

Chapter 3

Advance Sketching Tools

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 polygons*
- *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** tool 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.

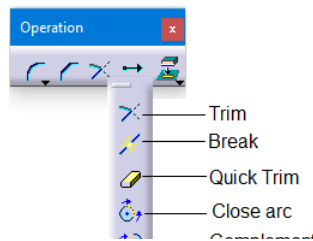


Figure 3-1 The **Profile** toolbar with the **Conic** sub-toolbar

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 the center of the ellipse; 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.

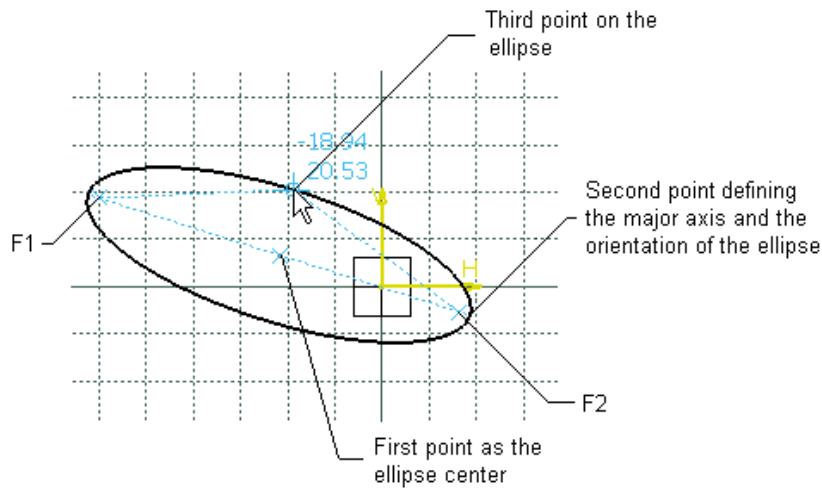


Figure 3-2 Specifying three points to draw an ellipse

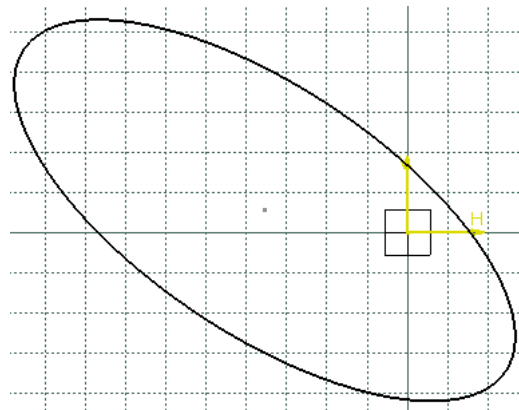


Figure 3-3 The resulting ellipse



Note

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

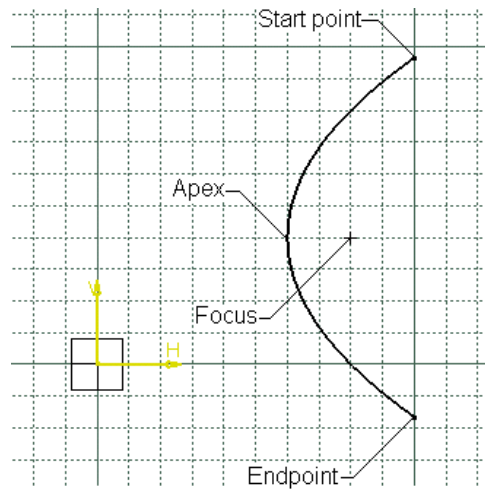


Figure 3-4 Points used to draw a parabola

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.

