Chapter *3*

Drawing Sketches in the Sketcher Workbench-II

Learning Objectives

After completing this chapter, you will be able to:

• *Draw ellipses.*
• *Draw splines.*
• *Connect two elements using an arc or a spline.*
• *Draw elongated holes.*
• *Draw cylindrical elongated holes.*
• *Draw key holes.*
• *Draw hexagons.*
• *Draw centered rectangles.*
• *Draw centered parallelograms.*
• *Draw different type of conics.*
• *Edit and modify sketches.*

OTHER SKETCHING TOOLS

You have learned about some of the sketching tools in the last chapter. In this chapter, you will learn about the remaining sketching tools in the **Sketcher** workbench.

Drawing Ellipses

Menu:	Insert > Profile > Conic > Ellipse
Toolbar:	Profile > Conic > Ellipse

To draw an ellipse, invoke the **Ellipse** tool by choosing the **Ellipse** button from the **Conic** toolbar, as shown in Figure 3-1.
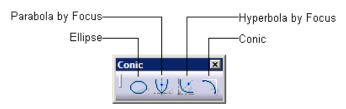

*Figure 3-1 The **Conic** toolbar*

When you invoke the **Ellipse** tool, the **Sketch tools** toolbar expands and you are prompted to specify the ellipse center. Click in the geometry area to specify it. You will be prompted to define the major axis and the orientation of the ellipse. In CATIA V5, the first axis of an ellipse is the major axis. To define it, you need to specify a point on the ellipse. The orientation of the ellipse is the angle formed by the major axis with the horizontal reference. Move the cursor away from the center point; the preview of the ellipse is also displayed. Click in the geometry area to define the major axis. You are then prompted to specify a point on the ellipse, which will determine the other axis. Figure 3-2 shows a point being specified on the preview of the ellipse. You will notice a few construction elements displayed on it. These
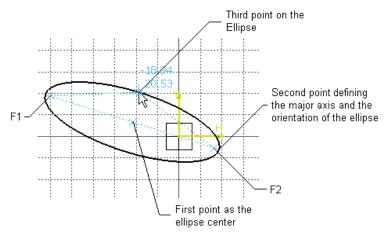

Figure 3-2 Specifying three points to draw an ellipse

elements define its major axis and orientation. Click in the geometry area to specify the third point on the ellipse. An ellipse, based on the specified parameters, is displayed in the geometry area, as shown in Figure 3-3.
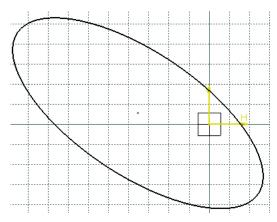

Figure 3-3 *The resulting ellipse*

Note
In CATIA V5, the first axis of an ellipse is termed as the major axis even if its length is less than the minor axis.

Drawing Splines

Menu:	Insert > Profile > Spline > Spline
Toolbar:	Profile> Spline > Spline

To draw a spline, choose the down arrow on the right of the **Spline** button in the **Profile** toolbar and invoke the **Spline** toolbar, as shown in Figure 3-4.

Figure 3-4 *The **Spline** toolbar*

Invoke the **Spline** tool from the **Spline** toolbar; you will be prompted to specify its first control point. Click to specify the first point; you are prompted to specify the next point of the spline or double-click to end the spline definition. Move the cursor to specify the second control point; the preview of the spline is displayed. Click to specify the second control point. Similarly, you can specify multiple points to draw a spline. Figure 3-5 shows a spline being drawn by specifying multiple points.

Note
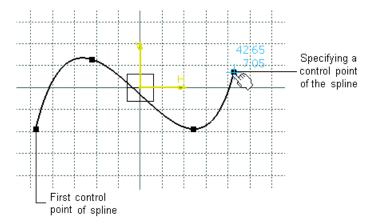*The control points on the spline are the construction elements, while the curve is a standard element.*

Figure 3-5 Specifying points for drawing a spline

Connecting Two Elements by a Spline or an Arc

Menu:	Insert > Profile > Spline > Connect
Toolbar:	Profile > Spline > Connect

Two elements such as lines, arcs, ellipses, circles, or splines can be connected together by an arc or a spline. To do so, invoke the **Connect** tool from the **Spline** toolbar; the **Sketch tools** toolbar expands, as shown in Figure 3-6.
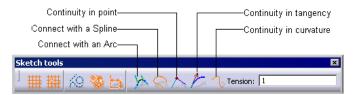

*Figure 3-6 The **Sketch tools** toolbar after invoking the **Connect** tool*

The next section discusses how the two selected elements are connected by a spline and an arc.

Connecting Two Elements with a Spline

By default, the **Connect with a Spline** button is chosen in the **Sketch tools** toolbar. Also, you are prompted to select the first element to be connected. After selecting the first element, you are prompted to select the second element. On doing so, the spline connecting the two elements is displayed in the geometry area.

While connecting the elements, you will notice that the **Continuity in curvature** button is chosen by default in the **Sketch tools** toolbar, Hence, the resulting spline maintains a continuity with the curvature. You can also set the tension value in the **Tension** edit box.

If you choose the **Continuity in tangency** button from the **Sketch tools** toolbar, the resulting spline will maintain a continuity with the tangency. You can also set the value of the tension in the **Tension** edit box.

If you choose the **Continuity in point** button from the **Sketch tools** toolbar, the resulting spline will maintain a continuity with the points of the selected elements. The resulting element will be a straight spline with only two control points at its start and end.

Connecting Two Elements with an Arc

To connect two selected elements with an arc, choose the **Connect** button from the **Spline** toolbar. Next, choose the **Connect with an Arc** button from the **Sketch tools** toolbar; you will be prompted to select the first element to be connected. On doing so, you will be prompted to select the second element. The arc connecting the two elements is displayed in the geometry area once you specify the second element.

Drawing Elongated Holes

Menu:	Insert > Profile > Predefined Profile > Elongated Hole
Toolbar:	Profile > Predefined Profile > Elongated Hole

An elongated hole is a geometry that comprises of two parallel lines and two tangent arcs, as shown in Figure 3-7. To draw an elongated hole, invoke the **Elongated Hole** tool from the **Predefined Profile** toolbar; you will be prompted to specify the center to center distance. This is the distance formed by joining the centers of the two arcs in the elongated hole. Click on the geometry area to specify the first center point; you will be prompted to locate the end point of the distance. Move the cursor away from the first center point; a construction line will be attached to the cursor. Click to specify the endpoint; you will be prompted to define a point on the elongated hole. Move the cursor to specify the point. While moving the cursor, the preview of the elongated hole is displayed in the geometry area. Figure 3-7 shows an elongated hole with the tangent and parallel constraints applied. These constraints will be discussed in later chapters.
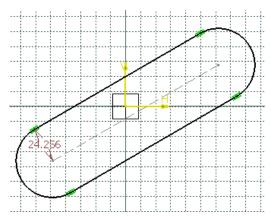

Figure 3-7 An elongated hole profile

Note

*You can enter the parameters required to define the elongated hole in the respective edit boxes of the expanded **Sketch tools** toolbar. The parameters include the coordinate values of the start point and endpoint of the line, angle value formed between the line and the horizontal reference, radius of the elongated hole, or the coordinate value of the point on the elongated hole.*

Drawing Cylindrical Elongated Holes

Menu:	Insert > Profile > Predefined Profile > Cylindrical Elongated Hole
Toolbar:	Profile > Predefined Profile > Cylindrical Elongated Hole

A cylindrical elongated hole is a geometry comprising of four arcs. Each arc is tangent to its adjacent arcs, as shown in Figure 3-8. To draw a cylindrical elongated hole, invoke the **Cylindrical Elongated Hole** tool from the **Predefined Profile** toolbar. On doing so, the **Sketch tools** toolbar expands and you will be prompted to specify the center to center arc. Click in the geometry area to specify the center point. You will be prompted to specify the radius and the start point of the arc. Move the cursor away from the center point; a dotted circle is attached to the cursor. Click to specify the start point. You will then be prompted to move the cursor and specify the end point of the arc. Move the cursor away from the start point; a dotted arc is attached to the cursor. Click in the geometry area to specify its endpoint; you will be prompted to specify a point on the cylindrical elongated hole. Move the cursor away from the third point to specify a point; the preview of the cylindrical elongated hole is displayed. Click on it to specify a point. The cylindrical elongated hole is created, as shown in Figure 3-8.
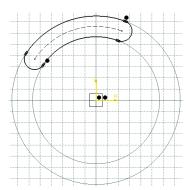

Figure 3-8 A cylindrical elongated hole profile

To draw a cylindrical elongated hole, you can also enter its parameters in the respective edit boxes of the **Sketch tools** toolbar.

