

A 3D CAD model of a mechanical assembly, featuring a grey cast part with a large central bore and a smaller side bore, and a yellow rod-like component with a hexagonal end. The assembly is shown in a perspective view.

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 by 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 IN THE SKETCHER WORKBENCH

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 Conics

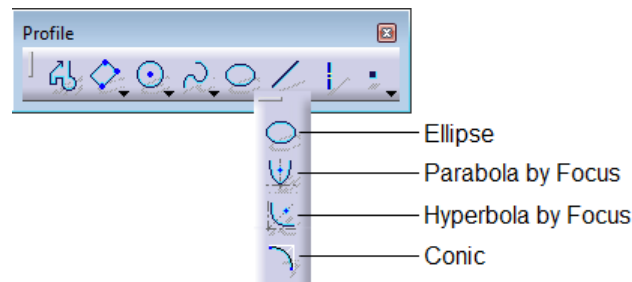
Conics are the geometrical elements that are formed by the intersection of a plane and a cone. By changing the angle and location of the intersection, you can produce an ellipse, parabola, or hyperbola. To draw a conic in CATIA V5, click on the down arrow available on the right of the **Ellipse** button in the **Profile** toolbar; the **Conic** sub-toolbar will be displayed. The tools available in this sub-toolbar are discussed next.

Drawing Ellipses

Menubar:	Insert > Profile > Conic > Ellipse
Toolbar:	Profile > Conic sub-toolbar > Ellipse



To draw an ellipse, choose the **Ellipse** tool from the **Conic** sub-toolbar in the **Profile** toolbar. Figure 3-1 shows the **Profile** toolbar with the **Conic** sub-toolbar.



*Figure 3-1 The **Profile** toolbar with the **Conic** sub-toolbar*

On choosing the **Ellipse** tool, the **Sketch tools** toolbar will expand and you will be 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 depends on the angle formed between the major axis and the **H** direction. 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 will now be prompted to specify a point on the ellipse, which will determine the other axis. Figure 3-2 shows a point being specified on the ellipse. You will notice a few construction elements displayed on it. These 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.

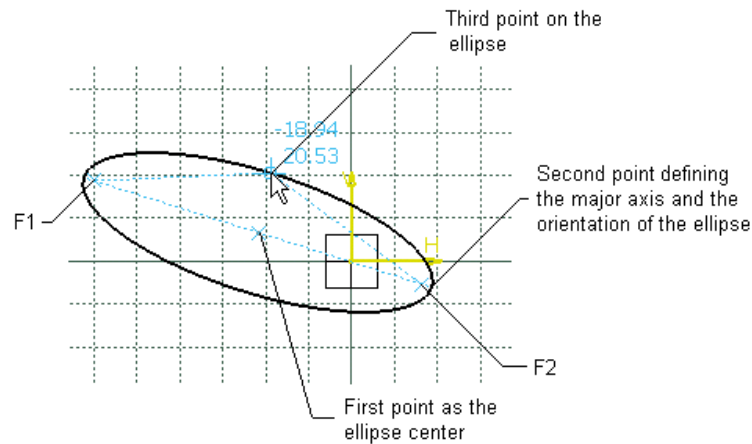


Figure 3-2 Specifying three points to draw an ellipse

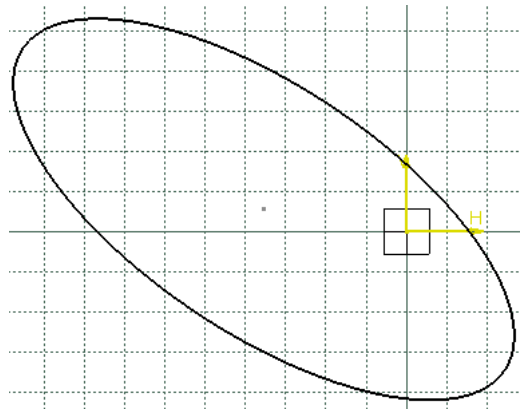


Figure 3-3 The resulting ellipse



Note

In CATIA V5, the first axis of an ellipse should be major axis.

Drawing a Parabola by Focus

Menubar: Insert > Profile > Conic > Parabola by Focus
Toolbar: Profile > Conic sub-toolbar > Parabola by Focus



To draw a parabola by focus, choose the **Parabola by Focus** tool from the **Conic** sub-toolbar in the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the focus. Click in the geometry area to specify the focus; you will be 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 the apex; you will be prompted to specify the start point. Move the cursor away from the apex and specify the start point; you will be prompted to specify the endpoint. Move the cursor along the path of the parabola and click to specify its end point. Figure 3-4 shows the points used to draw the parabola and the resulting parabola.

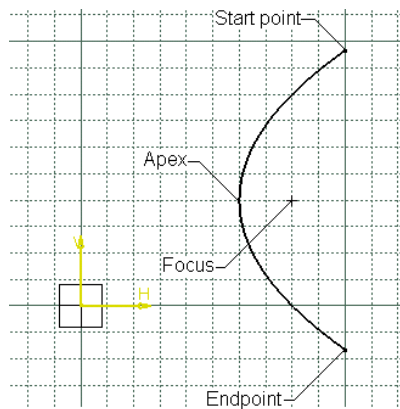


Figure 3-4 Points used to draw a parabola

Drawing a Hyperbola by Focus

Menubar: Insert > Profile > Conic > Hyperbola by Focus
Toolbar: Profile > Conic sub-toolbar > Hyperbola by Focus



To draw a hyperbola by focus, choose the **Hyperbola by Focus** tool from the **Conic** sub-toolbar in the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the focus. Click to specify the focus, which is referred to as F1 in Figure 3-5; you will be 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 3-5; you will be prompted to specify the apex of the hyperbola. Move the cursor toward focus F1 to specify the apex. 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 **e** edit box changes accordingly. Eccentricity, in case of 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 will now be 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 3-6. You can move the cursor in either direction to specify the start point. On doing so, you will be prompted to specify the endpoint. Move the cursor in the opposite direction of the start point; the preview of the hyperbola will follow the cursor. Click to specify the endpoint.



Note

In case of a parabola/hyperbola if the focus, center point, or both 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.

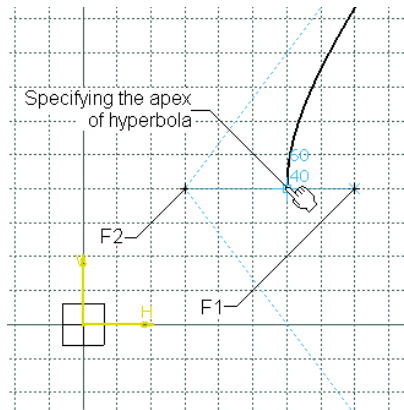


Figure 3-5 Specifying the focus and apex of the hyperbola

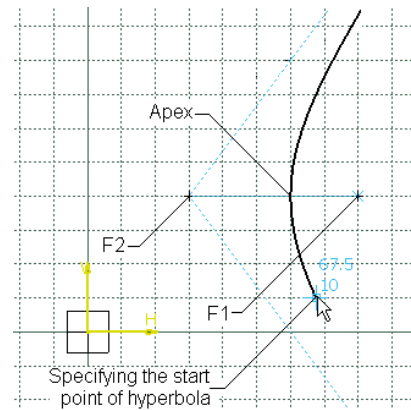


Figure 3-6 Specifying the start point of the hyperbola

Drawing Conics

Menubar: Insert > Profile > Conic > Conic
Toolbar: Profile > Conic sub-toolbar > Conic



To draw a conic, choose the **Conic** tool from the **Conic** sub-toolbar in the **Profile** toolbar; the **Sketch tools** toolbar will expand. In the expanded toolbar, the **Nearest End Point**, **Two Points**, and **Start and End Tangent** buttons will be chosen. Also, you will be prompted to specify the first endpoint. Click in the geometry area to specify the first endpoint of the conic; you will be prompted to specify the tangent at the first endpoint. Move the cursor away from the endpoint to define the constructional tangent line and then specify a point, as shown in Figure 3-7. Similarly, specify the second endpoint of the conic and its tangent line. Next, move the cursor between the two specified endpoints; the preview of the conic will be displayed. Finally, define a point on the preview to create the conic. Figure 3-7 shows the preview of conic.

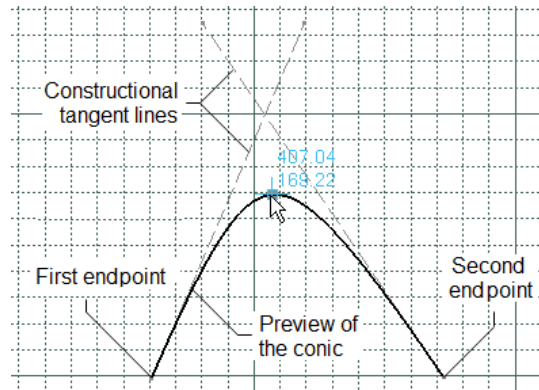


Figure 3-7 The preview of conic

Drawing Splines

Menubar: Insert > Profile > Spline > Spline
Toolbar: Profile > Spline sub-toolbar > Spline



Splines are the curves whose behavior is defined by piece wise function of polynomial equations. To draw a spline, choose the down arrow on the right of the **Spline** tool in the **Profile** toolbar; the **Spline** sub-toolbar will be displayed, as shown in Figure 3-8.