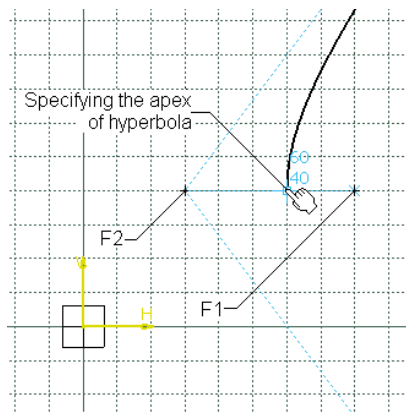


Figure 3-5 Specifying the focus and apex of the hyperbola

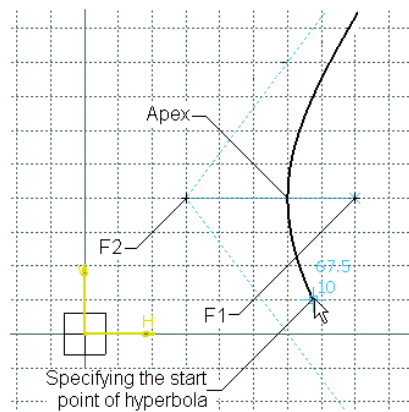


Figure 3-6 Specifying the start point of the hyperbola

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, by default the **Nearest End Point**, **Two Points**, and **Start and End Tangent** tools 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.

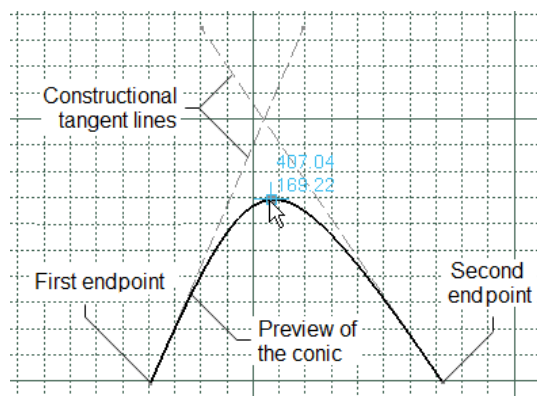


Figure 3-7 The preview of conic

Drawing Splines

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



Splines are the curves whose behavior is defined by piecewise 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.

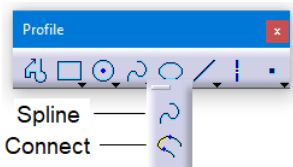


Figure 3-8 The **Profile** toolbar with the **Spline** sub-toolbar

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 and exit the **spline** tool. 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.

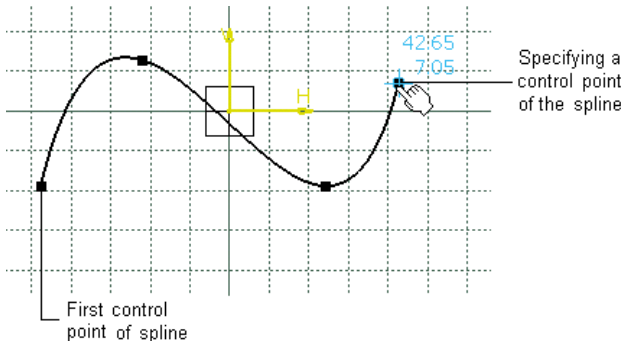


Figure 3-9 Drawing a spline by specifying multiple points

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.

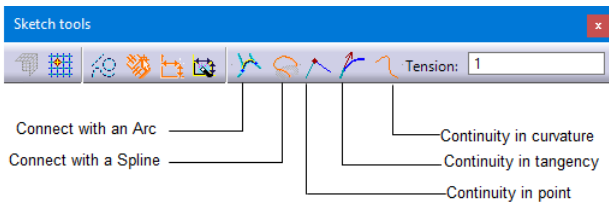


Figure 3-10 The **Sketch tools** toolbar after choosing the **Connect** tool

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** tool 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 last element. Select the last 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** tool 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** tool from the **Sketch tools** toolbar, the resulting spline will maintain a tangency continuity with the selected elements. You can also set the tension value in the **Tension** edit box.

If you choose the **Continuity in point** tool 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 element. Next, choose the required continuity tool for the second element and then select the second element. You can also set different tension values 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** tool 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 last element. Once you specify the last 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 between 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

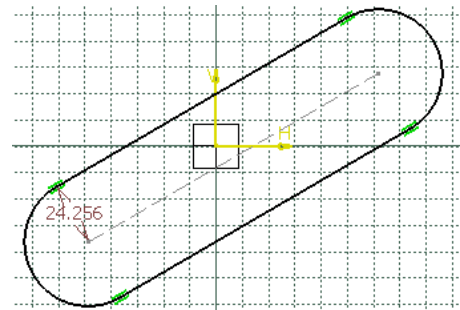


Figure 3-11 An elongated hole profile

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.



