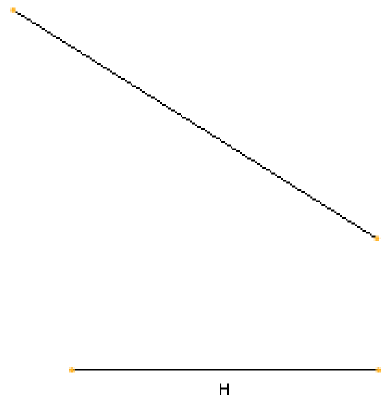


Figure 3-4 *Two lines that do not join*

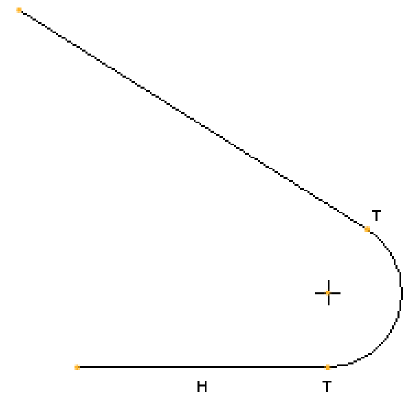


Figure 3-5 *Fillet created between the two lines*

Figure 3-6 shows two lines that join at a point and Figure 3-7 shows the circular fillet created at the joint.

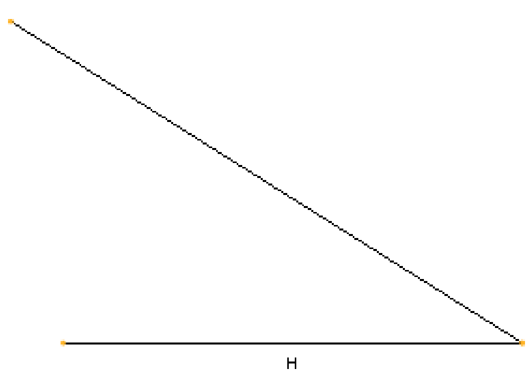


Figure 3-6 *Two lines joining at a point*

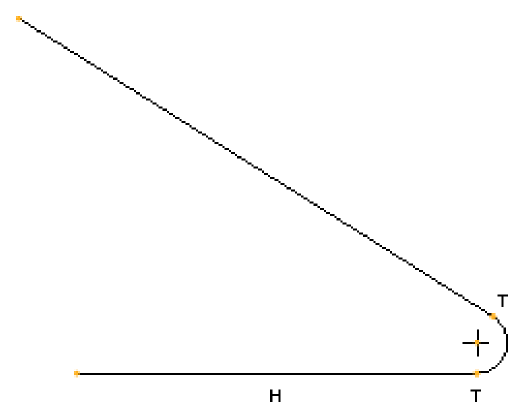


Figure 3-7 *Filleted corner*

Figure 3-8 shows two sets of arcs and Figure 3-9 shows the circular fillet created between the two arcs. The location where you select the arcs to create the fillet is important. The fillet is created tangent to the selection points on the arcs. Here, the endpoints of the arcs are selected to create the fillet.

If the points of selection on the two arcs are away from the endpoints of the arcs, then the fillet is created at the selection points. The portion of the arc that extends beyond the fillet should be manually deleted or trimmed. Figure 3-10 shows the points at which the two arcs are selected and Figure 3-11 shows the fillet created at the selection points.

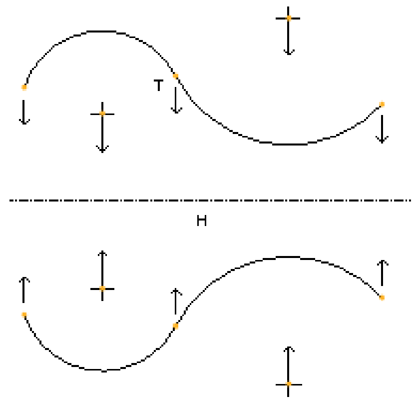


Figure 3-8 *Two sets of arcs*

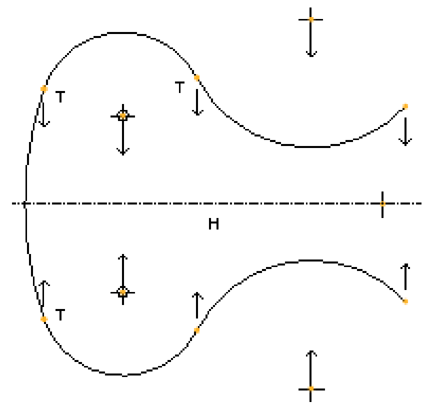


Figure 3-9 *Fillet created by selecting the endpoints*

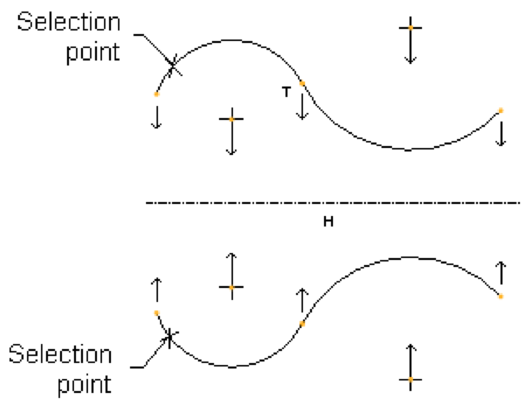


Figure 3-10 *Points selected on arcs*

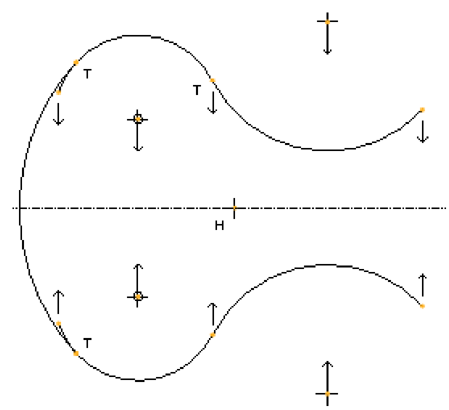


Figure 3-11 *Fillet created*

To create circular fillets, there are two options named **Circular** and **Circular Trim**. These options are available in the **Fillet** drop-down. These options are discussed next.

As mentioned earlier, the **Circular** option is used to create circular fillets. When you create fillet using this option, the filleting entities do not get trimmed but the part of entity after the fillet converts into construction geometry.

The **Circular Trim** option is also used to create circular fillets, but in this case the portion extending beyond the entities get trimmed automatically.

Procedure to create circular fillets using both the options is same and it is discussed next.

1. Choose the **Circular** or **Circular Trim** option from the **Fillet** drop-down in the **Sketching** group; you will be prompted to select two entities.

2. Select the first entity for filleting by using the left mouse button; the red color of the first entity changes to green. Now, select the second entity. It will be drawn between the two selected entities as soon as you select the second entity.

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

Creating Elliptical Fillets

Ribbon: Sketch > Sketching > Fillet > Elliptical

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

The advantage of elliptical fillets over circular fillets is that the geometry of elliptical fillets can be controlled by dimensions in two directions. Therefore, when an elliptical fillet is dynamically modified, its geometry can be controlled in either the x-direction or the y-direction resulting in more curved geometric shape than a circular fillet.