Note

You will observe that sometimes while moving the cursor to specify a point on the geometry or define its shape and size, a ⊜ sign is displayed above the cursor. This sign suggests that you cannot specify a point for the element at the current location of the cursor.

Drawing Keyhole Profiles

Menu:	Insert > Profile > Predefined Profile > Keyhole Profile
Toolbar:	Profile > Predefined Profile > Keyhole Profile

A keyhole profile is a keyhole shaped geometry that comprises of two arcs and two lines, as shown in Figure 3-9. To draw a keyhole profile, invoke the **Keyhole Profile** tool from the **Predefined Profile** toolbar. The **Sketch tools** toolbar expands and you

are prompted to specify the start point. Click in the geometry area to specify the center point. You are then prompted to define the center point of the small radius arc. Move the cursor away from the first center point to specify the second center point; a dashed line will be displayed with the cursor, which defines the length of the keyhole profile. Click to specify the center of the small radius. You are then prompted to specify a point on the keyhole profile to define the radius of the small arc. Move the cursor away from the center point of the small arc to preview the keyhole profile. Click on the preview to define the smaller radius. You are then prompted to specify a point on the keyhole profile to define the radius of the larger arc. Click on the preview of the keyhole to specify it. The final keyhole profile, with the specified values, will be displayed in the geometry area, as shown in Figure 3-9. You can also specify the required parameters in the **Sketch tools** toolbar to draw a keyhole profile.
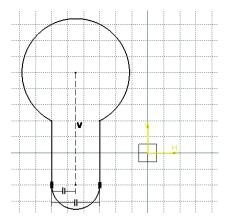

Figure 3-9 A keyhole profile

Drawing Hexagons

Menu:	Insert > Profile > Predefined Profile > Hexagon
Toolbar:	Profile > Predefined Profile > Hexagon

To draw a hexagon, choose the **Hexagon** button from the **Predefined Profile** toolbar; you will be prompted to select the hexagon center. Specify a point in the geometry area to define it. Now, move the cursor away from the hexagon center; the preview of the hexagon will be displayed. Specify a point on the hexagon to complete its creation. The resulting hexagon is shown in Figure 3-10.

Drawing Centered Rectangles

Menu:	Insert > Profile > Predefined Profile > Centered Rectangle
Toolbar:	Profile > Predefined Profile > Centered Rectangle

CATIA V5 also provides you with the tool to draw a rectangle that is centered about a point, which is specified while drawing the rectangle. To draw a centered rectangle, choose the **Centered Rectangle** button from the **Predefined Profile** toolbar; you will be prompted to select a point to create the center of the rectangle. Specify a point in the
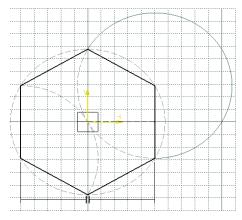
Figure 3-10 *Hexagon drawn using the **Hexagon** tool*

geometry area and move the cursor; the preview of the rectangle is displayed. Specify a point on any one corner of the rectangle. Figure 3-11 shows a centered rectangle, along with its center point and the point at its corner.
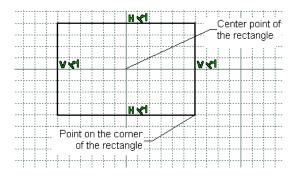

Figure 3-11 *Center rectangle along with the center point and the point at the corner of the rectangle*

Drawing Centered Parallelograms

Menu:	Insert > Profile > Predefined Profile > Centered Parallelogram
Toolbar:	Profile > Predefined Profile > Centered Parallelogram

CATIA V5 also allows you to draw a centered parallelogram. Note that to draw such a parallelogram, you need to select two lines. The opposite sides of the parallelogram will be parallel to these two lines. To create this type of parallelogram, choose the **Centered Parallelogram** button from the **Predefined Profile** toolbar; you will be prompted to select the first line. Select the first line to which one set of sides of the parallelogram will be parallel. Next, select the second line. The parallelogram will be created with its center at the intersection point of the selected lines. Also, the second set of the opposite sides of the parallelogram will be parallel to the second selected line. Next, you will be prompted to

select the end point to create a centered parallelogram. Move the cursor and specify a point on any one of the corners of the parallelogram. Figure 3-12 shows the centered parallelogram with the first and second reference lines and the point on the parallelogram.
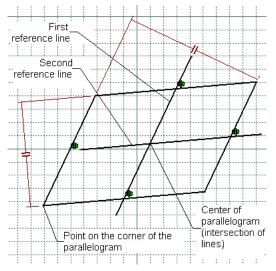

Figure 3-12 *Centered parallelogram with the first line, second line, and the point on the parallelogram*

Note

*After drawing the centered parallelogram, select the reference lines and convert them into a construction element using the **Construction/Standard Element** button in the **Sketch tools** toolbar.*

Drawing Conics

Conics are the geometrical elements that are formed by the intersection of a plane and a cone. By changing the angle and location of the intersection, you can produce an ellipse, parabola, or hyperbola. To draw various conics in CATIA V5R18, choose the down arrow available on the right of the **Ellipse** button in the **Profile** toolbar. The tools in the **Conic** toolbar to create conics are discussed next.

Drawing a Parabola by Focus

Menu:	Insert > Profile > Conic > Parabola by Focus
Toolbar:	Profile > Conic > Parabola by Focus

To draw a parabola by focus, invoke the **Parabola by Focus** tool from the **Conic** toolbar; the **Sketch tools** toolbar expands and you are prompted to specify the focus. Click in the geometry area to specify it. You are then prompted to specify the apex. Move the cursor away from the focus; the preview of the parabola, attached to the cursor, is displayed. Click to specify it. You are then prompted to specify the start point. Move the cursor away from the apex and specify the start point. You are then prompted to specify the end point. Move the cursor

along the path of the parabola and click to specify its end point. Figure 3-13 shows the points used to draw the parabola and the resulting parabola.
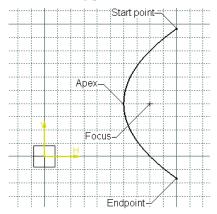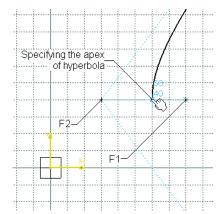

Figure 3-13 *Points used to draw a parabola*

Drawing a Hyperbola by Focus

Menu:	Insert > Profile > Conic > Hyperbola by Focus
Toolbar:	Profile > Conic > Hyperbola by Focus

To draw a hyperbola by focus, invoke the **Hyperbola by Focus** tool from the **Conic** toolbar; the **Sketch tools** toolbar expands and you are prompted to specify the focus. Click to specify the focus, which is referred to as F1 in Figure 3-14. Next, you are prompted to specify the center. Move the cursor away from the focus. As you move the cursor, you will notice that the preview of the hyperbola is attached to the cursor. Click to specify its center, which is referred to as F2 in Figure 3-14. You are then prompted to specify the apex of the hyperbola. Move the cursor away from the center point to specify it. You will notice that the preview of the hyperbola moves along with the cursor. Also, in the **Sketch tools** toolbar, the value of eccentricity in the edit box changes. Eccentricity, in case of a hyperbola, is defined as the ratio of the distance of the apex from the center point to the distance of the center point from the focus point.