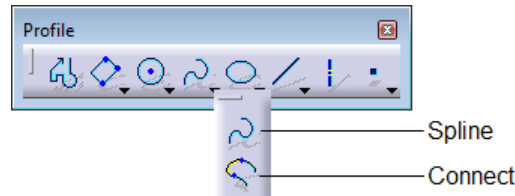


Figure 3-8 The **Profile** toolbar with the **Spline** sub-toolbar

Choose the **Spline** tool from the **Spline** sub-toolbar; you will be prompted to specify the first control point of the spline. Click to specify the first point; you will be prompted to specify the next point of the spline or double-click to specify the endpoint. Move the cursor; 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 3-9 shows a spline being drawn by specifying multiple points.



Note

In a spline, control points are construction elements, while curve is a standard element.

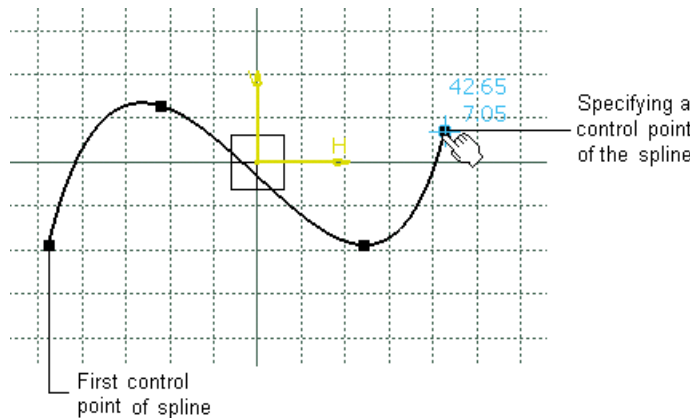


Figure 3-9 Drawing a spline by specifying multiple points

Connecting Two Elements by a Spline or an Arc

Menubar: Insert > Profile > Spline > Connect
Toolbar: Profile > Spline sub-toolbar > Connect



Two elements such as lines, arcs, ellipses, circles, or splines can be connected together by using an arc or a spline. To do so, choose the **Connect** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand, as shown in Figure 3-10.

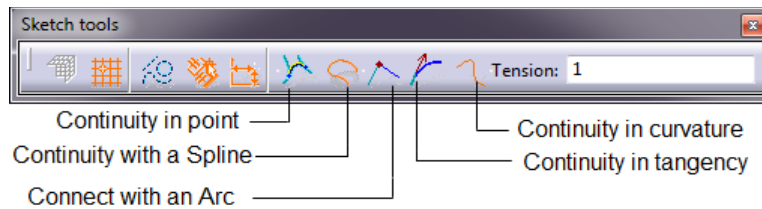


Figure 3-10 The Sketch tools toolbar after choosing the Connect tool

The next section discusses how the two selected elements can be connected by a spline or 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. Select the first element; you will be prompted to select the second element. Select the second element; both the selected elements will get connected by a spline in the geometry area.

When you connect the elements, you will notice that, by default, the **Continuity in curvature** button is chosen in the **Sketch tools** toolbar. As a result, the resulting spline will maintain a curvature continuity with the selected elements. You can set the tension value in the **Tension** edit box, if required.

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

If you choose the **Continuity in point** button from the **Sketch tools** toolbar, the resulting spline will maintain a point continuity with the selected elements. In this case, the resulting element will be a straight spline with two control points.

You can also assign different attributes at the two ends of the **Connect Curve**. To do so, choose the required continuity tool from the **Sketch tools** toolbar and then select the first curve. Next, choose the required continuity tool for the second curve and then select the second curve. You can also set different tension value for curves to be connected in the **Tension** edit box.

Connecting Two Elements with an Arc

To connect two selected elements with an arc, choose the **Connect** tool from the **Profile** 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. Select the first element; you will be prompted to select the second element. Once you specify the second element, the connecting arc generated using this tool will become tangent to the two elements.

Drawing Elongated Holes

Menubar: Insert > Profile > Predefined Profile > Elongated Hole
Toolbar: Profile > Predefined Profile sub-toolbar > Elongated Hole



An elongated hole is a geometry that consists of two parallel lines and two tangent arcs, as shown in Figure 3-11. To draw an elongated hole, choose the Elongated Hole tool from **Predefined Profile** sub-toolbar in the 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 endpoint of the distance. Move the cursor away from the first center point; a center 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 will be displayed in the geometry area. Figure 3-11 shows an elongated hole with the tangent and parallel constraints applied. These constraints will be discussed in later chapters.

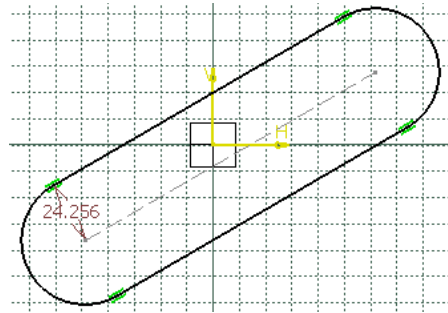


Figure 3-11 An elongated hole profile



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

Menubar: Insert > Profile > Predefined Profile > Cylindrical Elongated Hole
Toolbar: Profile > Predefined Profile sub-toolbar > Cylindrical Elongated Hole



A cylindrical elongated hole is a geometry that comprises of four arcs. Each arc is tangent to its adjacent arcs, as shown in Figure 3-12. To draw a cylindrical elongated hole, choose the **Cylindrical Elongated Hole** tool from **Predefined Profile** sub-toolbar in the **Profile** toolbar. On doing so, the **Sketch tools** toolbar will expand 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 will be attached to the cursor. Click to specify the start point; you will now 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 will be 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 will be created, as shown in Figure 3-12.

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

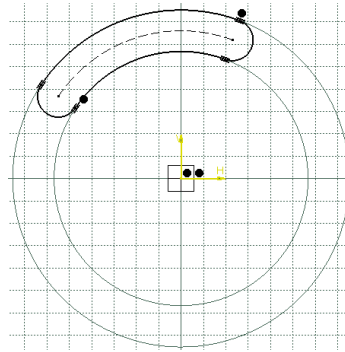



Figure 3-12 A cylindrical elongated hole profile



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

Menubar: Insert > Profile > Predefined Profile > Keyhole Profile
Toolbar: Profile > Predefined Profile sub-toolbar > Keyhole Profile



A keyhole profile is a keyhole shaped geometry that comprises of two arcs and two lines, as shown in Figure 3-13. To draw a keyhole profile, invoke the **Keyhole Profile** tool from **Predefined Profile** sub-toolbar in the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the start point. Click in the geometry area to specify the start point; you will be prompted to define the center point of the smaller radius arc. Click in geometric area to specify the center point of smaller arc; a dashed line will be displayed which defines the length of the keyhole profile. Keyhole attached to cursor will be displayed in the geometric area and you will be 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 will be 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, refer to Figure 3-13. You can also specify the required parameters in the **Sketch tools** toolbar to draw a keyhole profile.

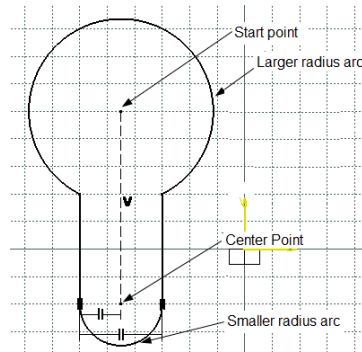


Figure 3-13 A keyhole profile

Drawing Hexagons

Menubar: Insert > Profile > Predefined Profile > Hexagon
Toolbar: Profile > Predefined Profile sub-toolbar > Hexagon



To draw a hexagon, choose the **Hexagon** tool from **Predefined Profile** sub-toolbar in the **Profile** toolbar; you will be prompted to select the center of the hexagon. 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 resultant hexagon is shown in Figure 3-14.

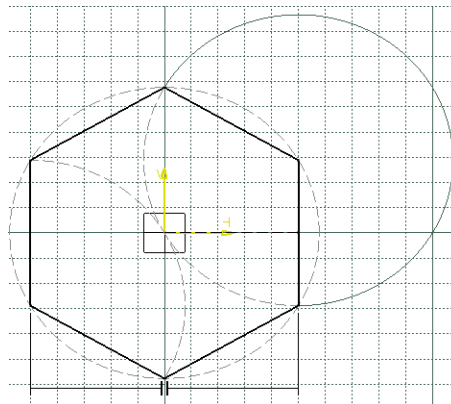


Figure 3-14 Hexagon drawn using the **Hexagon** tool

Drawing Centered Rectangles

Menubar: Insert > Profile > Predefined Profile > Centered Rectangle
Toolbar: Profile > Predefined Profile sub-toolbar > Centered Rectangle



In CATIA V5, you can draw a rectangle that is centered about a point. To draw a centered rectangle, choose the **Centered Rectangle** tool from **Predefined Profile** sub-toolbar in the **Profile** toolbar; you will be prompted to select a point to create the

center of the rectangle. Specify a point in the geometry area and move the cursor; the preview of the rectangle will be displayed. Specify a point on any one corner of the rectangle. Figure 3-15 shows a centered rectangle, along with its center point and the point at its corner.