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 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 the **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 construction circle will get 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 construction arc will get 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.

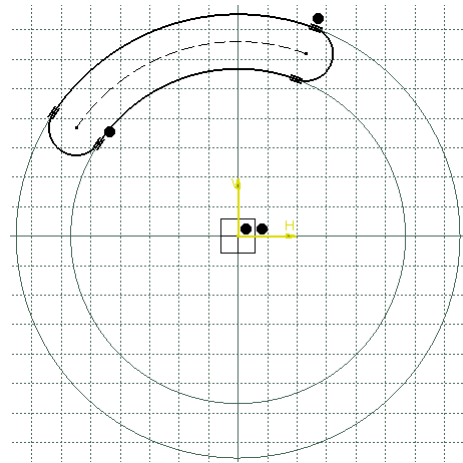


Figure 3-12 A cylindrical elongated hole profile

To draw a cylindrical elongated hole, you can also enter its parameters in various 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 red circle with a slash sign is displayed over 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 geometry 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 the 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.

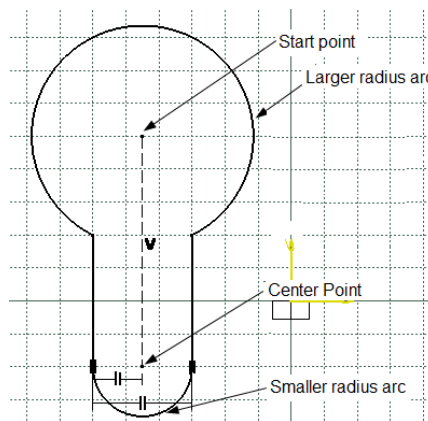


Figure 3-13 A keyhole profile

Drawing Polygons

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



To draw a polygon, choose the **Polygon** tool from the **Predefined Profile** sub-toolbar in the **Profile** toolbar; you will be prompted to select the center of the circum circle or incircle. Also, the **Sketch tools** toolbar will expand displaying the coordinates of the polygon center. You can choose the **Circumcircle** or **Incircle** tool from the **Sketch tools** toolbar to create a circum circle polygon or an incircle polygon. After defining the center of the polygon, specify the radius and angle of the polygon. Specify the number of sides in the **Number of Sides** edit box in the **Sketch tools** toolbar or it can be directly varied by moving the mouse in clockwise and anti-clockwise direction to increase or decrease it respectively; a preview of the polygon with number of sides is displayed. Note that you can create a polygon with minimum and maximum

number of sides as 3 to 24 respectively. You can also lock the number of sides of the polygon for next command by selecting the **As default number of sides** tool from the **Sketch tools** toolbar. This tool will be available on specifying the center point of the polygon. On selecting this tool the **Number of Sides** edit box becomes inactive at the pre specified value. Click in the graphics area to create the polygon. A pentagon created using the **Polygon** tool is shown in Figure 3-14.



Note

1. A circle passing through the vertices of the polygon is a **circumcircle**.
2. A circle touching all the sides of the polygon tangentially is called an **incircle**.
3. The default polygon that you can create will be a hexagon.

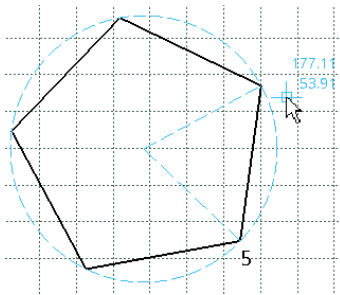


Figure 3-14 Pentagon drawn using the **Polygon** tool

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 the **Predefined Profile** sub-toolbar in the **Profile** toolbar; you will be prompted to select a point to create the center of the rectangle. Click in the geometry area to specify the point and move the cursor away from that point; a preview of the rectangle will be displayed. Now, specify a point on any corner of the previewed rectangle. Figure 3-15 shows a centered rectangle, along with its center point and the point at its corner.