Figures 3-12 and 3-13 illustrate the elliptical fillet. Notice that a strong tangent constraint **T** is automatically applied when you create a fillet.

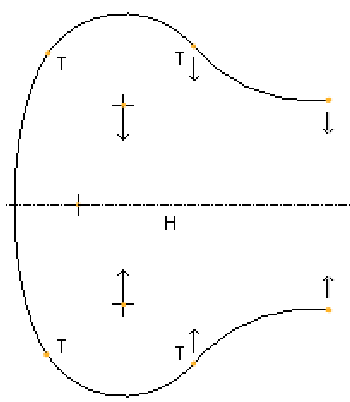


Figure 3-12 *Arcs to be filleted*

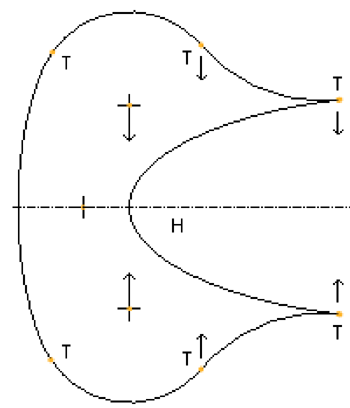


Figure 3-13 *Elliptical fillet created*

Similar to circular fillet, there are two options to create elliptical fillet, **Elliptical** and **Elliptical Trim**. The procedure to create elliptical fillets using any of these options is same. The procedure to create elliptical fillets is discussed next.

1. Choose the **Elliptical** or **Elliptical Trim** tool from the **Fillet** drop-down; you will be prompted to select two entities.

2. Select the first entity by clicking; the color of the entity changes to green.

3. Select the second entity. As soon as you select the second entity, the elliptical fillet is created. The shape of the elliptical fillet depends upon the specified points. After the fillet is created, you will be again prompted to select two entities for elliptical fillet.

4. Repeat steps 2 and 3 until you have created all fillets.

When you select an elliptical fillet to dimension, the **Ellipse Rad** dialog box will be displayed, as shown in Figure 3-14. There are two radio buttons in this dialog box. When the **Major Axis** radio button is selected, the elliptical fillet will be dimensioned radially along the X-direction. The **Minor Axis** radio button, when selected, dimensions the elliptical fillet radially in the Y-direction.

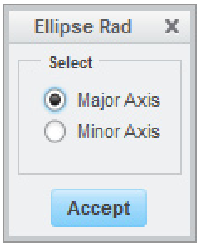
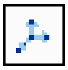


*Figure 3-14 The **Ellipse Rad** dialog box*

CREATING A REFERENCE COORDINATE SYSTEM

There are two types of coordinate systems, construction and geometric. As discussed earlier, the construction entities cannot be referenced outside the Sketcher environment whereas, the geometric entities can be. The **Coordinate System** tool in the **Datum** group is used to create a coordinate system that will act as a reference for dimensioning. You can dimension the splines using the coordinate system. Thus, it provides you the flexibility to modify the spline points by specifying different coordinates with respect to the coordinate system.



The user-defined coordinate system is used in blend features to align different sections in a blend. It is also used in the **Assembly** and **Manufacturing** modes of PTC Creo Parametric.

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

1. Choose the **Coordinate System** tool from the **Datum** group; you will be prompted to select the location for the coordinate system. The coordinate system symbol is attached to the cursor.

2. Place the coordinate system at the desired points on the screen by clicking the left mouse button. The coordinate system will be placed at as many places as you click in the graphics window. You can end the coordinate system creation by using the middle mouse button.

Note

If you add a coordinate system to a sketch, it must be dimensioned. But if 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, it need not be dimensioned. In other words, a coordinate system must be referenced to an entity in a sketch.

WORKING WITH SPLINES

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

Creating a Spline

Ribbon: Sketch > Sketching > Spline

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

1. Choose the **Spline** tool from the **Sketching** group; you will be prompted to select the location for spline.

2. Use the left mouse button to select the start point for the spline. Similarly, select additional points in the graphics window; a spline will be drawn passing through all specified points. Press the middle mouse button to end the creation of spline. All points through which the spline passes are called interpolation points.

Dimensioning of Splines

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

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

Dimensioning the Endpoints

To dimension a spline by selecting the endpoints, you need to follow the steps given below:

1. Choose the **Normal** tool from the **Dimension** group.

2. Select the two endpoints of the spline and place the horizontal or vertical dimension by pressing the middle mouse button. Figure 3-15 shows a spline that is dimensioned by selecting the endpoints.

Radius of Curvature Dimensioning

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

1. Choose the **Normal** tool from the **Dimension** group.

2. Select the endpoint of the spline where the tangency is defined.

3. Press the middle mouse button to place the dimension. Figure 3-16 shows the radius of curvature dimensioning of a spline.

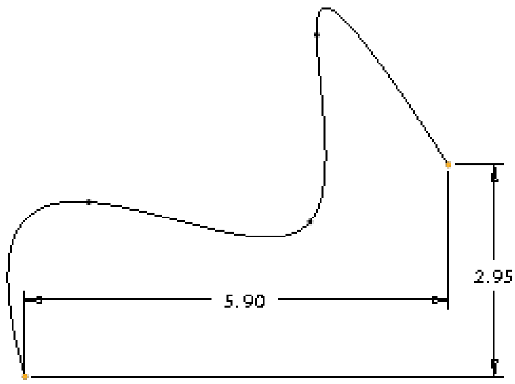


Figure 3-15 Endpoint dimensioning

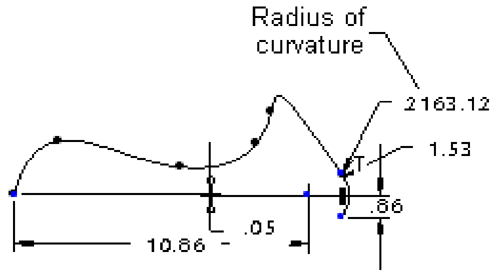


Figure 3-16 Radius of curvature dimensioning

Tangency or Angular Dimensioning

A spline can be dimensioned angularly with respect to a line tangent to it. This type of dimensioning is also called angular dimensioning. To angular dimension a spline and a line tangent to it, you need to follow the steps given below:

1. Choose the **Normal** tool from the **Dimension** group.
2. Select the spline by clicking the left mouse button.
3. Select the entity tangent to the spline by clicking the left mouse button.
4. Select the interpolation point of the spline that is to be dimensioned tangentially.
5. Press the middle mouse button to place the dimension.

Coordinate Dimensioning

The spline can be dimensioned with respect to a user-defined coordinate system. Choose the **Coordinate System** tool from the **Sketching** group. The coordinate system is attached to the cursor. Place the coordinate system in the graphics window. Now, the spline can be dimensioned with respect to the coordinate system.