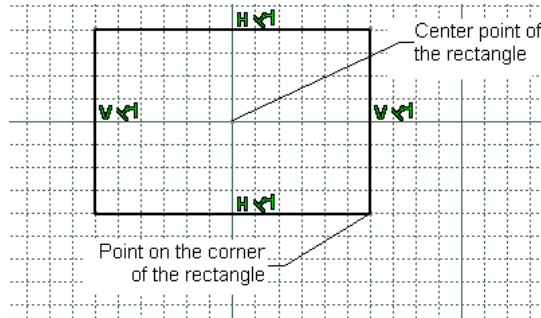


Figure 3-15 Centered rectangle along with the center point and the point at its corner

Drawing Centered Parallelograms

Menubar: Insert > Profile > Predefined Profile > Centered Parallelogram
Toolbar: Profile > Predefined Profile sub-toolbar > 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** tool from the **Predefined Profile** sub-toolbar in the **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, and the second set of the opposite sides parallel to second selected line. Also, you will be prompted to select the endpoint to create a centered parallelogram. Move the cursor and specify a point on any one of the corners of the parallelogram. Figure 3-16 shows the centered parallelogram with the first and second reference lines and the point on the parallelogram.



Note

After drawing the centered parallelogram, you can convert the reference lines into construction elements by selecting them and using the **Construction/Standard Element** button in the **Sketch tools** toolbar.

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 tools are used 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.

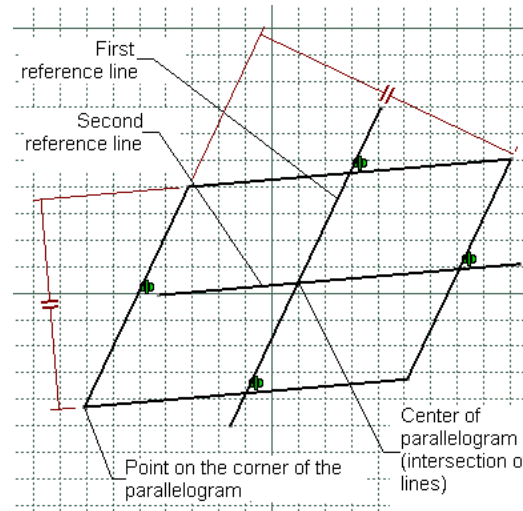


Figure 3-16 Centered parallelogram with the first and second reference lines, and the point on the parallelogram

Trimming Unwanted Sketched Elements

Menubar: Insert > Operation > Relimitations > Trim
Toolbar: Operation > Relimitations sub-toolbar > Trim



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

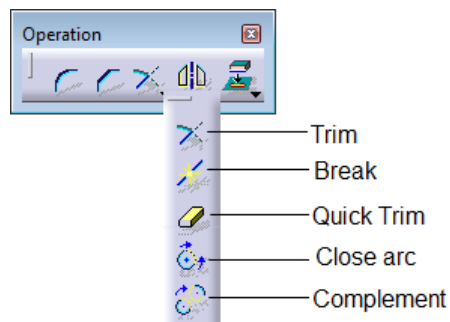


Figure 3-17 The **Operation** toolbar with the **Relimitations** sub-toolbar

Choose the **Trim** tool from the **Relimitations** sub-toolbar in the **Operation** toolbar; the **Sketch tools** toolbar will expand and you will be 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. Note that the sides that you click while selecting the elements will be retained after trimming.

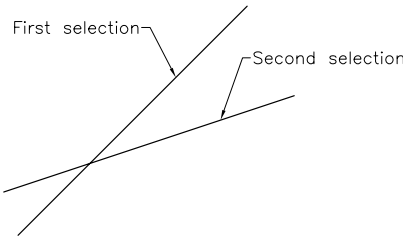


Figure 3-18 Elements to be selected for trimming

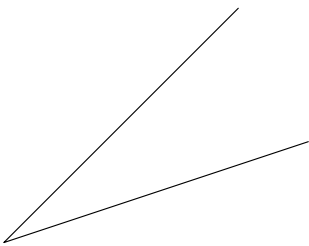


Figure 3-19 The resulting trimmed elements

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

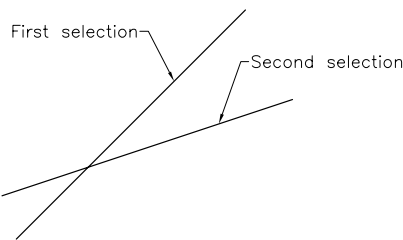


Figure 3-20 Elements to be selected for trimming

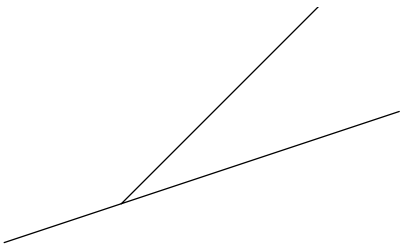


Figure 3-21 The resulting trimmed element

Extending Sketched Elements

Menubar: Insert > Operation > Relimitations > Trim
Toolbar: Operation > Relimitations sub-toolbar > Trim



In CATIA V5, you can also extend the sketched elements by using the **Trim** tool. To do so, invoke this tool; you will be 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.

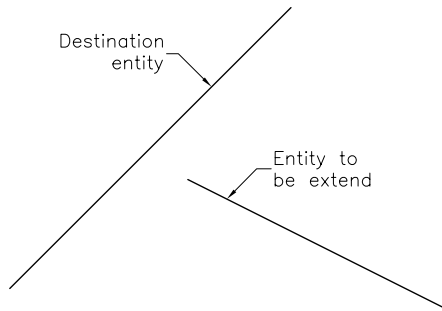


Figure 3-22 Element selected to be extended

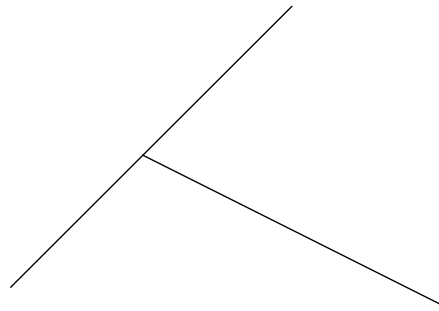


Figure 3-23 The resulting extended element

Trimming by Using the Quick Trim Tool

Menubar: Insert > Operation > Relimitations > Quick Trim
Toolbar: Operation > Relimitations sub-toolbar > Quick Trim



The **Quick Trim** tool is used to quickly trim the unwanted sketched elements. To trim an element, choose the **Quick Trim** tool from the **Relimitations** sub-toolbar in the **Operation** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to select a curve type element. By default, the **Break And Rubber In** option is chosen in the **Sketch tools** toolbar. This option results in the breakage of selected element with respect to the intersecting elements and the selected portion of the first element will be removed from the geometry. The **Break And Rubber Out** option also breaks the first selected element. But the selected portion will be retained. The **Break And Keep** option in the **Sketch tools** toolbar is used to break the selected element at the point of intersection. In this case, no portion will be removed from geometry. 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. You can close an arc or a trimmed circle to form a complete circle using the **Close arc** tool in the **Relimitations** toolbar. Choose the **Close arc** tool 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 arc**; the trimmed circle or arc will be closed.

You can also convert the complementary side of the trimmed circle or an arc to a standard element and remove its existing portion. To do so, choose the **Complement** tool 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

Menubar: Insert > Operation > Corner > 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 choose this tool, the **Sketch tools** toolbar will expand and you will be prompted to select the first curve or a common point. Select the first element to be filleted; you will be prompted to select the second curve. Select it and specify the fillet radius in the **Radius** edit box in the expanded **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:







- the **Trim All Elements** button, both the selected elements will be trimmed beyond the fillet region. This button is chosen by default. 
- 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. 
- the **No Trim** button, the resulting fillet will be created by retaining both the selected elements. 
- 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. 
- 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. 
- 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 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.

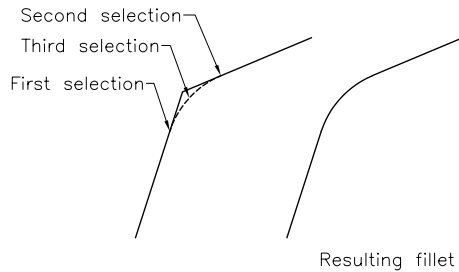


Figure 3-24 Fillet created using the **Trim All Elements** button

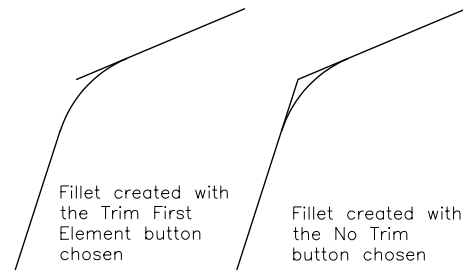


Figure 3-25 Fillets created using the **Trim First Element** and **No Trim** buttons

Chamfering Sketched Elements

Menubar: Insert > Operation > Chamfer
Toolbar: Operation > Chamfer



The **Sketcher** workbench of CATIA V5 also provides you with the **Chamfer** tool to chamfer the sketched elements. On invoking this tool, the **Sketch tools** toolbar will expand and you will be prompted to select the first curve or a common point. Select the first element; you will be prompted to select the second element. When you select the second element, the **Sketch tools** toolbar will expand and the **Angle** and **Length** edit boxes will be activated. Specify the values in these edit boxes 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 resultant chamfer.

After selecting the geometries to be chamfered, the **Sketch tools** toolbar expands providing you with some options to specify the parameters of the chamfer. These options are explained next.