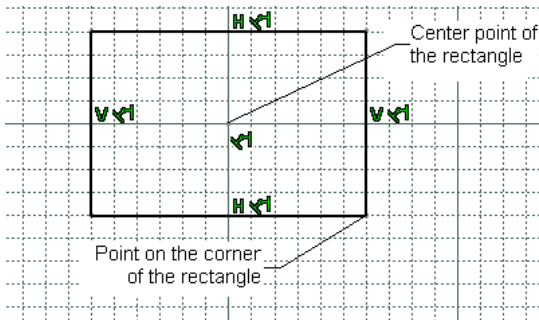


Figure 3-15 Centered rectangle along with the center point and the point at its corner

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 or construction 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 click to 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.

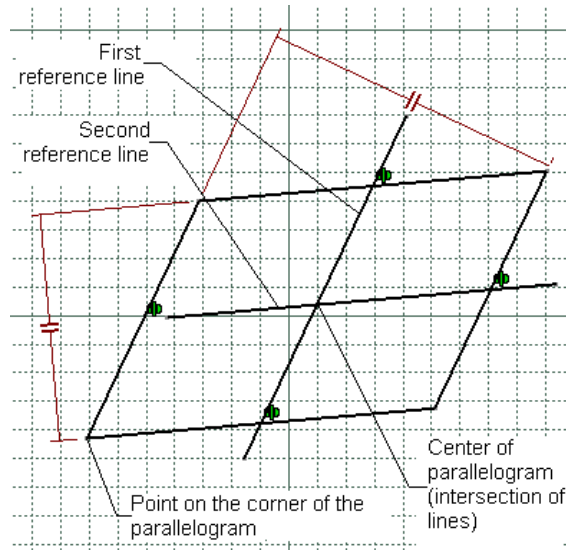


Figure 3-16 Centered parallelogram with the first and second reference lines and the point on the parallelogram



Note

After drawing the centered parallelogram, you can convert the reference lines of standard elements into construction elements by selecting them and using the **Construction/Standard Element** tool 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.

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** tool in the **Operation** toolbar. The **Relimitations** sub-toolbar is shown in Figure 3-17.

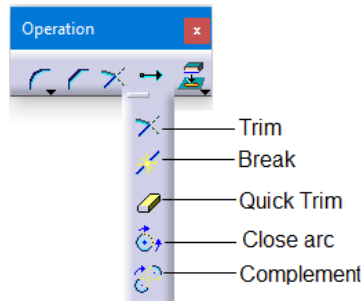


Figure 3-17 The **Operation** toolbar with the **Relimitations** sub-toolbar

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** tool 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.

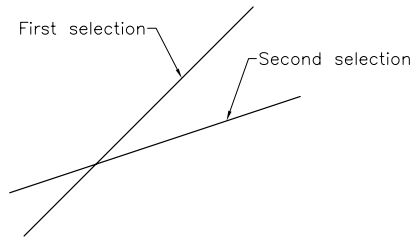


Figure 3-18 Elements to be selected for trimming

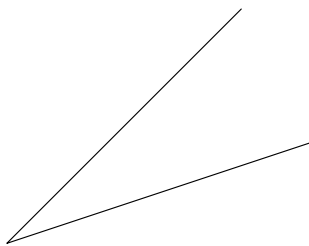


Figure 3-19 The resulting trimmed elements

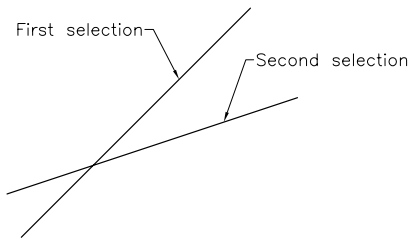


Figure 3-20 Elements to be selected for trimming

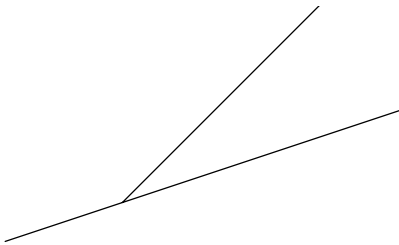


Figure 3-21 The resulting trimmed element

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. 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** tool 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.