Click to specify the apex. You are then prompted to specify the start point of the hyperbola. Move the cursor away from the apex and specify the start point, as shown in Figure 3-15. You can move the cursor in either direction to specify the start point. You are then prompted to specify the endpoint. Move the cursor in the opposite direction of the start point; the preview of the hyperbola follows the cursor. Click to specify the endpoint.

Note
In case the focus, center point, or both of a parabola/hyperbola do not lie on any of the axes or any sketched element, they will not be displayed as construction points after the parabola/hyperbola is drawn.
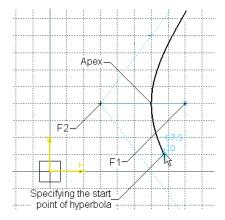
Figure 3-14 *Specifying the apex of the hyperbola*

Figure 3-15 *Specifying the start point of the hyperbola*

Drawing Conics

Menu:	Insert > Profile > Conic > Conic
Toolbar:	Profile > Conic > Conic

To draw a conic, invoke the **Conic** tool from the **Conic** toolbar; you are prompted to locate the first endpoint. After you have done so, you are prompted to locate the tangent at the first endpoint. Move the cursor away from the endpoint to define the tangent. Similarly, define the second point and the tangent at that point. Finally, define a point on the conic to create it. Figure 3-16 shows a conic.
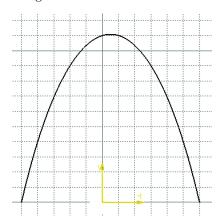

Figure 3-16 *A conic*

EDITING AND MODIFYING SKETCHES

In this section of the chapter, you will learn about the editing and modification tools used in the **Sketcher** workbench. These include tools for trimming the sketches using the quick trim, breaking a sketched element, filleting the sketches, adding chamfer to the sketches, and so on. These tools are discussed next.

Trimming Unwanted Sketched Elements

Menu:	Insert > Operation > Relimitations > Trim
Toolbar:	Operation > Relimitations > Trim

In the **Sketcher** workbench, you are provided with the **Trim** tool to remove the unwanted intersected portion of a sketched element. Invoke the **Relimitations** toolbar by choosing the down arrow provided on the right of the **Trim** button in the **Operation** toolbar. The **Relimitations** toolbar is shown in Figure 3-17.
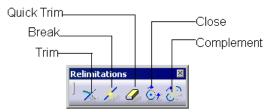
*Figure 3-17 The **Relimitations** toolbar*

Choose the **Trim** button from the **Relimitations** toolbar; the **Sketch tools** toolbar expands and you are prompted to select a point or a curve type element. By default, the **Trim All Elements** button is chosen in the expanded **Sketch tools** toolbar. Select the side of the first element that you need to retain. Next, select the second element that will act as the cutting edge to trim the first element. Figure 3-18 shows the elements selected to be trimmed and Figure 3-19 shows the resulting trimmed elements.
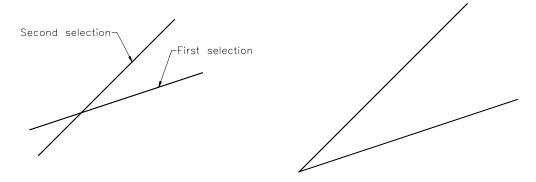
Figure 3-18 Elements to be selected for trimming   *Figure 3-19 The resulting trimmed elements*

After invoking the **Trim** tool, if you choose the **Trim First Element** button from the **Sketch tools** toolbar, then only the first element will be trimmed. Figure 3-20 shows the elements selected to be trimmed and Figure 3-21 shows the resulting trimmed elements.
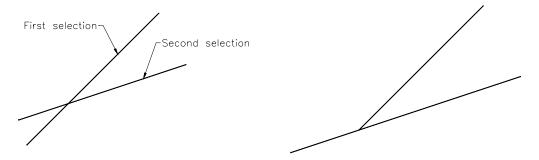
Figure 3-20 *Elements to be selected for trimming*   ***Figure 3-21*** *The resulting trimmed element*

Extending Sketched Elements

Menu:	Insert > Operation > Relimitations > Trim
Toolbar:	Operation > Relimitations > Trim

In CATIA, you can also extend the sketched elements by using the **Trim** tool. To do so, invoke this tool; you are prompted to select a point or a curve type element. Select the sketched element to be extended and then select the destination up to which you need to extend it. You can also click anywhere in the drawing window to dynamically extend the selected element. If you are using the **Trim** tool to extend the elements, it is recommended to choose the **Trim First Element** button from the **Sketch tools** toolbar. This is because if the destination to extension is another element, then the other portion of the element will be deleted. Figure 3-22 shows the element selected to be extended and also the destination element. Figure 3-23 shows the resulting extended element.
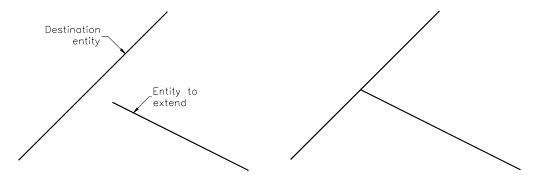

Figure 3-22 *Element selected to be extended*   ***Figure 3-23*** *The resulting extended element*

Trimming by Using the Quick Trim Tool

Menu:	Insert > Operation > Relimitations > Quick Trim
Toolbar:	Operation > Relimitations > Quick Trim

The **Quick Trim** tool is used to quickly trim the unwanted sketched elements. To invoke this tool, choose the **Quick Trim** button from the **Relimitations** toolbar; the

Sketch tools toolbar expands and you are prompted to select a curve type element. Select the portion of the sketch that you need to remove. You can also remove the non-intersecting sketched elements using the **Quick Trim** tool. As a result, this tool also works as the **Delete** tool on the entities that are not intersected by any other entity.

Tip. *When you choose the **Quick Trim** tool, the **Sketched tools** toolbar expands and three additional buttons are displayed. By default, the **Break And Rubber In** button is chosen. If you keep this button chosen and select the element to be trimmed, then it will break the selected element along the intersections and remove the selected portion of the selected element. If you choose the **Break and Rubber Out** button from the **Sketch tools** toolbar and select a portion of an intersected element to be removed, it will break the selected sketched element, along its intersections with the other elements. The selected portion of the element will be retained, while the other part will be removed.*

*If you select the **Break And Keep** button from the **Sketch tools** toolbar, after invoking the **Quick Trim** tool, then it will only break the selected element along its intersection.*

*You can close an arc or a trimmed circle to form a complete circle using the **Close** button in the **Relimitations** toolbar. Choose the **Close** button from the **Relimitations** toolbar and select the arc or trimmed circle to be closed. You can also close an arc or a trimmed circle by selecting it first and then right-clicking to invoke the shortcut menu. From the shortcut menu, choose "**Name of the Element**" > **Close**. The trimmed circle or arc is closed.*

*You can also convert the complementary side of the trimmed circle or an arc to a standard element and remove its existing portion. Choose the **Complement** button from the **Relimitations** toolbar and select the element. You can also use the shortcut menu to convert the complementary portion into an element, as discussed while closing the elements.*

Filleting Sketched Elements

Menu:	Insert > Operation > Corner
Toolbar:	Operation > Corner

In the **Sketcher** workbench of CATIA V5, you are provided with the **Corner** tool to fillet the sketched elements. When you invoke this tool, the **Sketch tools** toolbar expands and you are prompted to select the first curve or a common point. Select the first element to be filleted. Next, you are prompted to select the second curve. Select it and specify the fillet radius in the **Radius** edit box that is displayed in the **Sketch tools** toolbar. You can also specify the fillet radius by dynamically moving the cursor and then specifying a point on the arc.