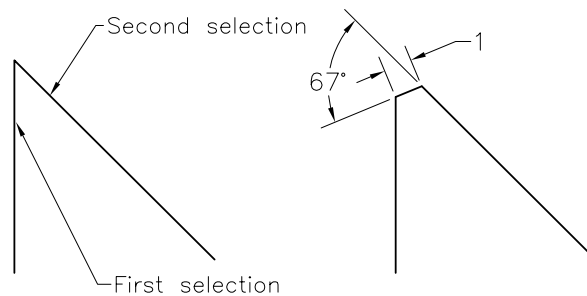


Figure 3-26 Elements to be selected and the resulting chamfer

If you choose the **Hypotenuse And Angle** 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.



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



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



You can also 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 those discussed while filleting the elements.

Mirroring Sketched Elements

Menubar: Insert > Operation > Transformation > Mirror
Toolbar: Operation > Transformation sub-toolbar > Mirror



You can mirror the sketched elements along the mirror line using the **Mirror** tool in the **Sketcher** workbench of CATIA V5. For mirroring the sketched elements, choose the down arrow on the right of the **Mirror** button provided in the **Operation** toolbar; the **Transformation** sub-toolbar will be displayed, as shown in Figure 3-27. The tools in this sub-toolbar are known as the transformation tools.

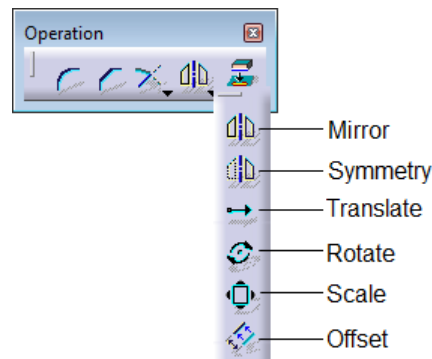


Figure 3-27 The **Operation** toolbar with the **Transformation** sub-toolbar

Select the sketched elements that you need to mirror by dragging a window around them. Alternatively, you can press and hold the CTRL key and select the elements for multiple element selection. Next, choose the **Mirror** button from the **Transformation** toolbar; you will be prompted to select the line or axis from which the elements will remain equidistant. Select a line, center line, or any of the axes as the mirror axis; the selected elements will be mirrored about the mirror axis and the symmetry constraints will be 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.

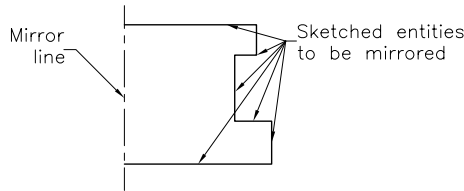


Figure 3-28 Elements selected to be mirrored and the mirror line to be selected

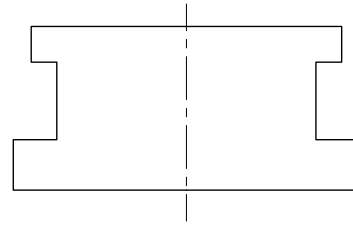


Figure 3-29 The resulting mirrored sketch

Mirroring Elements without Duplication

Menubar: Insert > Operation > Transformation > Symmetry
Toolbar: Operation > Transformation sub-toolbar > 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 **Transformation** sub-toolbar in the **Operation** toolbar; you will be 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

Menubar: Insert > Operation > Transformation > Translate
Toolbar: Operation > Transformation sub-toolbar > Translate



The **Sketcher** workbench provides you the **Translate** tool to move the selected sketched elements from their initial position to the required place. To move the sketched elements, select them and then choose the **Translate** button from **Transformation** sub-toolbar in the **Operation** toolbar; the cursor will be replaced by a point cursor and the **Translation Definition** dialog box will be displayed, as shown in Figure 3-30. Also, you will be 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 will not be displayed any more. Specify a point in the geometry area to place the selected sketched element. As the **Duplicate mode** check box

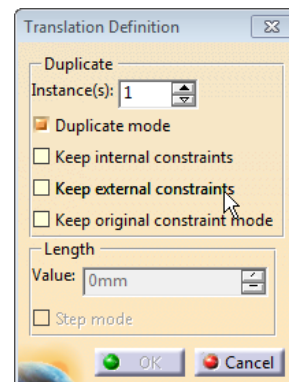


Figure 3-30 The **Translation Definition** dialog box

is selected by default and the value of the instance is set to 1, a copy of the selected element will be created at the specified distance. You can also increase the value of the increment using the **Instance(s)** spinner.

You can select the **Keep internal constraints** and **Keep external constraints** check boxes to retain the internal and external constraints, respectively. Select the **Keep original constraint mode** to keep the constraint in original mode. You will learn more about them in later chapters.

While specifying the start point and destination point of the translation, if you select the **Step 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 specify a location where you need to place it.



Note

If the **Duplicate mode** check box is cleared, then you can only move the selected elements but cannot copy them.

Rotating Sketched Elements

Menubar: Insert > Operation > Transformation > Rotate
Toolbar: Operation > Transformation sub-toolbar > 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 **Transformation** sub-toolbar in the **Operation** toolbar; the cursor will be replaced by the point cursor and the **Rotation Definition** dialog box will be displayed, as shown in Figure 3-31. Also, you will be prompted to select the rotation center point.

Specify a point around which the selected sketched elements will be rotated; you will be prompted to select a point to define a reference line for the angle. Specify a point; you will be 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 will also be displayed. Select a point to specify the rotation angle. Since the **Duplicate mode** check box is selected by default, another copy of the rotated element will be created. Figure 3-32 shows the points to be selected and the preview of the rotated instance of the selected elements.

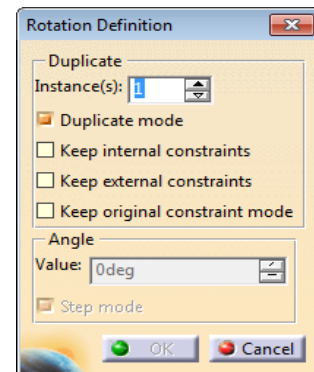


Figure 3-31 The **Rotation Definition** dialog box

You can rotate the sketch elements in either direction (clockwise or counterclockwise) by entering a negative or positive value for the rotational angle in the **Value** spinner.

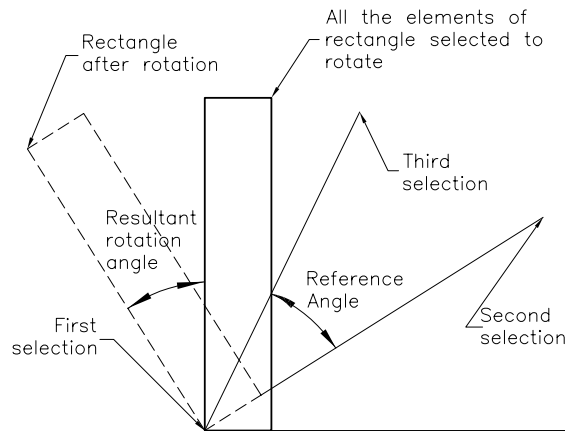


Figure 3-32 Points to be selected and the preview of the rotated elements

Scaling Sketched Elements

Menubar: Insert > Operation > Transformation > Scale
Toolbar: Operation > Transformation sub-toolbar > Scale



To scale the sketched elements, select them and then choose the **Scale** button from **Transformation** sub-toolbar in the **Operation** toolbar; the **Scale Definition** dialog box will be displayed, as shown in Figure 3-33, and you will be prompted to select the scaling center point. Select a point in the drawing window; you will be 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.

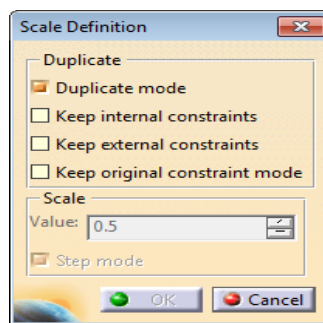


Figure 3-33 The Scale Definition dialog box

Offsetting Sketched Elements

Menubar: Insert > Operation > Transformation > Offset
Toolbar: Operation > Transformation sub-toolbar > Offset



To offset the sketched elements, select them and then choose the **Offset** button from **Transformation** sub-toolbar in the **Operation** toolbar; the **Sketch tools** toolbar will

expand. Specify the direction to offset the selected sketched elements and also the offset distance. Move the cursor on the side to which you want 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 **Point Propagation** buttons are used to define the elements that will be selected to 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 will be automatically selected. If you choose the **Point Propagation** button, all elements connected end to end with the selected element and forming a closed loop will be selected automatically.

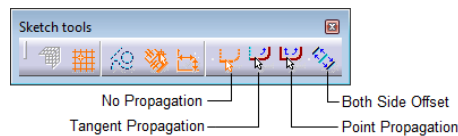


Figure 3-34 The options in the expanded **Sketch tools** toolbar

If you choose the **Both Side Offset** button, the offset elements will be 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.

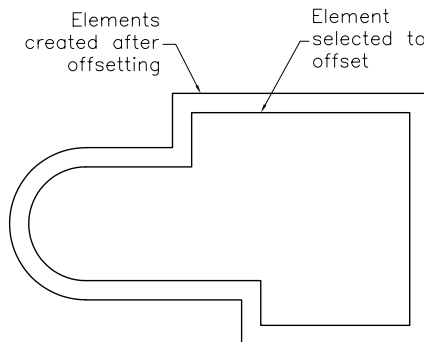


Figure 3-35 Elements created after offsetting the selected element

Modifying Sketched Elements

In the **Sketcher** environment of CATIA V5, you can modify a sketched element by double-clicking on it. 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 modify a sketched line, double-click on it; the **Line Definition** dialog box will be 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.