Dimensioning the Interpolation Points

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

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

Modifying a Spline

In PTC Creo Parametric, a spline can be modified by:

1. Moving the interpolation points of the spline.
2. Adding points to a spline.
3. Deleting points of a spline.
4. Creating a control polygon and moving its control points.
5. Modifying the dimensions of the spline.

Moving the Points of a Spline

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

Alternatively, select the spline and hold the right mouse button to invoke the shortcut menu. Choose the **Modify** option from the shortcut menu or double-click on the spline; the **Spline** tab will be displayed in the **Ribbon** with the options and buttons to modify a spline.

The interpolation points of the spline appear in black crossmarks in the graphics window. Drag the interpolation points to modify the shape of the spline.

Adding Interpolation Points to a Spline

To add interpolation points to a spline, choose the **Spline** tab. Now, right-click on the spline to invoke a shortcut menu. Next, choose the **Add Point** option; a point is added to the spline where the spline was selected. The new point appears in black crossmark. You cannot increase the length of the spline by adding points before the start point and after the endpoint of the spline.

Deleting Interpolation Points of a Spline

To delete a point or a vertex, choose the **Spline** tab. Next, right-click on the vertex to be removed to invoke the shortcut menu. Choose the **Delete Point** option from the shortcut menu; the selected point is deleted. You can continue deleting vertices or points from a spline until only two end points are left in the spline.

Creating a Control Polygon and Moving its Control Points

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

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

Modifying the Dimensions of the Spline

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

WRITING TEXT IN THE SKETCHER ENVIRONMENT

Ribbon: Sketch > Sketching > Text

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

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

1. Choose the **Text** tool from the **Sketching** group; you will be prompted to select the start point of line to determine the text height and orientation.

2. Specify the start point on the screen by clicking the left mouse button; you will be prompted to select the second point of line to determine the text height and orientation.

3. Note that to write the text upright, the second point should be above the start point and in a straight line. If the second point is below the start point, the text will be written down from right to left. Specify the second point on the screen by clicking the left mouse button; the **Text** dialog box will be displayed, as shown in Figure 3-17.

   After specifying the second point, a construction line is drawn having height equal to the distance between the two points. The height and orientation of the text depends on the height and angle of the construction line. If the construction line is drawn at an angle, then the text will be written at that angle.

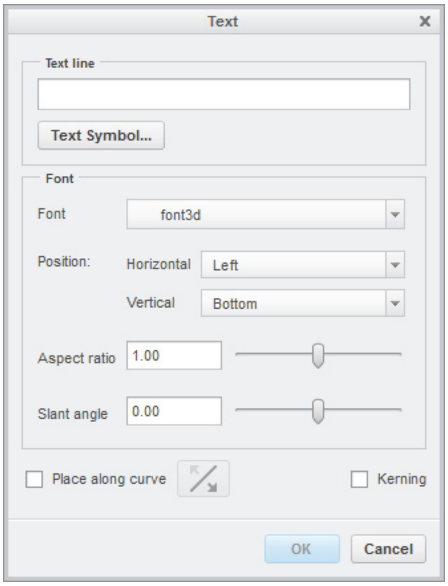


*Figure 3-17 The **Text** dialog box*

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

5. Choose the **OK** button in the **Text** dialog box to exit it.

Note

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

ROTATING AND RESIZING ENTITIES

Ribbon: Sketch > Editing > Rotate Resize

The sketches can be scaled or rotated by using the **Rotate Resize** tool available in the **Editing** group. Select a sketch and then choose the **Rotate Resize** tool from the **Editing** group. On choosing this tool, the sketch, which consists of various entities, will act as a single entity. Also, the sketch appears green in color and is enclosed within a boundary box, as shown in Figure 3-18.

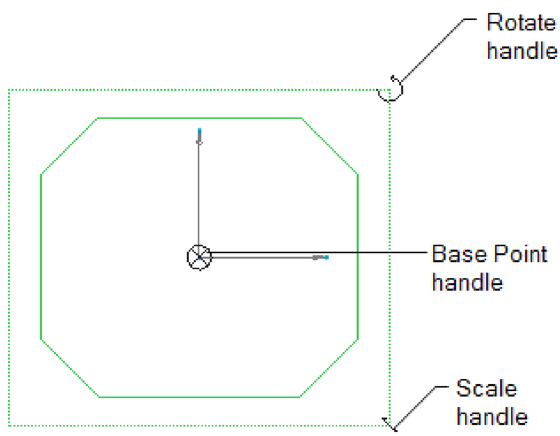


Figure 3-18 *Selected entities enclosed within a boundary box with three handles*

There are three handles that facilitate in scaling, rotating, and moving the selected sketch. The rotate handle is used to dynamically rotate the selected entities. The scale handle is used to dynamically scale the selected entities. The base point handle is used to pick the sketch and place it at any other location in the graphics window. To change the location of any of the three handles, right-click on the handle and drag it by pressing and holding the right mouse button; the selected handle will move along with the cursor. Place the symbol at the desired location. The following steps explain the procedure to move and resize a sketch:

1. Select the sketch to be rotated and scaled, and then choose the **Rotate Resize** tool; the **Rotate Resize** tab will be activated in the **Ribbon**. The options in these areas are used to move, scale, and rotate the sketch dynamically or by entering a value in the respective edit boxes. You can also select a reference about which you want to translate, rotate, or scale the sketch.

2. To move the sketch dynamically, select the move handle and then drag the handle to the required location; the sketch will be repositioned at the new location. You can also select a reference from the drawing area about which you want to move the sketch and the reference will be displayed in move collector of the dashboard.

3. To dynamically rotate the sketch, select the rotate handle and then move the cursor; the sketch will rotate as you move the cursor. You can also enter the rotation angle in the **Rotate** edit box.

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

5. After the sketch has been moved and resized, choose the **Done** button from the **Rotate Resize** tab.

IMPORTING 2D DRAWINGS IN THE SKETCH MODE

Ribbon: Sketch > Get Data > File System

The two-dimensional (2D) drawings when opened in the sketcher environment can be saved in the *.sec* format. The *.sec* file can be used to create a solid model. You can use a prestored sketch by importing it in the modeling environment. The **File System** button in the **Get Data** group is used to import 2D sketches. Using this button, you can save time in drawing the same or similar section again. The file formats from which the data can be imported are shown in Figure 3-19.

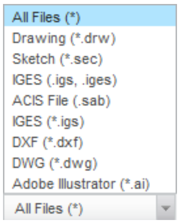


Figure 3-19 *File formats*

When you choose the **File System** button from the **Get Data** group; the **Open** dialog box will be displayed, as shown in Figure 3-20. You can use this dialog box to select and open the file.