Note

*The creation of the fillet depends on the point that is selected to specify the fillet radius in the dynamic fillet creation. You can also fillet two parallel lines using the **Corner** tool.*

The **Sketch tools** toolbar, which expands on invoking the **Corner** tool, displays various options that are used to create a fillet with different types of trimming options. If you choose:

a.   the **Trim All Element** button, both the selected elements are trimmed beyond the fillet region. This button is chosen by default.

b.   the **Trim First Element** button and then fillet the sketched elements, the resulting fillet will be created by trimming only the first element. The second element will be retained.

c.   the **No Trim** button, the resulting fillet will be created by retaining both the selected elements.

d.   the **Standard Lines Trim** button, the resulting fillet will be created by retaining both the selected elements, and the retained elements will remain as standard elements. But if the elements extend beyond the corner selected to be trimmed, the extended portion will be removed.
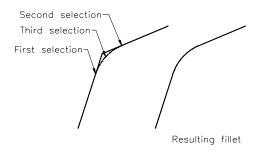
e.   the **Construction Lines Trim** button, the resulting fillet will be created by retaining the selected elements, but the retained elements will be converted to construction elements.

f.   the **Construction Lines No Trim** button, the lines that extend beyond the corner will be retained as the construction elements.

Figure 3-24 shows the elements to be selected and the resulting fillet created using the **Trim All Elements** button. Figure 3-25 shows the fillet created using the **Trim First Element** button and the **No Trim** button.
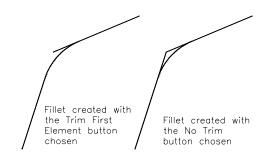

*Figure 3-24 Fillet created using the **Trim All Elements** button*

*Figure 3-25 Fillets created using the **Trim First Element** and **No Trim** buttons*

Chamfering Sketched Elements

Menu:	Insert > Operation > Chamfer
Toolbar:	Operation > Chamfer

The **Sketcher** workbench of CATIA V5 also provides you with a **Chamfer** tool to chamfer the sketched elements. After invoking this tool, the **Sketch tools** toolbar expands and you are prompted to select the first curve or a common point. Select the first element; you are prompted to select the second element. When you select the second element, the **Sketch tools** toolbar expands and you are provided with the **Angle** and the **Length** edit boxes to specify the respective values. Specify the values and press the ENTER key. The chamfer will be created and some dimensions will be applied to it. You can also dynamically specify the parameters of a chamfer. Figure 3-26 shows the elements selected and the resulting chamfer created.
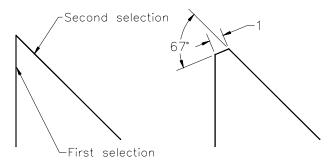

Figure 3-26 Elements to be selected and the resulting chamfer created

After selecting the geometries to be chamfered, the **Sketch tools** toolbar provides you with some options to specify the parameters of the chamfer. If you choose:

a. the **Angle And Hypotenuse** button, you need to specify the angle and length of the hypotenuse in the edit boxes in the **Sketch tools** toolbar. This button is chosen by default.

b. the **First and Second Length** button, then you need to specify the chamfer distances in the **First length** and the **Second length** edit boxes.

c. the **Angle and First Length** button, then you need to specify the length of the chamfer from the first selection and also the angle of the chamfer.

You can specify whether you want to trim or retain the elements using the other buttons in the **Sketch tools** toolbar. These options are the same as discussed while filleting the elements.

Mirroring Sketched Elements

Menu:	Insert > Operation > Transformation > Mirror
Toolbar:	Operation > Transformation > Mirror

You can mirror the sketched elements along the mirror line in the **Sketcher** workbench of CATIA V5 using the **Mirror** tool. Choose the down arrow on the right of the **Mirror** button provided in the **Operation** toolbar to invoke the **Transformation** toolbar, as shown in Figure 3-27. The tools in this toolbar are also known as the transformation tools.
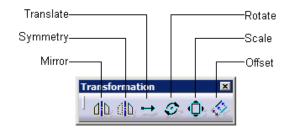
*Figure 3-27 The **Transformation** toolbar*

Select the sketched elements that you need to mirror by dragging a window around them. Alternatively, you can also press and hold the CTRL key and select the elements. Next, choose the **Mirror** button from the **Transformation** toolbar; you are prompted to select the line or axis from which the elements will remain equidistant. Select a line, center line, or the vertical axis of origin as the mirror axis; the selected elements are mirrored about the mirror axis and the symmetry constraints are applied to the sketch on both sides of the mirror axis. Figure 3-28 shows the elements selected to be mirrored and the mirror line to be selected. Figure 3-29 shows the resulting mirrored sketch.
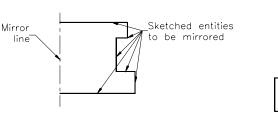

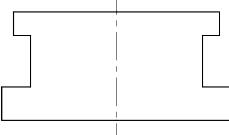*Figure 3-28 Elements selected to be mirrored and the mirror line to be selected*

Figure 3-29 The resulting mirrored sketch

Mirroring Elements Without Duplication

Menu:	Insert > Operation > Transformation > Symmetry
Toolbar:	Operation > Transformation > Symmetry

The **Symmetry** tool mirrors the sketched elements about a mirror axis but deletes the original elements. To mirror the elements without duplication, select the elements by dragging a window around them. Next, choose the **Symmetry** button from the **Transformation** toolbar; you are prompted to select the line or axis from which the elements will remain equidistant. Select the symmetry line; the selected elements will be mirrored on the other side of the symmetry line, while the original elements will be removed.

> **Tip**. *If you select the elements after invoking any of the transformation tools, you need to drag a window to select multiple elements. In such a case, you are not allowed to hold the CTRL key and select multiple elements.*

Translating Sketched Elements

Menu:	Insert > Operation > Transformation > Translate
Toolbar:	Operation > Transformation > Translate

The **Sketcher** workbench provides you the **Translate** tool to move the selected sketched elements from their initial position to the required destination. To move the sketched elements, select them and then choose the **Translate** button from the **Transformation** toolbar; the cursor is replaced by a point cursor and the **Translation Definition** dialog box wil be displayed, as shown in Figure 3-30. Also, you are prompted to select the transition start point. Select a point in the geometry area that will be used as the base point of translation. Set the incremental translation distance in the **Value** spinner in the **Length** area of the **Translation Definition** dialog box and press the ENTER key. The dialog box is not displayed any more. Specify a point in the geometry area to place the selected sketched element. As the **Duplicate mode** check box is selected by default and the value of the instance is set to 1, a copy of the selected element is created at the specified distance. You can also increase the value of the increment using the **Instance(s)** spinner.

*Figure 3-30 The **Translation Definition** dialog box*

The **Keep internal constraints** and **Keep external constraints** check boxes are selected to retain the internal and external constraints, respectively. You will learn more about them in later chapters.

While specifying the start point and destination point of the translation, if you select the **Snap Mode** check box, you will be able to snap to the grid points.