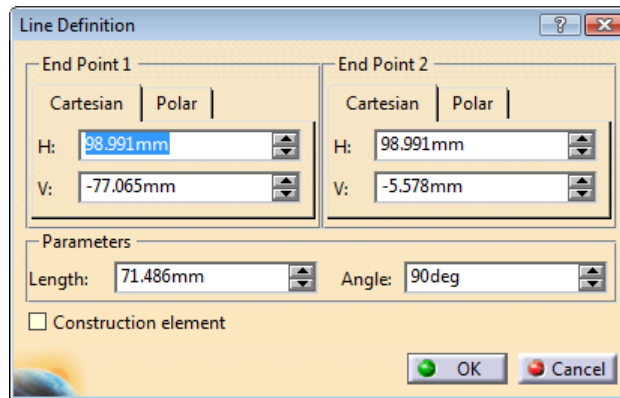


Figure 3-36 The *Line Definition* dialog box

Modifying the Sketched Circle

You can modify a sketched circle by 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 by selecting the **Construction element** check box.

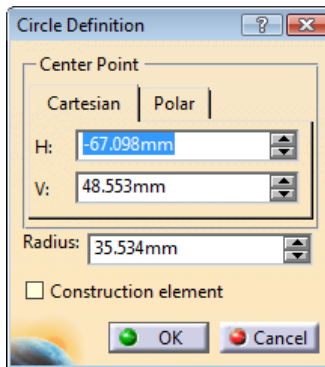


Figure 3-37 The *Circle Definition* dialog box

Modifying the Sketched Arc

The arcs are also modified using the **Circle Definition** dialog box. To invoke this dialog box, 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.

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. 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 in this dialog box and it is used to add a control point to the spline after the specified control point. As a result, you will be prompted to select the new control point. Click in the geometry area;

the new point will be added in the spline as a control point. You can also select a reference point, after which the control point should be added, from the selection area provided in the dialog box.

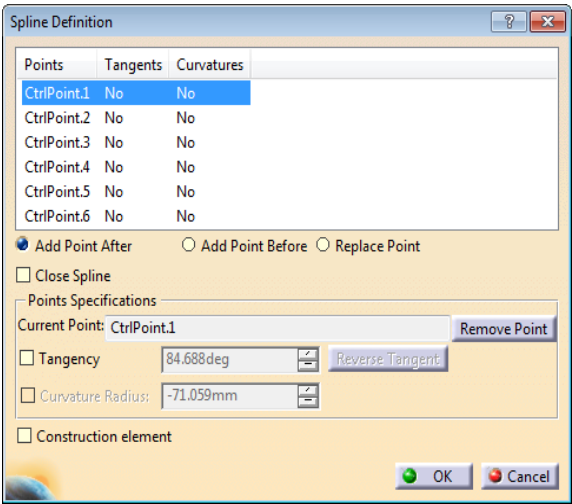


Figure 3-38 The Spline Definition dialog box

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 control point.

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

Modifying the Sketched Point

To modify a sketched point, double-click on it; the **Point Definition** dialog box will be 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.

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

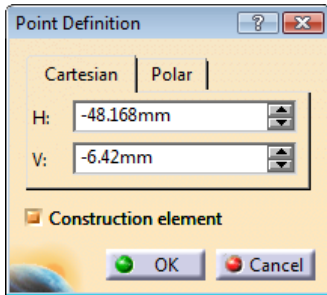


Figure 3-39 The Point Definition dialog box

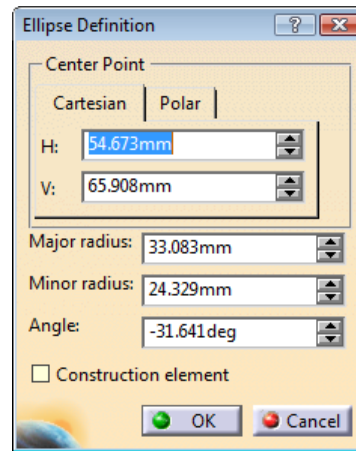


Figure 3-40 The Ellipse Definition dialog box

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, endpoint, profile, or control points.

Deleting Sketched Elements

To delete a sketched element, select it and press the DELETE key. Alternatively, select the entity to be deleted and then right-click to invoke the contextual menu. Then, 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. Its sketch is shown in Figure 3-42. Do not dimension the sketch. The solid model and its dimensions are given only for reference.

(Expected time: 30 min)

The following steps are required to complete this tutorial:

- Start a new part file in the **Part** workbench.
- 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.
- 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.
- Save and close the file.

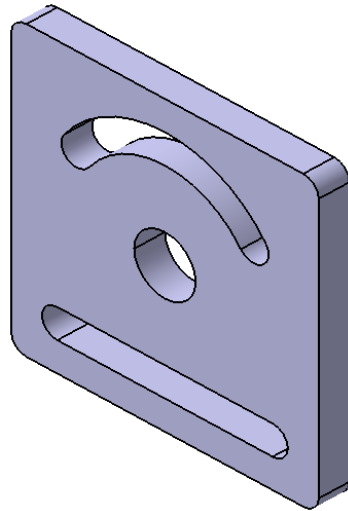


Figure 3-41 The model for Tutorial 1

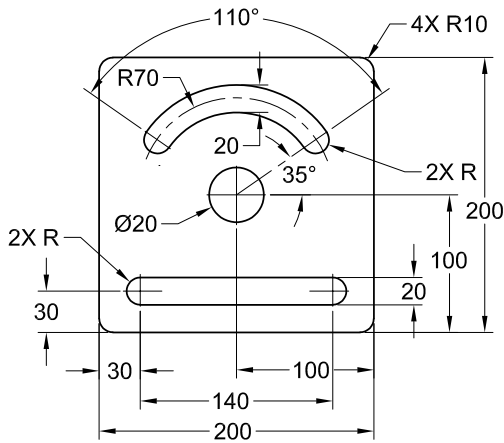




Figure 3-42 The sketch for Tutorial 1

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 from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
3. Enter the name of the part as *c03tut1* in the **Enter part name** edit box of the **New Part** dialog box and select the **Enable hybrid design** check box, if it is not already selected. Next, 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 Specification tree to enter in the **Sketcher** environment. 

Drawing the Outer Loop of the Sketch

To draw the outer loop of sketches, you need to draw a 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 the geometry area to draw the rectangle conveniently.

1. Choose the **Zoom Out** button from the **View** toolbar and make sure that the **Snap to Point** button is chosen in the **Sketch tools** toolbar.
2. Choose the **Centered Rectangle** button from the **Predefined Profile** sub-toolbar in the **Profile** toolbar; you are prompted to specify a point to create the center of the rectangle.

**Note**

To invoke the **Predefined Profile** subtoolbar, choose the down arrow available on the right of the **Rectangle** tool in the **Profile** toolbar; the **Predefined Profile** sub-toolbar is displayed.

3. Move the cursor to the origin and click to specify a point to define the center point of the rectangle when the value of the coordinates above the cursor is displayed as 0,0; you are prompted to specify the second point to create a centered rectangle.
4. Move the cursor to a location whose coordinates are close to 100,100 and click when 200 is displayed in the **Height** and **Width** edit boxes in the **Sketch tools** toolbar; a rectangle is drawn, as shown in Figure 3-43. Now, click anywhere in the geometry area to make sure that the rectangle is no more selected.

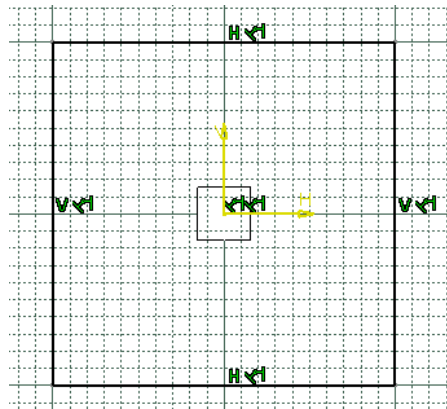



Figure 3-43 The sketch after drawing the centered rectangle

Next, you need to edit the rectangle by filleting its corners using the Corner tool. The vertices to be selected are shown in Figure 3-44.

5. Choose the **Corner** button from the **Operation** toolbar; you are prompted to select the first curve or a common point. 
6. Select the upper right corner of the rectangle; the **Sketch tools** toolbar expands.
7. Press the TAB key and enter **10** in the **Radius** edit box. Next, press the ENTER key; the selected corner of the rectangle is filleted and the radius value is displayed on the fillet.
8. Similarly, fillet the other corners of the rectangle by following the procedure mentioned in preceding steps. The final outer loop of the sketch, after filleting all vertices, is shown in Figure 3-45.