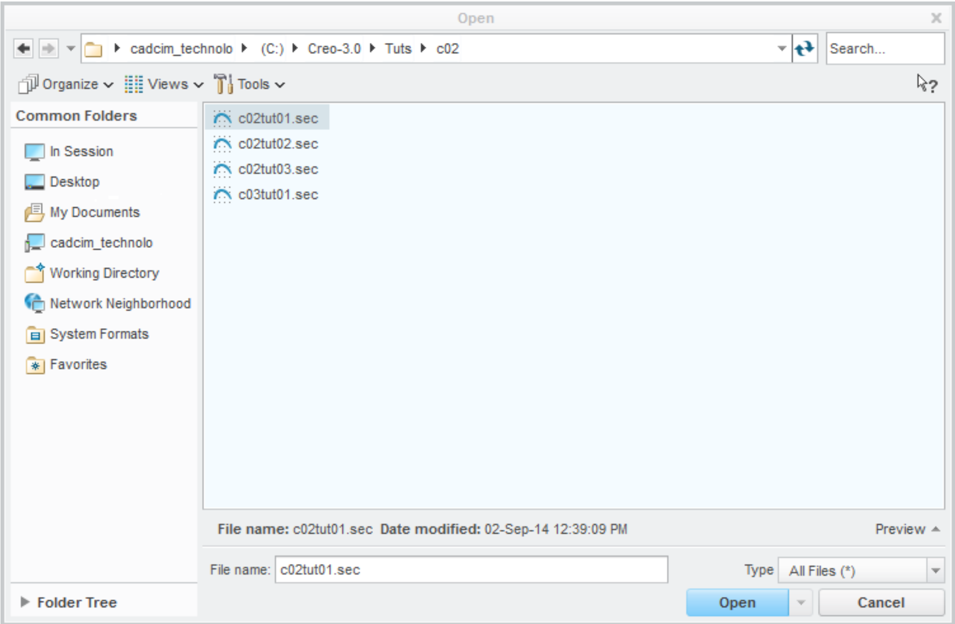


Figure 3-20 *The* ***Open*** *dialog box*

In the **Open** dialog box, if you select a drawing file created in the **Drawing** mode of PTC Creo Parametric, the draft entities of that file will be imported and selected drawing will be opened in a window. Also, you will be prompted to select the entities to copy from the window. Select the draft entities and then press the middle mouse button; the window disappears and a plus sign gets attached to the cursor indicating that you need to select a point on the PTC Creo Parametric screen to insert the file. Select a point on the screen; the selected entities get inserted and are displayed within an enclosed boundary. Also, the **Import Section** tab will be displayed. Use this dialog box to set the position, scale, and orientation of the sketch. Note that if the *.drw* file does not consist of draft entities, no data will be imported.

In the **Open** dialog box, if you select a *.sec* file that will be created in the sketcher environment, the sketch will be displayed in the graphics window enclosed within a boundary.

The section imported using the **File System** button in the current sketch is an independent copy. The imported section will be no longer associated with the source section. The units, dimensions, grid parameters, and accuracy are acquired from the current sketch.

Tip: *You can turn on or off the display of the vertices of the section, the dimensions, and the constraints from the* ***Sketching*** *group.*

TUTORIALS

Tutorial 1

In this tutorial, you will import an existing sketch that you had drawn in Tutorial 3 of Chapter 2. After placing the sketch, draw the keyway, as shown in Figure 3-21.

(Expected time: 15 min)

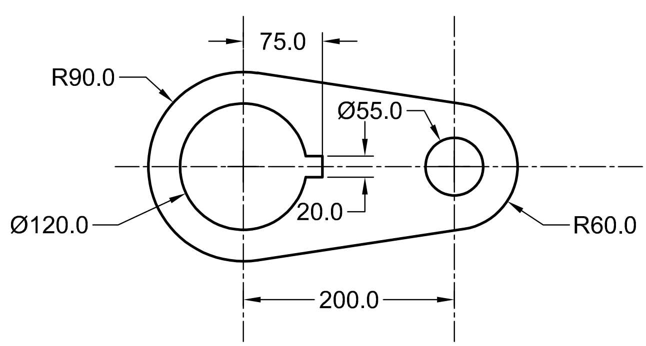


Figure 3-21 *Sketch for Tutorial 1*

The following steps are required to complete this tutorial:

a. Start PTC Creo Parametric.
b. Set the working directory and create a new object file.

c. Import the section by using the **File System** button, refer to Figure 3-22.
d. Draw the keyway and dimension it, refer to Figures 3-23 and 3-24.
e. Modify the dimensions, refer to Figure 3-25.
f. Save the sketch and exit the sketcher environment.

Starting PTC Creo Parametric

1. Start PTC Creo Parametric by double-clicking on the PTC Creo Parametric icon on the desktop of your computer.

Setting the Working Directory

When the PTC Creo Parametric session starts, 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 PTC Creo Parametric. You can set any existing directory on your system as the working directory. Since this is the first tutorial of this chapter, you need to create a folder named *c03* in the *C:\Creo-3.0* folder.

1. Choose **Manage Session > Select Working Directory** option from the **File** menu; the **Select Working Directory** dialog box is displayed.

2. Select *C:>Creo-3.0*. If this folder does not exist, then first create it and then set the working directory. Alternatively, you can use the Folder Tree available on the bottom left corner of the screen to set the working directory. To do so, click on the **Folder Tree** node; the Folder Tree will expand. In the Folder Tree, browse to the desired location using the nodes corresponding to the folders and select the required folder. After selecting the folder, the **Folder Content** window will be displayed. Close this window and the selected folder will become your current working directory.

3. Choose the **Organize** button from the **Select Working Directory** dialog box or right-click in this dialog box to display a shortcut menu. From the shortcut menu, choose the **New Folder** option; the **New Folder** dialog box is displayed.

4. Enter **c03** in the **New Directory** edit box of the **New Folder** dialog box and then choose **OK**; a folder with the name *c03* is created at *C:\Creo-3.0*.

5. Choose **OK** from the **Select Working Directory** dialog box; *C:\Creo-3.0\c03* is set as the working directory.

Starting a New Object File

1. Choose the **New** button from the **Data** group; the **New** dialog box is displayed. Select the **Sketch** radio button from the **Type** area of the **New** dialog box; the default name of the sketch appears in the **Name** edit box.

2. Enter *c03tut1* in the **Name** edit box and choose the **OK** button.

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

Navigator by clicking on the sash present on its right edge. Now, the drawing area is increased.

Importing the Section

1. Choose **Get Data > File System** from the **Ribbon**; the **Open** dialog box is displayed with the working directory as the current directory.

2. Click on the black arrow beside the **Creo-3.0** option in the address bar and choose **c02** from the flyout displayed. Make sure the **Sketch (*.sec)** option is selected in the **Type** drop-down list. Select *c02tut3.sec* and choose the **Open** button from the **Open** dialog box.

3. Move the cursor in the drawing area. Notice that the cursor is attached with a plus mark. Now, click anywhere in the drawing area to place the sketch. The sketch is displayed in the drawing area and the **Import Section** tab is displayed in the **Ribbon**.

4. Enter **1** as a scale factor in the respective edit box and choose the **Done** button to complete importing the sketch.

5. Choose the **Refit** button from the **Graphics** toolbar. The sketch, similar to the one shown in Figure 3-22, is displayed in the drawing area.