Tip. *You can also specify the translate distance dynamically. To translate an element using this method, select it and invoke the **Translation Definition** dialog box. Next, specify the start point of the translation and then move the cursor to a location where you need to specify its end point.*

Note
*If the **Duplicate mode** check box is cleared, then you can only move the selected elements but cannot copy them. The sketch will only move in the direction, in which its degree of freedom is available. If the sketch is fully constrained, you cannot move the sketched elements using the **Translate** tool. You will learn more about the constraints later in this book.*

Rotating Sketched Elements

Menu:	Insert > Operation > Transformation > Rotate
Toolbar:	Operation > Transformation > Rotate

The **Rotate** tool is used to rotate the sketched elements around a rotation center point. Select the elements by drawing a window around them and then choose the **Rotate** button from the **Transformation** toolbar; the cursor is replaced by the point cursor and the **Rotation Definition** dialog box is displayed, as shown in Figure 3-31.

You are prompted to select the rotation center point. Specify a point around which the selected sketched elements will be rotated. Next, you are prompted to select a point to define a reference line for the angle; hence do so. Next, you are prompted to select a point to define an angle. As you move the cursor to specify the third point, the preview of the rotated selected elements is also displayed. Select a point to specify the rotation angle. As the **Duplicate mode** check box is selected, another copy of the rotated element is created. Figure 3-32 shows the points to be selected and the preview of the rotated instance of the selected elements.
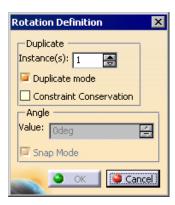
*Figure 3-31 The **Rotation Definition** dialog box*

Scaling Sketched Elements

Menu:	Insert > Operation > Transformation > Scale
Toolbar:	Operation > Transformation > Scale

To scale the sketched elements, select them and then choose the **Scale** button from the **Transformation** toolbar; the **Scale Definition** dialog box is displayed, as shown in Figure 3-33 and you are prompted to select the scaling center point. Select a point in the drawing window. Next, you are prompted to select a point to define the scaling value. You can define the scaling factor dynamically in the geometry area or set its value in the **Value** spinner in the **Scale** area of the **Scale Definition** dialog box.
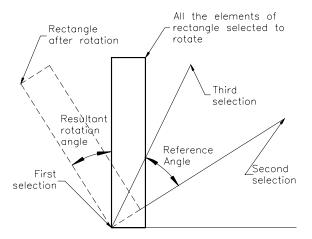
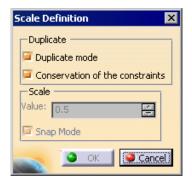*Figure 3-32 Points to be selected and the preview of the rotated elements*

*Figure 3-33 The **Scale Definition** dialog box*

Offsetting Sketched Elements

Menu:	Insert > Operation > Transformation > Offset
Toolbar:	Operation > Transformation > Offset

To offset the sketched elements, select them and then choose the **Offset** button from the **Transformation** toolbar; the **Sketch tools** toolbar expands. Specify the direction to offset the selected sketched elements and also the offset distance. Move the cursor on the side you need to specify the direction of the offset and then click in the geometry area; the selected element will be offset. You can also specify the offset distance in the **Offset** edit box in the expanded **Sketch tools** toolbar.

There are four additional buttons in the expanded **Sketch tools** toolbar, as shown in Figure 3-34. The **No Propagation**, **Tangent Propagation**, and the **Point Propagation** buttons are used to define the elements that will be selected to be offset. By default, the **No Propagation** button is chosen. This button ensures that only the selected element is offset. If you choose the **Tangent Propagation** button, all elements that are tangent to the selected element are

automatically selected. If you choose the **Point Propagation** button, all elements connected end to end with the selected element and forming a closed loop are selected automatically.
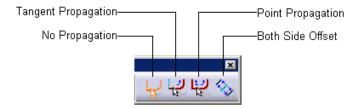

*Figure 3-34 Four additional buttons in the expanded **Sketch tools** toolbar*

If you choose the **Both Side Offset** button, the offset elements are created on both sides of the selected element. Figure 3-35 shows the elements selected to be offset and the elements created after offsetting. In this figure, only the horizontal line is selected and then the **Point Propagation** button is chosen. As a result, the entire closed loop is selected.
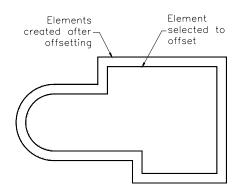
Figure 3-35 Element created after offsetting the selected element

Modifying Sketched Elements

You can modify the sketched elements using the tools in the **Sketcher** environment of CATIA V5. The modification of the sketched elements is discussed next.

Modifying the Sketched Line

You can modify a sketched line using the **Line Definition** dialog box. To invoke it, double-click on the sketched line; the **Line Definition** dialog box is displayed, as shown in Figure 3-36. You can modify the start point, endpoint, length, and angle of the line using the options available in it. After modifying the parameters, choose the **OK** button from the **Line Definition** dialog box. You can also convert the standard element to the construction element by selecting the **Construction element** check box at the bottom.
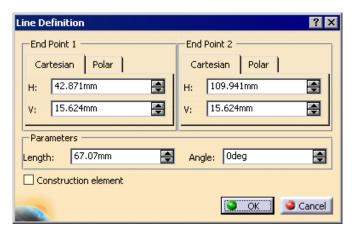
Figure 3-36 *The **Line Definition** dialog box*

Modifying the Sketched Circle

You can modify a sketched circle using the **Circle
Definition** dialog box. You can invoke this dialog box
by double-clicking on the sketched circle. The **Circle
Definition** dialog box is shown in Figure 3-37. Using
this dialog box, you can modify the coordinates of the
center point and the radius of the circle. You can also
change the standard element to a construction element
using the **Construction element** check box.

Modifying the Sketched Arc

The arcs are also modified using the **Circle Definition**
dialog box. To invoke it, double-click on the arc to be
modified. You can modify the coordinates of the center
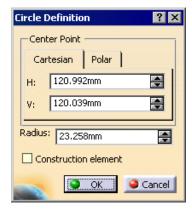point and radius of the arc using the options in this dialog
box.

Figure 3-37 *The **Circle Definition**
dialog box*

Modifying the Sketched Spline

You can modify a spline using the **Spline Definition** dialog box, which is displayed when you
double-click on the spline that needs to be modified. The **Spline Definition** dialog box is
shown in Figure 3-38. The main objective of modifying a spline is to reshape it by selecting a
sketched point that will be added as a control point to it. By default, the **Add Point After**
radio button is selected, which is used to add a control point to the spline after the specified
control point. You can also select a reference point from the selection area provided in the
dialog box, after which the point should be added. You are prompted to select the new
control point. Select the sketched point from the geometry area.

The **Add Point Before** radio button is selected to add a new control point before the selected
control point. The **Replace Point** radio button is selected to replace the selected control
point with the new sketched point.

The **Close Spline** check box is used to close the spline. You can also set the tangencies of the selected control point using the other options in this dialog box.
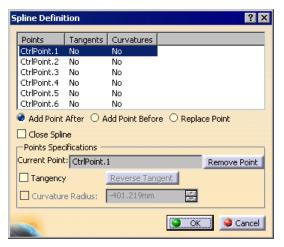

Figure 3-38 *The* **Spline Definition** *dialog box*

Modifying the Sketched Point

To modify a sketched point, double click on it; the **Point Definition** dialog box is displayed, as shown in Figure 3-39. You can modify the coordinates of the point using the options in this dialog box.

Modifying the Sketched Ellipse

To modify a sketched ellipse, double-click on it; the **Ellipse Definition** dialog box is displayed, as shown in Figure 3-40. You can modify the coordinates of the center point, major radius, minor radius, and angle of the ellipse using the options available in this dialog box.

Figure 3-39 *The* **Point Definition** *dialog box*

Similarly, you can modify the other sketched elements such as parabola, hyperbola, and so on.