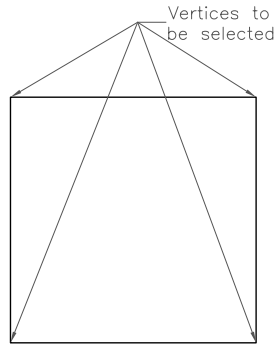


Figure 3-44 The vertices to be selected

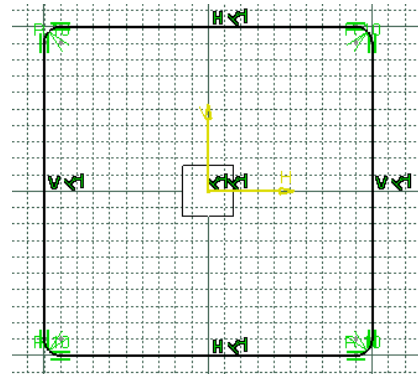


Figure 3-45 The final outer loop of the sketch

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 center of the circle. Specify the center point of the circle at the origin.
2. Move the cursor horizontally toward the right and click to 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** sub-toolbar in the **Profile** toolbar; you are prompted to define the center to center distance of the elongated hole.
4. Move the cursor to a location whose coordinates are -70,-70 and click to specify the start point of the center to center distance of the elongated hole.
5. Move the cursor horizontally toward the right and click to specify the endpoint at the location whose coordinates are 70,-70; you are prompted to define a point on the elongated hole.
6. Move the cursor vertically upward and click to specify a point on the elongated hole at the location where the radius value is displayed as **10** in the **Radius** edit box. Figure 3-47 shows the sketch after drawing the elongated hole.



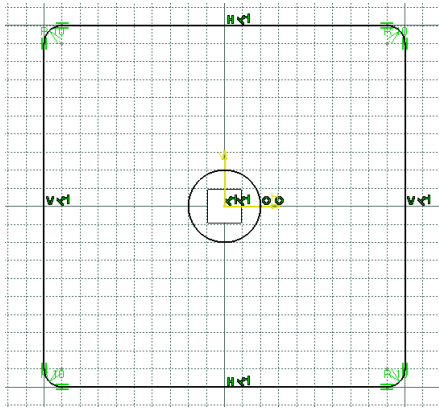


Figure 3-46 The sketch after drawing the circle

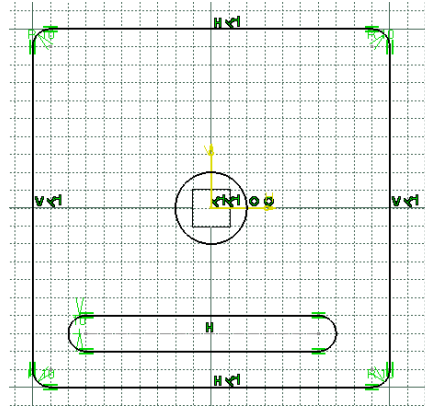



Figure 3-47 The sketch after drawing the elongated hole

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

7. Choose the **Cylindrical Elongated Hole** button from the **Predefined Profile** sub-toolbar in the **Profile** toolbar ; you are prompted to define the center to center arc. 
8. Move the cursor to the origin and click to specify it as the center point of the reference arc.
9. Press the TAB key four times and enter the value **70** in the **R** edit box to specify the radius at the start point of the elongated hole. Next, press the ENTER key.
10. Press the TAB key once to specify the angular location of the start point of the elongated hole with respect to the horizontal reference; the **A** edit box gets highlighted.
11. Enter the value **35** in the **A** edit box and press the ENTER key.
12. Press the TAB key four times and enter the value **110** in the **S** edit box to specify the angle between the start point and the endpoint of the elongated hole. Next, press the ENTER key.
13. Now, press the TAB key five times and enter the value **10** in the **Radius** edit box. Next, press the ENTER key; the final sketch is created, as shown in Figure 3-48.



Note

You will notice that some dimensional and geometrical constraints are applied to the sketch, because the **Geometrical Constraints** and **Dimensional Constraints** buttons are chosen in the **Sketch tools** toolbar by default.

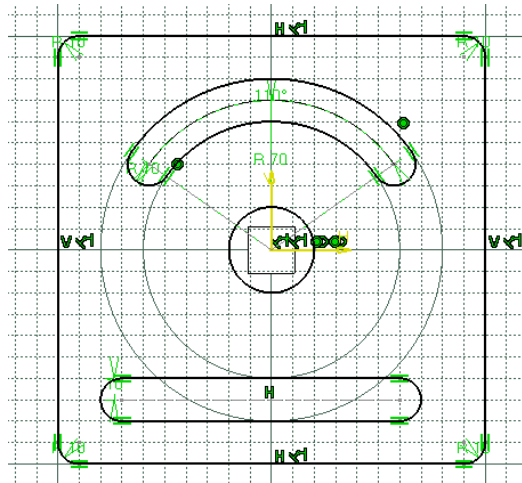


Figure 3-48 The final sketch

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. Choose the **Save** button from this dialog box; the file is saved at *C:\CATIA\c03*.
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. Do not dimension the sketch. The solid model and its dimensions are given only for your reference.

(Expected time: 30 min)

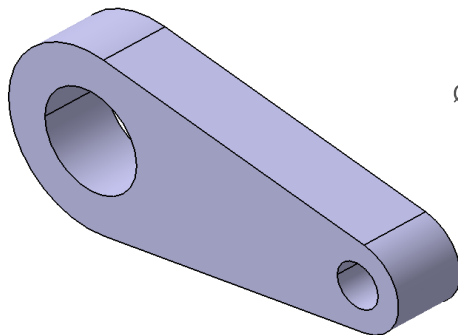


Figure 3-49 The model for Tutorial 2

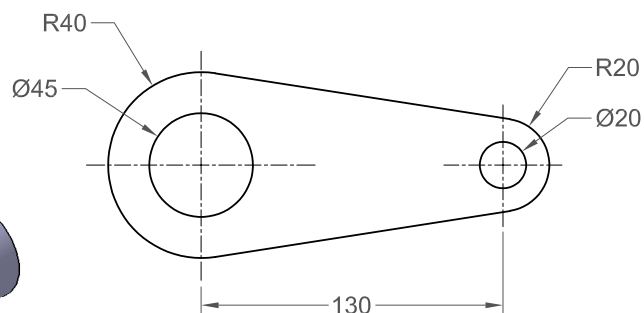




Figure 3-50 The sketch of the model for Tutorial 2

The following steps are required to complete this tutorial:

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

- Choose the **New** button from the **Standard** toolbar; the **New** dialog box is displayed. 
- Select the **Part** option from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
- 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 **Part** file is started in the **Part** workbench.
- Choose the **Sketch** button from the **Sketcher** toolbar. 
- 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.

- Choose the **Circle** button from the **Profile** toolbar; you are prompted to select a point to define the center of the circle. 
- Move the cursor toward the origin and click to specify the center point of the circle when the coordinates above the cursor display 0,0. Make sure the **Snap to Point** button is chosen in the **Sketch tools** toolbar.
- Move the cursor horizontally toward the right and click when the **R** edit box in the **Sketch tools** toolbar displays **40**; the circle is drawn, as shown in Figure 3-51. 
- Again, choose the **Circle** button from the **Profile** toolbar.
- Move the cursor to a location whose coordinates are 130, 0 and click to specify the center point of the circle.

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

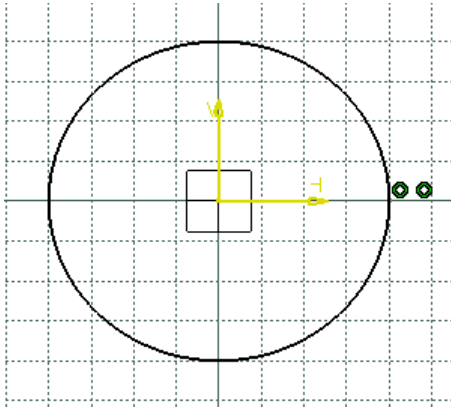


Figure 3-51 The sketch after drawing the first circle

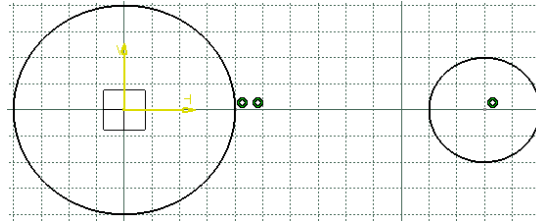


Figure 3-52 The sketch after drawing the second circle

After drawing both 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.

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



Note

Choose the down arrow available on the right of the **Line** tool in the **Profile** toolbar; the **Line** sub-toolbar is displayed.

- Move the cursor to the first quadrant of the first circle and specify the start point of the line on its circumference; you are prompted to select the geometry to create a tangent line.
- Move the cursor to the first quadrant of the second circle and specify the endpoint of the line on its circumference; a tangent line is drawn, as shown in Figure 3-53.
- Similarly, draw a tangent line on the lower side of the sketch by selecting the fourth quadrants of the first and second circles. Figure 3-54 shows the sketch after drawing the second tangent lines.

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.

- Click on the arrow available on the right of the **Trim** tool in the **Operation** toolbar; the **Relimitations** sub-toolbar is displayed. Double-click on the **Quick Trim** tool from the **Relimitations** sub-toolbar.



- Click on the unwanted portion of the sketch, refer to Figure 3-55. The final sketch is shown in Figure 3-56.