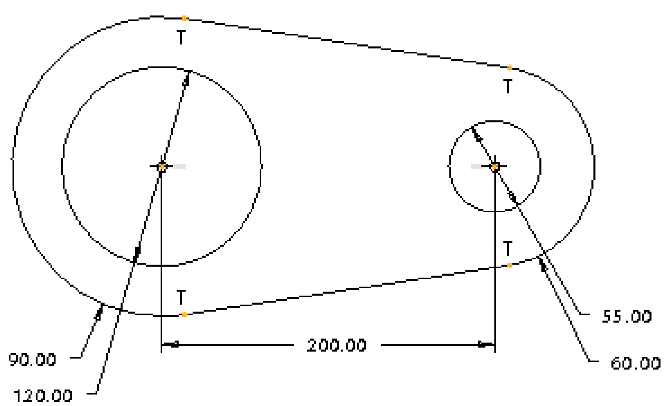


Figure 3-22 *Sketch imported and placed in the current file*

Drawing the Keyway

To create the keyway, you need to sketch a small rectangle and then remove the portion of the circle that lies between the rectangle.

1. Choose the **Line Chain** tool from **Line** drop-down in the **Sketching** group.

2. Draw the keyway, as shown in Figure 3-23; the weak dimensions and constraints are automatically applied to the sketch of the keyway.

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

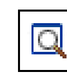
3. Choose the **Zoom In** button from the **Orientation** group of the **View** tab; the cursor is converted into a magnifying glass symbol.

4. Draw a window around the keyway to zoom in it. Now, the display of the keyway is enlarged.

5. Choose the **Delete Segment** tool from the **Editing** group.

6. Click to select the part of the circle that lies between the two horizontal lines; the selected part is deleted.

7. Choose the **Refit** button from the **Graphics** toolbar to view the full sketch.



Dimensioning the Keyway

Now, you need to apply dimensions to the keyway.

1. Choose the **Normal** tool from the **Dimension** group.

2. Dimension the keyway, as shown in Figure 3-24.

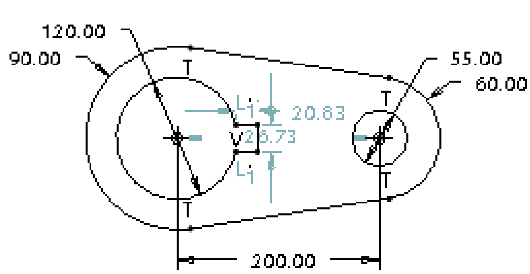


Figure 3-23 Sketch of the keyway with weak dimensions and constraints

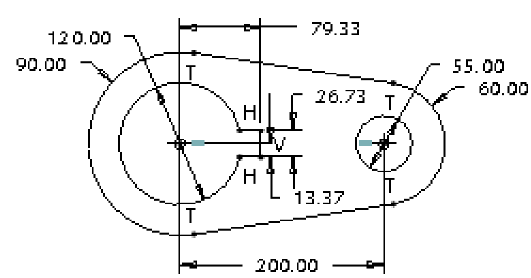


Figure 3-24 Sketch after dimensioning the keyway

Modifying the Dimensions

The dimensions of the keyway need to be modified as per the given dimension values.

1. Select the three dimensions of the keyway by pressing CTRL+left mouse button.

2. Choose the **Modify** tool from the **Editing** group; the **Modify Dimensions** dialog box is displayed.

3. Clear the **Regenerate** check box and then modify the dimensions of the keyway. 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 gets enclosed in a blue box in the sketch.

4. Modify all dimensions. Refer to Figure 3-21 for dimension values.

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

   The sketch after modifying the dimension values of the sketch is shown in Figure 3-25.

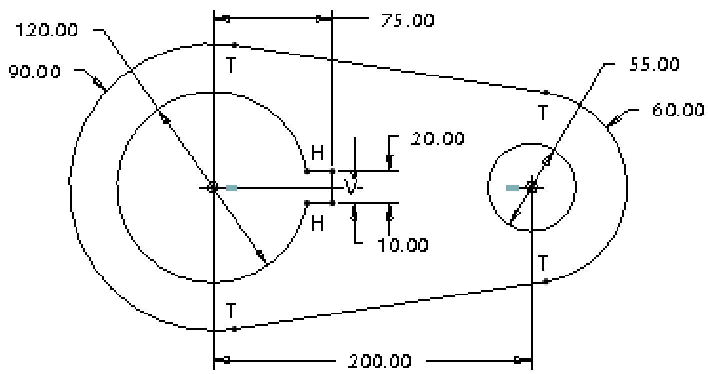


***Figure 3-25** Sketch after modifying the dimensions*

Saving the Sketch

As you may need the sketch later, you must save it.

1. Choose the **Save** button from the **File** menu; the **Save Object** dialog box is displayed with the name of the sketch entered earlier.

2. Choose the **OK** button; the sketch is saved.

3. After saving the sketch, choose the **Close** button from the **Quick Access** toolbar to exit the **Sketch** mode.

Tutorial 2

In this tutorial, you will draw the sketch for the model shown in Figure 3-26. The sketch is shown in Figure 3-27. **(Expected time: 30 min)**

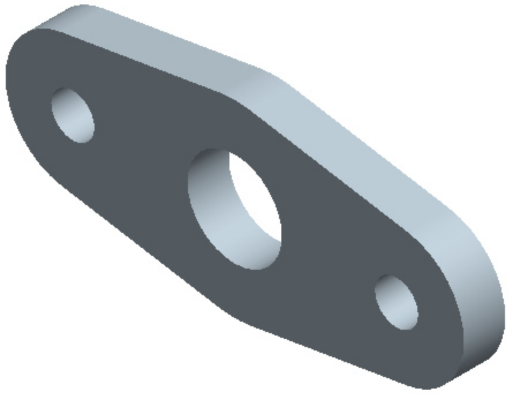


Figure 3-26 Model for Tutorial 2

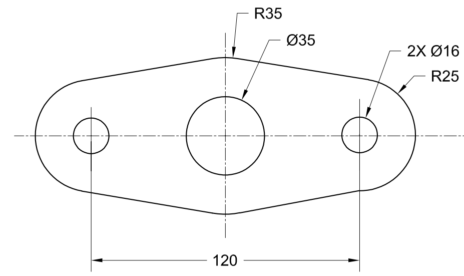


Figure 3-27 Sketch of the model

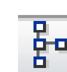
The following steps are required to complete this tutorial:

a. Set the working directory and create a new object file.
b. Draw the sketch using sketcher tools, refer to Figures 3-28 and 3-29.
c. Apply the required constraints and dimensions to the sketched entities, refer to Figure 3-32.
d. Modify the dimensions of the sketch, refer to Figure 3-33.
e. Save the sketch and exit the **Sketch** mode.

Setting the Working Directory

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

1. Open the Navigator by choosing the **Show Navigator** button on the bottom left corner of the PTC Creo Parametric screen; the Navigator slides out. In the Navigator, the Folder Tree is displayed at the bottom. Click on the black arrow, which is available on the right of the Folder Tree; the Folder Tree expands.



2. Click on the node adjacent to the *Creo-3.0* folder in the Navigator; the contents of the *Creo-3.0* folder are displayed.