Modifying the Sketched Elements by Dragging

You can also modify the parameters such as the size, shape, and position of the sketched elements by dragging. The modification of the sketched element can be done by dragging its start point, end point, profile, or control points.
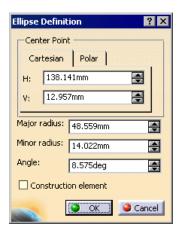
*Figure 3-40 The **Ellipse Definition** dialog box*

Deleting Sketched Elements

To delete the sketched element, select it and choose the DELETE key. You can also delete the sketched elements by selecting them and then right-clicking to invoke the contextual menu. Choose the **Delete** option from it.

TUTORIALS

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 3-41. The sketch is shown in Figure 3-42. You will not dimension the sketch. The solid model and its dimensions are given for your reference.   **(Expected time: 30 min)**
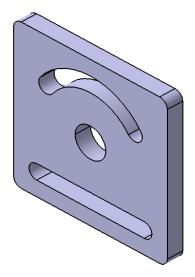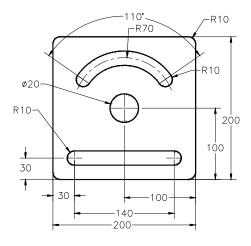

Figure 3-41 The model for Tutorial 1

Figure 3-42 The sketch for Tutorial 1

The following steps are required to complete this tutorial:

a.   Start a new file in the **Part** workbench.
b.   Draw the outer loop of the sketch using the **Rectangle** tool and then edit it using the **Corner** tool, refer to Figures 3-43 through 3-45.
c.   Draw the inner loop of the sketch using the **Circle**, **Elongated Hole**, and **Cylindrical Elongated Hole** tools, refer to Figures 3-46 through 3-48.
d.   Save and close the file.

Starting a New File in the Part Workbench and Invoking the Sketcher Workbench

1.   Choose the **New** button from the **Standard** toolbar to display the **New** dialog box.

2.   Select the **Part** option and choose the **OK** button.

3   Specify the name of the part as *c03tut1* in the **Enter part name** edit box of the **New Part** dialog box. Select the **Enable hybrid design** check box, if it is not already selected, and then choose the **OK** button; a new file in the **Part** workbench is started.

4.   Choose the **Sketch** button from the **Sketcher** toolbar and select the yz plane from the geometry area to enter in the sketcher environment.

Drawing the Outer Loop of the Sketch
To draw the outer loop of the sketches, you need to draw a centered rectangle using the **Centered Rectangle** tool. Next, you need to edit it by filleting its corners using the **Corner** tool.

Before you draw the rectangle, you need to zoom out of the geometry area to make sure you can conveniently draw it.

1.   Choose the **Zoom Out** button from the **View** toolbar.

2.   Choose the **Centered Rectangle** button from the **Predefined Profile** toolbar; you are prompted to select a point to create the center of the rectangle.

3.   Move the cursor to the origin and specify a point to define the center point of the rectangle when the value of the coordinates is displayed as 0,0 above the cursor. You are prompted to select the second point to create a centered rectangle.

4.   Move the cursor to a location whose coordinates are close to 100,100. At this location, the height and width of the rectangle is displayed as 200 and 200, respectively, in both the **Height** and **Width** edit boxes, respectively of the **Sketch tools** toolbar.

5.   Specify the point at this location to draw the rectangle, which is shown in Figure 3-43. Click anywhere in the geometry area to make sure that the rectangle is no more selected.
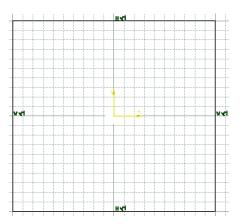
Figure 3-43 *The sketch after drawing the centered rectangle*

Next, you need to edit the rectangle by filleting its corners using the **Corner** tool.

6. Choose the **Corner** button from the **Operation** toolbar; you are prompted to select the first curve or a common point.

7. Select the upper right corner of the rectangle, as shown in Figure 3-44; the sketch toolbar expands.

8. Press the TAB key once and enter the value in the **Radius** edit box as **10**. The selected corner of the rectangle is filleted and the radius value is displayed on the fillet.

9. Similarly, fillet the other corners of the rectangle by following the procedure mentioned above. The vertices to be selected are shown in Figure 3-44. The final outer loop of the sketch, after filleting all vertices, is shown in Figure 3-45.
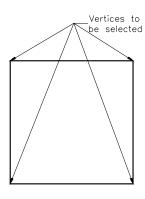

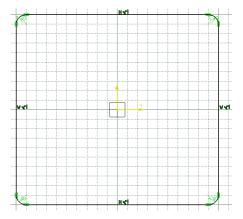***Figure 3-44*** *The vertices to be selected*

Figure 3-45 *The final outer loop of the sketch*

Drawing the Inner Loop of the Sketch

The inner loop will be drawn using the **Circle**, **Elongated Hole**, and **Cylindrical Elongated Hole** tools.

1. Choose the **Circle** button from the **Profile** toolbar; you are prompted to select a point to define the circle center. Specify the center point of the circle at the origin.

2. Move the cursor horizontally toward the right and specify a point on the circle when the value of the radius is displayed as 20 in the **R** edit box. The sketch, after drawing the circle, is shown in Figure 3-46.

    Next, you need to draw an elongated hole using the **Elongated Hole** tool.

3. Choose the **Elongated Hole** button from the **Predefined Profile** toolbar; you are prompted to define the center to center distance.

4. Move the cursor to a location whose coordinates are -70, -70 and specify the start point of line defining the center to center distance at this point.

5. Move the cursor horizontally toward the right and specify the end point on the location whose coordinates are 70, -70. Next, you are prompted to define a point on the elongated hole.

6. Move the cursor vertically upward and specify a point on the elongated hole when the **Radius** edit box shows a value of 10 as the radius. Figure 3-47 shows the sketch after drawing the elongated hole.
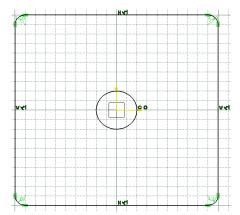
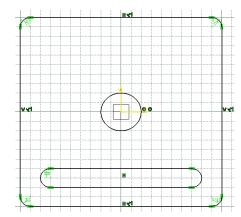*Figure 3-46* *The sketch after drawing the circle*

Figure 3-47 *The sketch after drawing the elongated hole*

After drawing the elongated hole, you need to draw a cylindrical elongated hole.

7. Choose the **Cylindrical Elongated Hole** button from the **Predefined Profile** toolbar; you are prompted to define the center to center arc.

8.   Move the cursor to the origin and specify the center point of the reference arc at this location.

9.   Enter the start H and V values as **60** and **50** in the **Sketch tools** toolbar and press ENTER.

10. Move the cursor in the counterclockwise direction to specify the end point of the reference arc at a location whose coordinates are -60, 50.

     You are prompted to define a point on the cylindrical elongated hole. You need to maintain the radius of the cylindrical elongated hole to 10. Therefore, specify the radius value in the **Radius** edit box.

11. Press the TAB key, set the value in the **Radius** edit box to **10**, and press the ENTER key. The final sketch is shown in Figure 3-48.
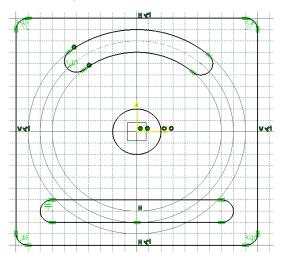

Figure 3-48 The final sketch

Saving and Closing the Sketch

1.   Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create the *c03* folder inside the *CATIA* folder.