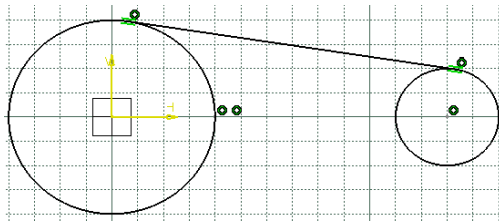


Figure 3-53 The sketch after drawing the first tangent line

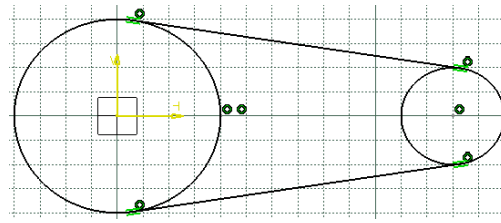


Figure 3-54 The sketch after drawing the second tangent line

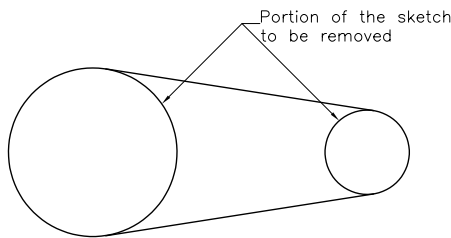


Figure 3-55 The unwanted portion of the sketch to be trimmed

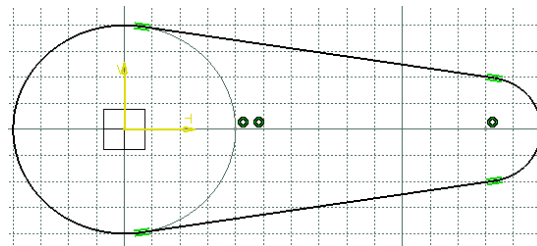



Figure 3-56 The sketch after trimming the unwanted portion



Tip. If you click on the button once to invoke a tool, the tool will be active for only one time use. However, if you double-click to choose a tool, the tool will remain active unless you terminate/exit it.

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.

- Double-click on the **Circle** button from the **Profile** toolbar; you are prompted to select a point to define the center of the circle. 
- Move the cursor to the origin and click to specify the center point of the circle when the value of the coordinates is 0,0.

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

- Press the TAB key three times to display the **R** edit box in the **Sketch tools** toolbar and enter $45/2$ (which is the radius of the circle). Next, press the ENTER key; the inner circle is created.

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

4. As 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.
5. Move the cursor horizontally toward the right and click when the coordinate values above the cursor are 140,0. The final sketch, after creating the outer and inner loops, is shown in Figure 3-57. Press the ESC key to exit the selection set and the **Circle** tool.

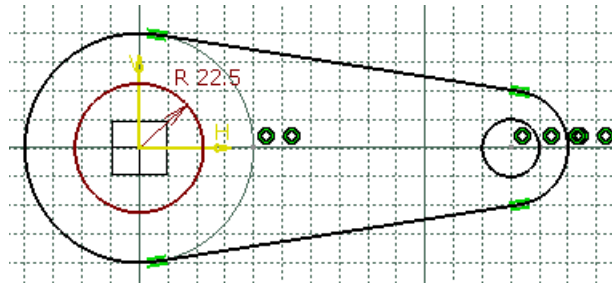


Figure 3-57 Final sketch after creating the outer and inner loops

Saving and Closing the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.
2. Choose the **Save** button from this dialog box; the file is saved at *C:\CATIA\c03*.
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. Do not dimension the sketch. The solid model and its dimensions are given only for your reference.
(Expected time: 30 min)

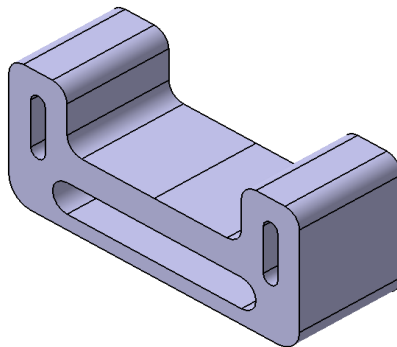


Figure 3-58 The model for Tutorial 3

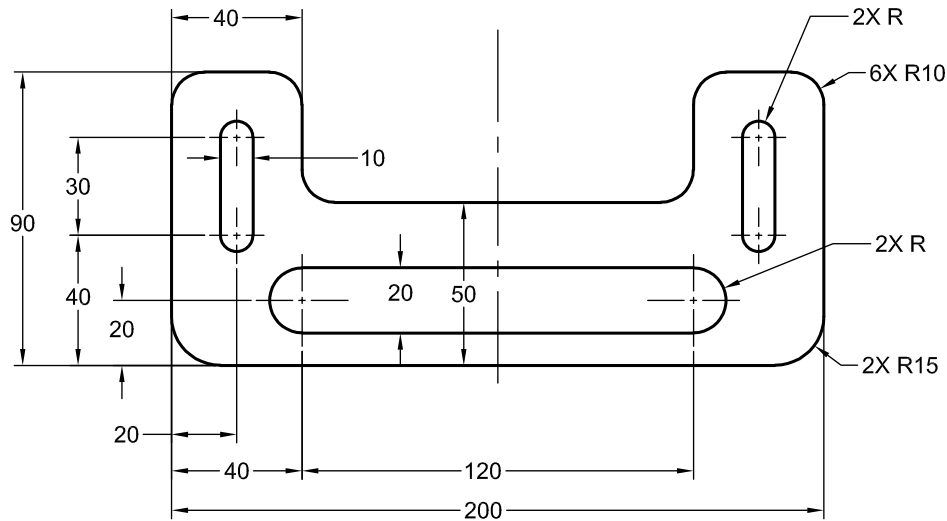


Figure 3-59 The sketch of the model for Tutorial 3

The following steps are required to complete this tutorial:

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

Starting a New File and Invoking the Sketcher Workbench

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

- Choose the **Profile** tool from the **Profile** toolbar.
- Move the cursor to the origin and specify the start point of the line at this location.
- Move the cursor horizontally toward the right and click to specify the endpoint at a location where the coordinates are 100,0; a rubber-band line is attached to the cursor.
- Move the cursor vertically upward and click to specify the endpoint at a location where the coordinates are 100,90; another rubber-band line is attached to the cursor.




5. Move the cursor horizontally toward the left and click to specify the endpoint at a location where the coordinates are 60,90.
6. Move the cursor vertically downward and click to specify the endpoint at a location where the coordinates are 60,50.
7. Move the cursor horizontally toward the left and click to specify the endpoint at a location where the coordinates are 0,50.
8. Again, choose the **Profile** button from the **Profile** toolbar 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 **15** as the value of the radius in the **Radius** edit box in the **Sketch tools** toolbar. Next, press ENTER.
11. Similarly, fillet other corners of the sketch with a radius value 10, refer to Figure 3-59.
12. Draw a vertical elongated hole on the right of the sketch using the **Elongated Hole** tool, such that the coordinate value of start and end points are 80,40 and 80,70, respectively. Specify the value of radius as **5** in the **Radius** edit box of the **Sketch tools** toolbar, refer to Figure 3-60.

Mirroring the Sketch

After drawing the right half of the sketch, you need to mirror it about 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. Next, press and hold the CTRL key and select the vertical and horizontal axes displayed at the origin to remove them from the selection set, if they are also selected.
2. Choose the **Mirror** button from the **Operation** toolbar; you are prompted to select the line or axis from which the elements will remain equidistant. 
3. Select the vertical axis; the sketch is mirrored to the other side of the selected axis, as shown in Figure 3-61.
4. Draw the horizontal elongated hole, such that the coordinate values of the start and end points are 60,20 and -60,20, respectively. Specify the value of radius as **10** in the **Radius** edit box of the **Sketch tools** toolbar. The final sketch is shown in Figure 3-62. Press the ESC key to exit the selection set.

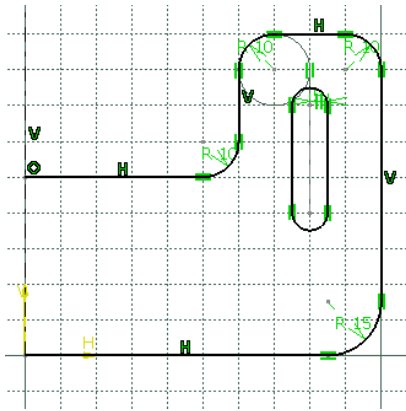


Figure 3-60 The sketch after drawing the elongated hole

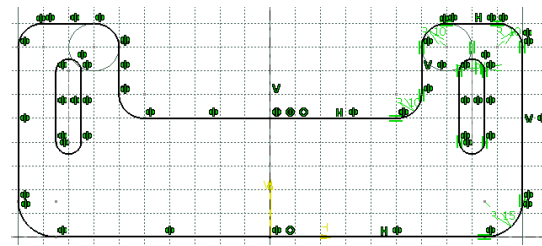


Figure 3-61 The sketch after mirroring

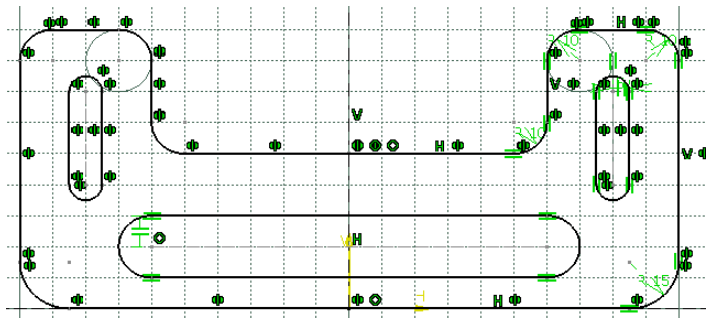



Figure 3-62 The sketch after creating the horizontal elongated hole

Saving and Closing the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.
2. Choose the **Save** button in this dialog box; the file is saved at *C:\CATIA\c03*. 
3. Close the part file by choosing **File > Close** from the menu bar.