3. Now, right-click on the *c03* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; *c03* is set as the working directory.

4. Close the Navigator by clicking the sash on the right edge of the Navigator; the Navigator slides in.

Starting a New Object File

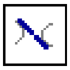
1. Start a new object file in the **Sketch** mode. Name the file as *c03tut2*.

Drawing the Sketch

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

1. Choose the **Center and Point** tool from the **Circle** drop-down. Draw the circles in a horizontal line, as shown in Figure 3-28.

2. Choose the **Line Tangent** tool from the **Line** drop-down in the **Sketching** group.



3. Select the left and middle circles on their respective top points; a tangent is drawn from the top of the left circle to the top of the middle circle.

4. Next, select the right and middle circles on their respective top points; a tangent is drawn from the top of the right circle to the top of the middle circle.

5. Similarly, using the **Line Tangent** tool, draw the other tangents through the bottom-most points of the left, middle, and right circles, as shown in Figure 3-29.

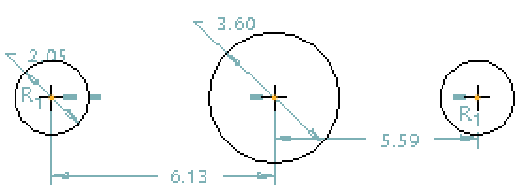


Figure 3-28 *Three circles with weak dimensions and constraints*

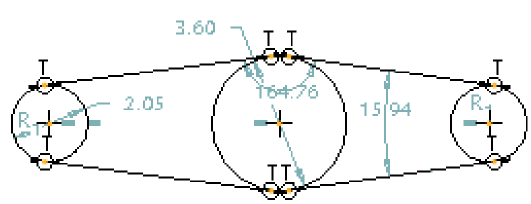


Figure 3-29 *Tangent lines drawn on the circles*

Trimming the Circles

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

1. Choose the **Delete Segment** tool from the **Editing** group.

2. Move the cursor close to the right portion of the left circle; the right part of the circle turns green in color. Next, click on it to delete it.

3. Similarly, trim the parts of the middle and right circles that are not required. The sketch after trimming the circles is shown in Figure 3-30.

Drawing the Circles

1. Choose the **Concentric** tool from the **Circle** drop-down in the **Sketching** group; you are prompted to select an arc.

2. Click on the left arc and move the mouse; a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button.

Tip: *While drawing a concentric circle, sometimes the circle snaps to the other circle or arc and it becomes difficult to draw a circle of the size you need. In such a case, you can disable the snapping of the circle to the other circle or arc. To do so, hold the SHIFT key to disable the snapping or to disable the equal radii constraint that the system tends to apply while drawing the circle.*

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

3. Click on the top arc and move the mouse; a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button to end the creation of circle.

4. Click on the right arc and move the mouse; a circle appears. Select a point inside the sketch to complete the circle. Press the middle mouse button to end the creation of circle.

   The sketch after drawing all three circles is shown in Figure 3-31.

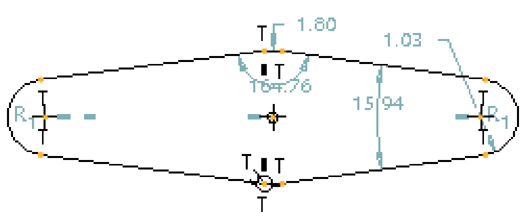


Figure 3-30 *Sketch after trimming the circles*

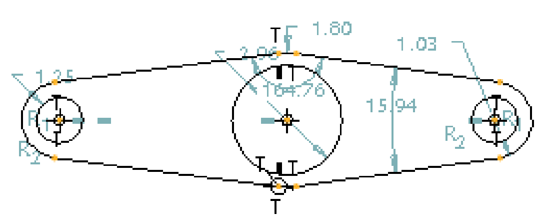


Figure 3-31 *Sketch after drawing all three circles*

Applying the Constraints

1. Choose the **Equal** tool from the **Constrain** group.

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

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

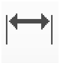
4. Choose the **Parallel** tool from the **Constrain** group.

5. Click to select the tangent line that connects the left arc and the middle arc at the top and then click to select the tangent line that connects the right arc and the middle arc at the bottom. It is evident from the parallel constraint symbol that the parallel constraint has been applied to the two tangent lines in the sketch.

Dimensioning the Sketch

PTC Creo Parametric applies weak dimensions to the sketch automatically. These dimensions are not the needed dimensions because these dimensions do not help in machining the model. Therefore, you need to dimension the sketch with the dimensions that will be used to machine the model. To do so, follow the steps given next.

1. Choose the **Normal** tool from the **Dimension** group.



2. Select the center of the right and left circles; the centers of the circles turn black in color. Now, using the middle mouse button, place the dimension below the sketch. The sketch after applying constraints is shown in Figure 3-32.

   The rest of the weak dimensions are the needed dimensions.

Modifying the Dimensions

All constraints and dimensions have been applied to the sketch. Now, you need to modify the dimensions.

1. Select all dimensions using the CTRL+ALT+A keys.
2. Choose the **Modify** tool from the **Editing** group; the **Modify Dimensions** dialog box is displayed.

3. Clear the **Regenerate** check box and then modify the values of the dimensions, refer to Figure 3-27 for dimension values.

   When you clear the check box, the sketch does not regenerate while modifying the dimensions. The dimension that you edit in the **Modify Dimensions** dialog box is enclosed in a blue box in the sketch.

4. After the dimensions are modified, choose the **OK (Regenerate the section and close the dialog)** button from the **Modify Dimensions** dialog box; the message **Dimension modifications successfully completed** is displayed in the message area.

   The sketch after modifying the dimension values is shown in Figure 3-33.

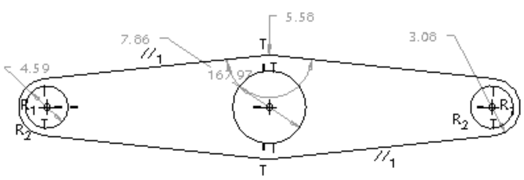


Figure 3-32 Sketch with constraints

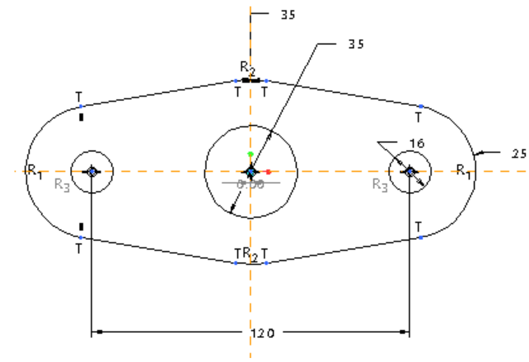


Figure 3-33 Sketch after modifying the dimensions

Saving the Sketch

1. Choose the **Save** button from the **File** menu and save the sketch.

Exiting the Sketch Mode

1. Choose the **Close** button from the **Quick Access** toolbar to exit the Sketch.

Tutorial 3

In this tutorial, you will draw the sketch of the model shown in Figure 3-34. The sketch is shown in Figure 3-35. **(Expected time: 30 min)**