2.   Enter the name of the file as *c03tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c03* folder.

3.   Close the part file by choosing **File > Close** from the menu bar.

Tutorial 2

In this tutorial, you will draw the sketch of the model shown in Figure 3-49. The sketch is shown in Figure 3-50. You will not dimension the sketch. The solid model and its dimensions are given for your reference.   **(Expected time: 30 min)**
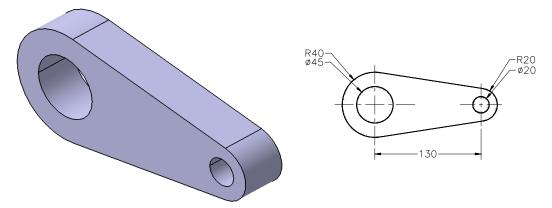
Figure 3-49 *The model for Tutorial 2*

Figure 3-50 *The sketch for Tutorial 2*

The following steps are required to complete this tutorial:

a. Start a new file in the **Part** workbench and draw the outer loop of the sketch using the **Circle** and **By-Tangent Line** tool, refer to Figures 3-51 through 3-54.
b. Trim the unwanted portion of the outer loop of the sketch using the **Quick Trim** tool, refer to Figures 3-55 and 3-56.
c. Draw the inner loops of the sketch using the **Circle** tool, refer to Figure 3-57.
d. Save and close the file.

Starting a New File in the Part Workbench

Before proceeding further, you need to start a new file in the **Part** workbench.

1. Choose the **New** button from the **Standard** toolbar; the **New** dialog box is displayed.

2. Select the **Part** option from this dialog box and choose the **OK** button.

3   Specify the name of the part as *c03tut2* in the **Enter part name** edit box of the **New Part** dialog box. Select the **Enable hybrid design** check box from the **New Part** dialog box, if it is not already selected and then choose the **OK** button; a new file in the **Part** workbench is started.

4. Choose the **Sketch** button from the **Sketcher** toolbar.

5. Select the yz plane from the specification tree or from the graphics area to enter the **Sketcher** workbench.

Drawing the Sketch

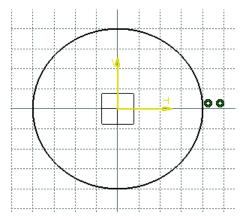The outer loop of the sketch is drawn using the **Circle** and **Bi-Tangent Line** tools.

1. Choose the **Circle** button from the **Profile** or the **Circle** toolbar; you are prompted to select a point to define the circle center.

2. Move the circle to the origin and specify the center point of the circle when the coordinates display a value of 0,0 above the cursor. Make sure the **Snap to Point** button is chosen in the **Sketch tools** toolbar.

3. Move the cursor horizontally toward the right and define the circle radius when the radius of the circle in the **R** edit box provided in the **Sketch tools** toolbar displays a value of **40**. A circle is drawn, as shown in Figure 3-51.

4. Again, choose the **Circle** button from the **Circle** toolbar.

5. Move the cursor to a location whose coordinates are 130, 0 and specify the center point of the circle.

6. Move the cursor toward the right and specify the point on the circle, when the radius displays a value of 20 in the **R** edit box in the **Sketch tools** toolbar. The sketch, after drawing the second circle, is shown in Figure 3-52.
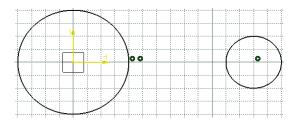

Figure 3-51 *The sketch after drawing the first circle*

Figure 3-52 *The sketch after drawing the second circle*

After drawing both the circles, you need to draw two lines in such a way that they are tangent to both of them. These lines will be drawn using the **Bi-Tangent Line** tool.

7. Choose the **Bi-Tangent Line** button from the **Line** toolbar; you are prompted to select the geometry to create a tangent line.

8. Move the cursor to the first quadrant of the first circle and specify the start point of the line; you are prompted to select the geometry to create a tangent line.

9. Move the cursor on the second quadrant of the second circle and specify the end point of the line at this location. A tangent line is drawn, as shown in Figure 3-53.

10. Similarly, draw a tangent line on the lower side of the sketch by selecting the fourth quadrant of the first circle and the third quadrant of the second circle. Figure 3-54 shows the sketch after drawing both the tangent lines.
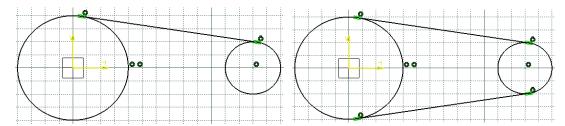
Figure 3-53 *The sketch after drawing the first tangent line*

Figure 3-54 *The sketch after drawing the second tangent line*

Trimming the Unwanted Portion of the Outer Loop of the Sketch

After drawing the outer loop of the sketch, you need to trim its unwanted portion using the **Quick Trim** tool.

1. Double-click on the **Quick Trim** tool from the **Relimitations** toolbar; you are prompted to select a curve type element.

> **Tip**. *If you click on the button once to invoke the tool, it will be active for only one time use. If you double-click to invoke a tool, it will be active even after you have used it once.*

2. Click on the unwanted portion of the sketch, as shown in Figure 3-55. The final result is shown in Figure 3-56.
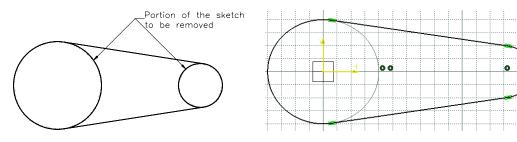

Figure 3-55 *The unwanted portion of the sketch to be trimmed*

Figure 3-56 *The sketch after trimming the unwanted portion*

Drawing the Inner Loop of the Sketch

After drawing and trimming the outer loop of the sketch, you need to draw its inner loops, which consist of two circles that will be drawn using the **Circle** tool.

1. Double-click on the **Circle** button from the **Circle** toolbar; you are prompted to select a point to define the circle center.

2. Move the cursor to the origin and specify the center point of the circle, when the value of the coordinates is 0,0.

Because the radius of this circle is not in multiples of 10, you cannot define the radius in the geometry area. Specify the radius of the circle in the R edit box in the expanded **Sketch tools** toolbar.

3. Press the **Tab** key three times to reach the **R** edit box in the **Sketch tools** toolbar.

4. Enter the value of the radius in this edit box as **45/2** (which is equal to the radius of the circle). Click in the geometry area to remove the circle from the current selection set.

    You will notice that the radius dimension value is displayed on the circle and the circle turns green. This means the circle is fully constrained. You will learn more about dimensions and constraints in later chapters.

5. Because you double-clicked on the **Circle** button, the **Circle** tool is still active. Specify the center point of the second circle at a location whose coordinates are 130,0.

6. Move the cursor horizontally toward the right and specify a point on the circle, when the coordinate values on the cursor shows 140,0. The final sketch, after creating the outer and inner loops, is shown in Figure 3-57.
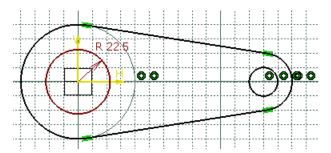

Figure 3-57 *Final sketch after creating the outer loop*

Saving and Closing the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter the name of the file as *c03tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c03* folder.

3. Close the part file by choosing **File > Close** from the menu bar.

Tutorial 3

In this tutorial, you will draw the sketch of the model shown in Figure 3-58. The sketch is shown in Figure 3-59. You will not dimension the sketch. The solid model and its dimensions are given for your reference. **(Expected time: 30 min)**
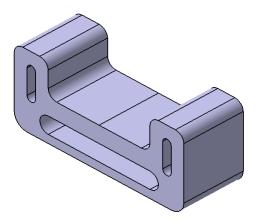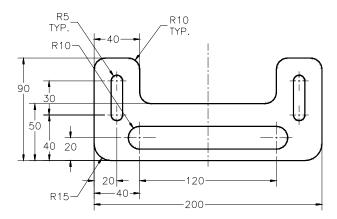

Figure 3-58 The model for Tutorial 3

Figure 3-59 The sketch for Tutorial 3

The following steps are required to complete this tutorial:

a. Draw the right half of the sketch using the **Profile** and **Elongated Hole** tool, refer to Figure 3-60.
b. Mirror the sketch along the vertical axis of origin, refer to Figure 3-61.
c. Draw the elongated hole in the lower portion of the sketch, refer to Figure 3-62.
d. Save and close the file.