SELF-EVALUATION TEST

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

1. You can create a parabola by using the **Parabola by Focus** tool from the **Conic** toolbar. (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 by 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 then select the sketched element, then the selected element will break at the intersection.
7. The _____ tool is also used to extend the sketched elements.
8. To offset the sketched elements, select them and then choose the _____ button from the **Transformation** sub-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** sub-toolbar and then select the element. (T/F)
4. In the **Sketcher** environment of CATIA V5, you can modify a sketched element by double-clicking on it. (T/F)
5. You can create a cylindrical elongated hole by using the **Elongated Hole** tool. (T/F)
6. Which of the following dialog boxes 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 of the following tools is used to fillet the sketched elements?
 - (a) **Fillet**
 - (b) **Corner**
 - (c) **Chamfer**
 - (d) None of these

9. Which of the following tools 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 of the following drop-downs 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)**

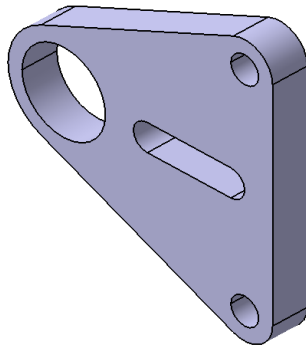


Figure 3-63 The model for Exercise 1

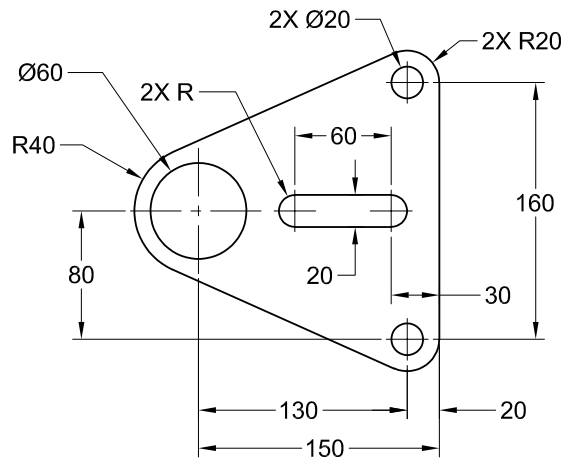


Figure 3-64 The sketch for Exercise 1

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)**

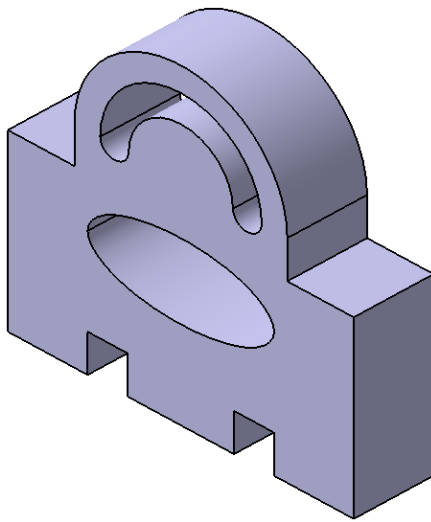


Figure 3-65 The model for Exercise 2

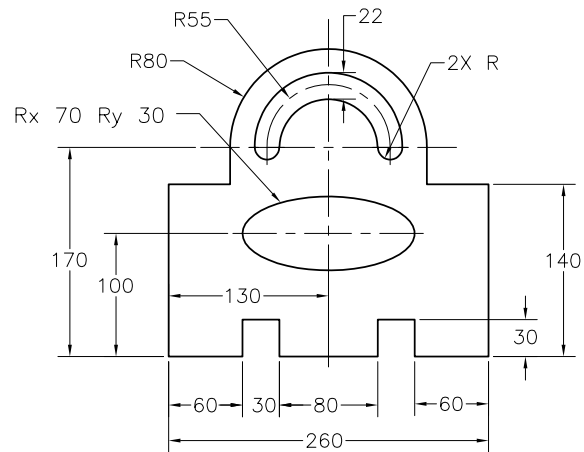


Figure 3-66 The sketch for Exercise 2

Exercise 3

Draw the sketch of the model shown in Figure 3-67. The sketch to be drawn is shown in Figure 3-68. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**

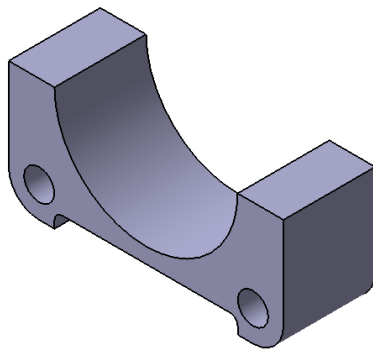


Figure 3-67 The model for Exercise 3

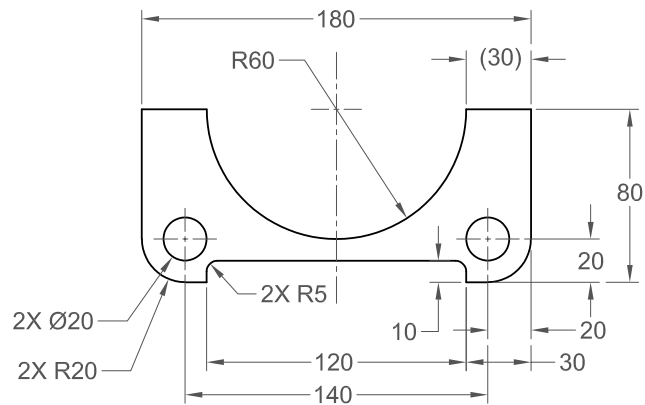


Figure 3-68 The sketch for Exercise 3

Exercise 4

Draw the sketch of the model shown in Figure 3-69. The sketch to be drawn is shown in Figure 3-70. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**

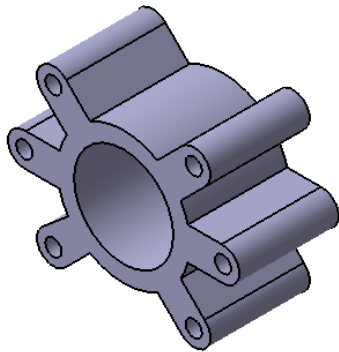


Figure 3-69 The model for Exercise 4

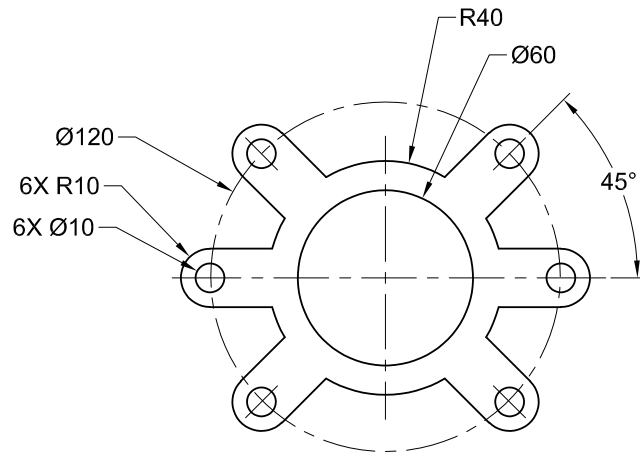


Figure 3-70 The sketch for Exercise 4

Exercise 5

Draw the sketch of the model shown in Figure 3-71. The sketch to be drawn is shown in Figure 3-72. Do not dimension the sketch. The solid model and dimensions are given only for your reference.
(Expected time: 30 min)

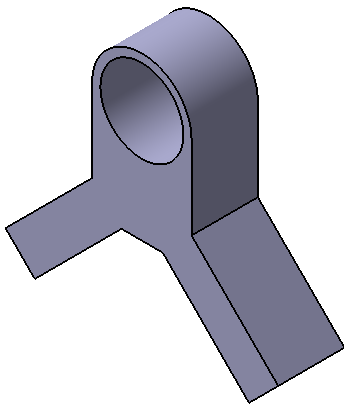


Figure 3-71 The model for Exercise 5

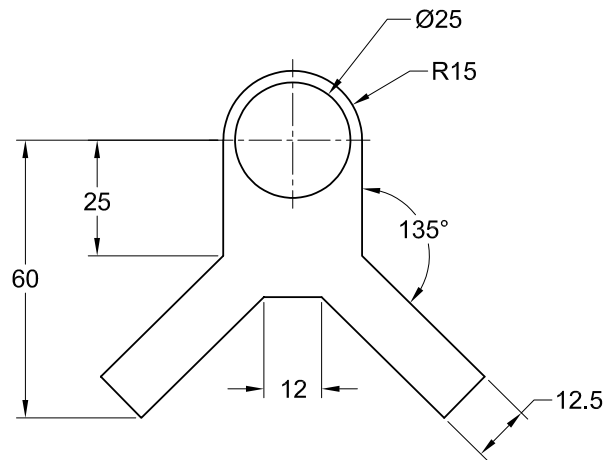


Figure 3-72 The sketch for Exercise 5

Answers to Self-Evaluation Test

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