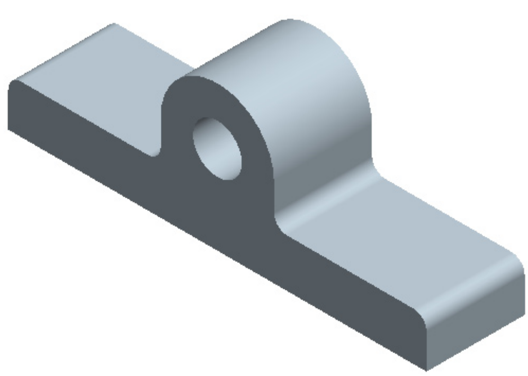


Figure 3-34 Model for Tutorial 3

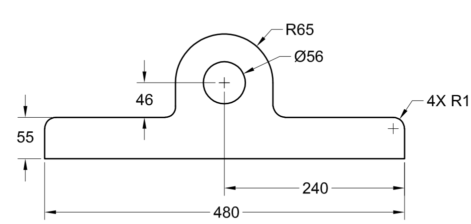


Figure 3-35 Sketch of the model

The following steps are required to complete this tutorial:

a. Set the working directory and create a new object file.
b. Draw the sketch using sketcher tools, refer to Figures 3-36 and 3-37.
c. Apply fillets at two corners of the sketch, refer to Figures 3-38 and 3-39.

d. Dimension the sketch, refer to Figure 3-40.
e. Modify dimensions of the sketch, refer to Figure 3-41.
f. Save the sketch and exit the **Sketch** mode.

Setting the Working Directory

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

1. Open the Navigator by clicking on the sash in the left edge of the PTC Creo Parametric window; the Navigator slides out. In the Navigator, the Folder Tree is displayed at the bottom. Click on the black arrow that is available at the right-side of the Folder Tree; to expand it.

2. Click on the node adjacent to the *Creo-3.0* folder in the Navigator to display the content of this folder.

3. Now, right-click on the *c03* folder to display a shortcut menu. From this shortcut menu, choose the **Set Working Directory** option; *c03* is set as the working directory.

4. Close the Navigator by clicking on the sash on the right edge of the Navigator; the Navigator slides in.

Starting a New Object File

1. Start a new object file in the **Sketch** mode. Name the file as *c03tut3*.

Drawing the Sketch

1. Choose the **Line Chain** tool from the **Line** drop-down of the **Sketching** group.

2. Draw the lines with constraints, as shown in Figure 3-36.

3. Choose the **3-Point / Tangent End** tool from the **Arc** drop-down available in the **Sketching** group.

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

5. Choose the **Concentric** tool from the **Circle** drop-down available in the **Sketching** group; you are prompted to select an arc.

6. Click on the arc; a rubber-band circle appears. Size the circle by moving the cursor and click to complete it. The sketch after drawing the circle is shown in Figure 3-37.

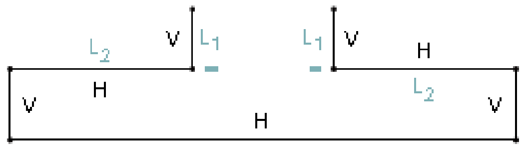


Figure 3-36 Lines in the sketch with the dimensions turned off for clarity

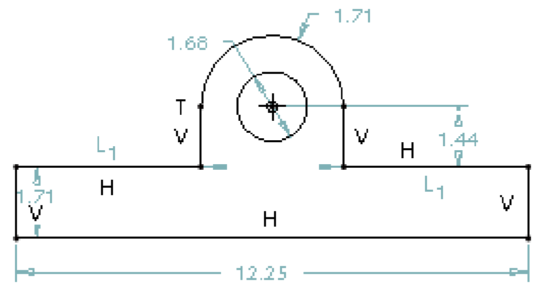


Figure 3-37 Sketch after drawing the arc and the circle

Note

*1. Choose the **Disp Dims** button from the **Display** group in the **View** tab to turn the dimensions on or off.*

2. PTC Creo Parametric does not have the options like midpoint, endpoint, or center of an arc or a circle. However, while drawing a sketch, these options are applied in the form of weak constraints. For example, while drawing an entity, the endpoint of the entity snaps to the cursor. The middle point constraint appears when you bring the cursor near to the middle point of the line to draw another line.

Filleting the Corners

1. Choose the **Circular Trim** option from the **Fillet** drop-down in the **Sketching** group; you are prompted to select the two entities to be filleted. The corners that you need to fillet are shown in Figure 3-38.

2. Select the two entities one by one using the left mouse button to fillet the corners of these entities.

   The sketch after creating the fillets is shown in Figure 3-39.

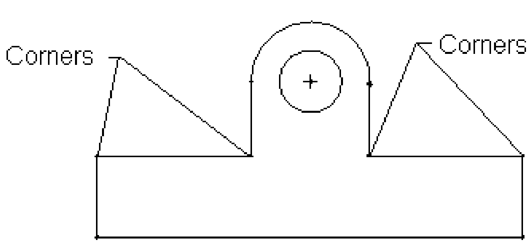


Figure 3-38 Corners to be filleted

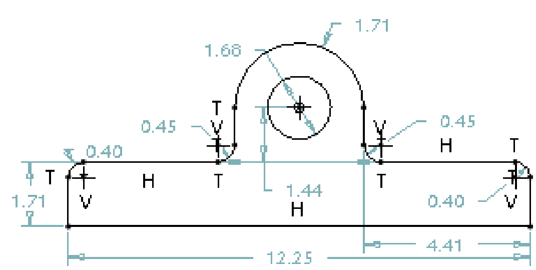


Figure 3-39 Sketch after creating fillets

Applying the Constraints

1. Choose the **Equal** tool from the **Constrain** group.

2. Click to select the fillets and apply the equal constraint to all fillets.

Dimensioning the Sketch

The weak dimensions are applied to the sketch automatically. These are not the required dimensions and therefore, you need to dimension the sketch manually.

1. Choose the **Normal** tool from the **Dimension** group.

2. Dimension the sketch, as shown in Figure 3-40.

Modifying the Dimensions

You need to modify the dimension values that are assigned to the sketch.

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

2. Choose the **Modify** tool from the **Editing** group; the **Modify Dimensions** dialog box is displayed.

3. Clear the **Regenerate** check box and then modify the values of the dimensions. If this check box is cleared, the sketch does not regenerate while modifying the dimensions.

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

4. Modify all dimensions. Refer to Figure 3-35 for dimension values.

5. Choose the **OK (Regenerate the section and close the dialog)** button from the **Modify Dimensions)** dialog box.

   The sketch after modifying the dimension values is shown in Figure 3-41.