Starting a New File and Invoking the Sketcher Workbench

1.   Start a new file with the name *c03tut3* in the **Part** workbench.

2.   Choose the **Sketch** button from the **Sketcher** toolbar and select the yz plane from the geometry area; the **Sketcher** workbench is invoked.

Drawing the Right Portion of the Sketch

It is evident from the figure that the sketch is symmetrical about the vertical axis. Therefore, you will draw only the right portion of the sketch and then mirror it about the vertical axis of the origin.

> **Tip**. *It is recommended that if a sketch is symmetric, you should mirror it. This helps in capturing the design intent of the model. If you modify one half of the sketch, the same modification is reflected in the other half.*

1.   Choose the **Profile** button from the **Profile** toolbar.

2.   Move the cursor to the origin and specify the start point of the line at this location.

3.   Move the cursor horizontally toward the right and specify the endpoint at a location whose coordinates are 100,0; a rubber-band line is attached to the cursor.

4.   Move the cursor vertically upward and specify the endpoint at a location whose coordinates are 100,90; another rubber-band line is attached to the cursor.

5.   Move the cursor horizontally toward the left and specify the end point at a location whose coordinates are 60,90.

6.   Move the cursor vertically downward and specify the end point at a location whose coordinates are 60,50.

7.   Move the cursor horizontally toward the left and specify the end point at a location whose coordinates are 0,50.

8.   Choose the **Profile** button from the **Profile** toolbar again to exit the tool.

Next, you need to fillet the corners of the sketch using the **Corner** tool.

9.   Choose the **Corner** button from the **Operation** toolbar.

10. Select the lower right vertex of the sketch and set the value of radius to **15** in the **Radius** edit box in the **Sketch tools** toolbar.

11. Similarly, fillet the other corners of the sketch with a radius of value 10, refer to Figure 3-59.

12. Draw a vertical elongated hole on the right of the sketch using the **Elongated Hole** tool, as shown in Figure 3-60.
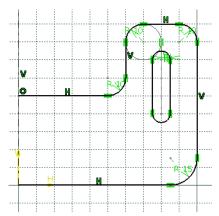
Mirroring the Sketch

After drawing the right half of the sketch, you need to mirror it along the vertical axis of the origin. The sketch is mirrored using the **Mirror** tool.

1. Drag a window around all the sketched elements to select them. Press and hold the CTRL key and select the vertical and horizontal axis of origin to remove it from the selection set, if it is also selected.

2 Choose the **Mirror** button from the **Operation** or the **Transformation** toolbar; you are prompted to selected the line or axis from which the elements will remain equidistant.

3. Select the vertical axis of origin. The sketch is mirrored to the other side of the selected axis, as shown in Figure 3-61.
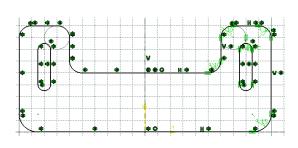

Figure 3-60 *The sketch after drawing the elongated hole*

Figure 3-61 *The sketch after mirroring*

4. Draw the horizontal elongated hole to complete the sketch. The final sketch is shown in Figure 3-62.

Saving and Closing the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.

2. Enter the name of the file as *c03tut3* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c03* folder.

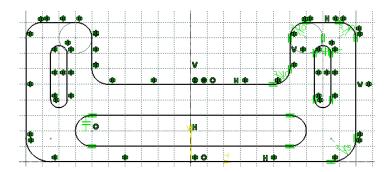3. Close the part file by choosing **File > Close** from the menu bar.

Figure 3-62 *The sketch after creating the horizontal elongated hole*

SELF-EVALUATION TEST

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

1. A parabola is a set of points whose distance from a fixed point, called focus, which equals the distance from a fixed line. (T/F)

2. In CATIA V5, you can draw a hexagon using the **Rectangle** tool. (T/F)

3. You cannot draw an n-sided polygon using the **Profile** tool. (T/F)

4. In the **Sketcher** workbench of CATIA V5, you cannot trim the sketched elements. (T/F)

5. You can draw a key hole profile in the **Sketcher** workbench of CATIA V5. (T/F)

6. After invoking the **Quick Trim** tool, if you choose the _____ button from the **Sketch tools** toolbar and select the sketched element, then it will be trimmed only along its intersection.

7. The _____ tool is used to extend the sketched elements.

8. To offset the sketched elements, select them and then choose the _____ button from the **Transformation** toolbar.

9. You can modify a sketched arc using the _____ dialog box.

10. You can modify a sketched ellipse using the _____ dialog box.

REVIEW QUESTIONS

Answer the following questions:

1. To scale the sketched elements, select them and then choose the **Translate** button from the **Transformation** toolbar. (T/F)

2. The **Rotate** tool is used to rotate the sketched elements. (T/F)

3. To create the complementary portion of an arc or a trimmed circle, choose the **Complement** button from the **Relimitations** toolbar and select the element. (T/F)

4. The **Sketcher** workbench of CATIA V5 also provides you with a tool to move the selected sketched elements from its initial position to the required destination. (T/F)

5. You can draw the cylindrical elongated hole using the **Elongated Hole** tool. (T/F)

6. Which dialog box is used to modify a sketched point?

    (a) **Sketched Point**   (b) **Point Definition**
    (c) **Modify Point**   (d) None of these

7. Which of the following properties of a line cannot be modified using the **Line Definition** dialog box?

    (a) **End Point 1**   (b) **End Point 2**
    (c) **Length**   (d) **Color**

8. Which tool is used to fillet the sketched elements?

    (a) **Fillet**   (b) **Corner**
    (c) **Chamfer**   (d) None of these

9. Which tool is used to draw a parallelogram by specifying the center point?

    (a) **Parallelogram with mid point**   (b) **Centered Rectangle**
    (c) **Centered Parallelogram**   (d) **Circle**

10. Which toolbar is used to invoke the **Keyhole Profile** tool?

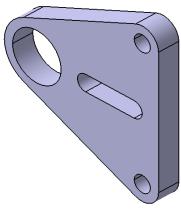   (a) **Transformation**   (b) **Relimitations**
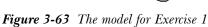   (c) **Operation**   (d) **Predefined Profile**

EXERCISES

Exercise 1

Draw the sketch of the model shown in Figure 3-63. The sketch to be drawn is shown in Figure 3-64. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**
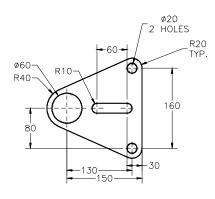

Figure 3-63 *The model for Exercise 1*

Figure 3-64 *The sketch for Exercise 1*

Exercise 2

Draw the sketch of the model shown in Figure 3-65. The sketch to be drawn is shown in Figure 3-66. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**
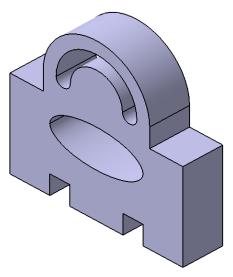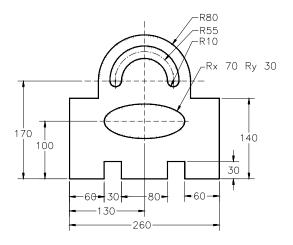

Figure 3-65 *The model for Exercise 2*

Figure 3-66 *The sketch for Exercise 2*

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** F, **4.** F, **5.** T, **6. Break and Rubber Keep**, **7. Trim**, **8. Offset**, **9. Circle Definition**,
10. Ellipse Definition