Saving the Sketch

1. Choose the **Save** button from the **File** menu and save the sketch.

Exiting the Sketch Mode

1. Choose the **Close** button from the **Quick Access** toolbar to exit the **Sketch** mode.

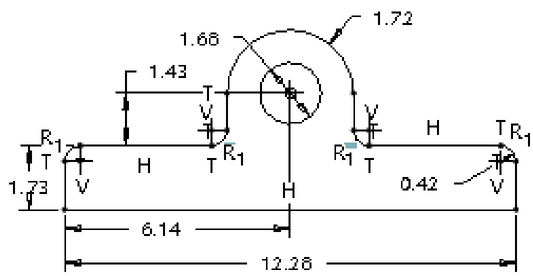


Figure 3-40 Sketch after dimensioning

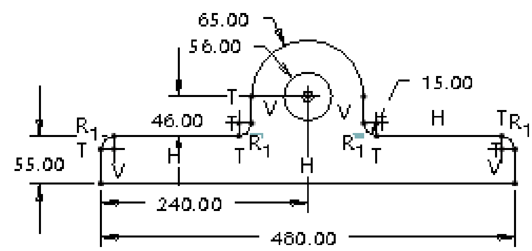


Figure 3-41 Sketch after modifying the dimensions

Self-Evaluation Test

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

1. The __________ dialog box is used to modify dimensions.

2. The display of dimensions and constraints can be turned on/off by choosing the __________ and __________ buttons respectively from the **Display** group.

3. The __________ tool is used to rotate selected entities.

4. You can delete entities by selecting them and then using the __________ key on the keyboard.

5. The shape of a spline can be modified dynamically. (T/F)

6. You can increase the length of the spline by adding points before the start point and after the endpoint of the spline. (T/F)

7. While copying the sketched entities, the **Paste** tab is also displayed. (T/F)

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

9. You can dimension the length of a centerline. (T/F)

10. The font of the text written in the sketcher environment cannot be modified. (T/F)

Review Questions

Answer the following questions:

1. How many handles are displayed in the graphics window while rotating and resizing entities?

   (a) one (b) two
   (c) three (d) four

2. Which of the following mouse buttons is used to place the dimension?

   (a) left (b) middle
   (c) right (d) mouse is not used for dimensioning

3. Which of the following is the default font for the text in the sketcher environment?

   (a) font (b) filled
   (c) font3d (d) isofont

4. Which of the following groups is used to toggle the display of dimensions and constraints in the sketcher environment?

   (a) **Sketching** (b) **Constrain**
   (c) **Display** (d) **Dimension**

5. In which type of dimensioning, the **Dim Orientation** dialog box is displayed while dimensioning the arcs and circles?

   (a) **Normal** (b) **Perimeter**
   (c) **Baseline** (d) None of the above

6. For placing a section in a new sketch, you can use the right mouse button. (T/F)

7. You can create elliptical fillets in PTC Creo Parametric. (T/F)

8. While creating text in the sketcher environment, you need to draw a construction line that will define the height of the text. (T/F)

9. You can modify the dimensions dynamically. (T/F)

10. You can modify a spline by moving its interpolation points. (T/F)

Exercises

Exercise 1

In this exercise, you will draw the sketch of the model shown in Figure 3-42. The sketch to be drawn is shown in Figure 3-43. **(Expected time: 30 min)**

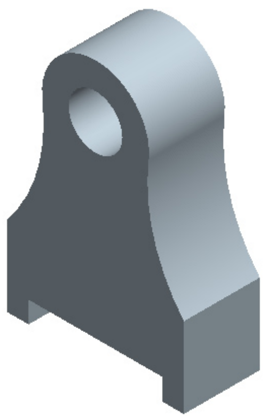


Figure 3-42 *Solid model for Exercise 1*

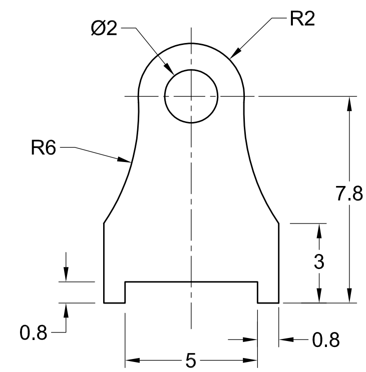


Figure 3-43 *Sketch of the model*

Exercise 2

In this exercise, you will draw the sketch of the model shown in Figure 3-44. The sketch to be drawn is shown in Figure 3-45. **(Expected time: 15 min)**

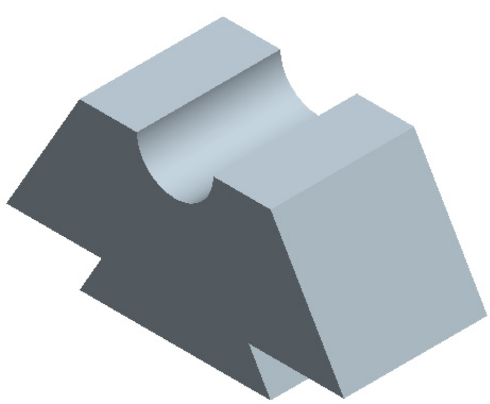


Figure 3-44 *Solid model for Exercise 2*

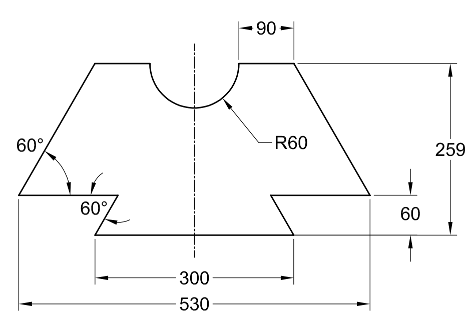


Figure 3-45 *Sketch of the model*

Exercise 3

In this exercise, you will draw the sketch of the model shown in Figure 3-46. The sketch to be drawn is shown in Figure 3-47. **(Expected time: 30 min)**

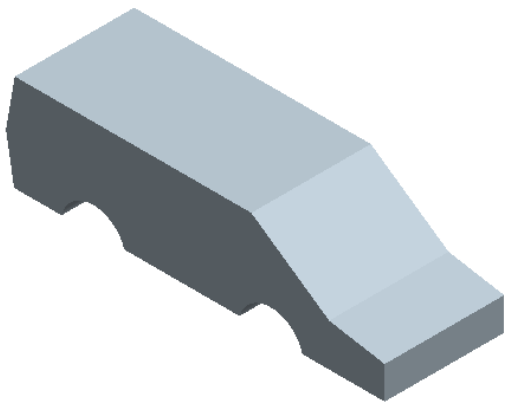


Figure 3-46 Solid model for Exercise 3

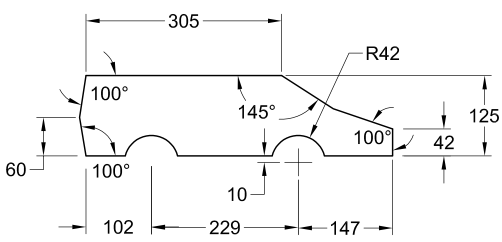


Figure 3-47 Sketch of the model

Answers to Self-Evaluation Test

1. Modify Dimensions, **2. Disp Dims**, **Disp Constr**, **3. Rotate Resize**, **4.** DELETE, **5.** T, **6.** F, **7.** T, **8.** T, **9.** F, **10.** F