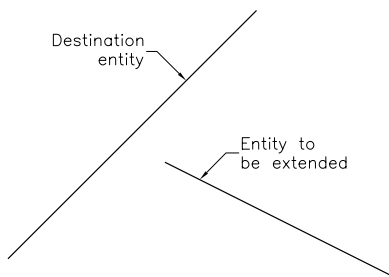


Figure 3-22 Element selected to be extended

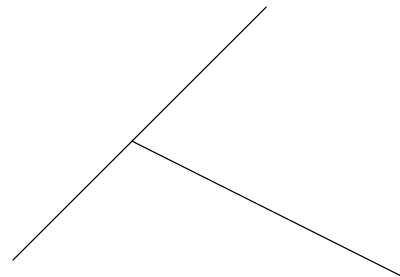


Figure 3-23 The resulting extended element

Breaking Elements

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



You can break a line or a curve at a desired position by using the **Break** tool. Different methods for breaking a line are discussed next.

To break a line at a point, choose the **Break** tool from the **Relimitations** sub-toolbar in the **Operation** toolbar; you will be prompted to select the element to be broken. Select the element to be broken; the start point and the probable break point will be displayed immediately. Next, click at the desired position to create the break point; the line will be broken at the desired point. The line is now divided into two segments and coincident constraints are created with both segments. These newly created coincident constraints make segments collinear to each other, refer to Figures 3-24 and 3-25.

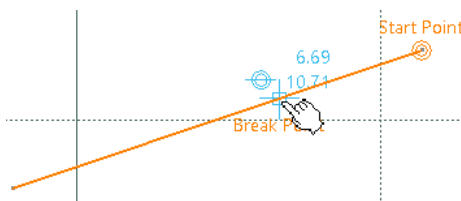


Figure 3-24 Line selected to be broken at a point

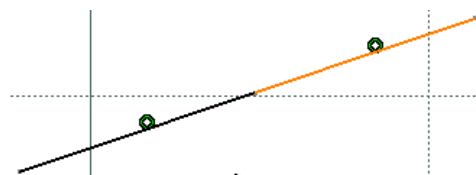


Figure 3-25 Break point with coincident constraints created

You can also break a line using a point that belongs to another line. To do so, select the line to be broken; the start point and the probable break point will be displayed immediately. Select the second line; the projection of the selected line will be displayed along with the possible constraint. The first line will be broken from the point where the projection of the second line intersects the first line. The line now comprises two segments with coincident constraint applied to them, refer to Figures 3-26 and 3-27.

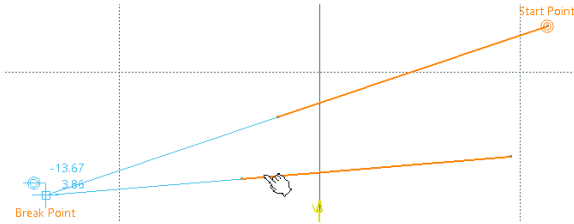


Figure 3-26 Projection of the break point with possible constraints

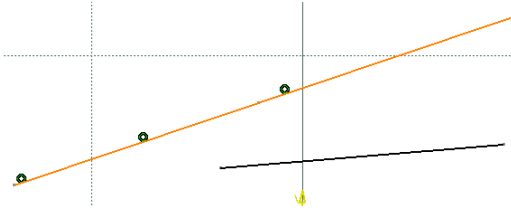


Figure 3-27 Break point created at the projection of the selected line

You can also break a line from the projection of any selected point. To do so, choose the **Break** tool and then select a line and a point; the line will be broken at the projection of the selected point. The resulting line will comprise two segments with coincident constraints applied to them, refer to Figures 3-28 and 3-29.



Note

If you do not want to consider the possible constraints while breaking the element, hold down the **SHIFT** key while selecting the second point.

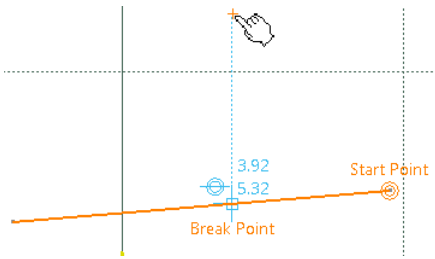


Figure 3-28 Projection of the point displaying the break point

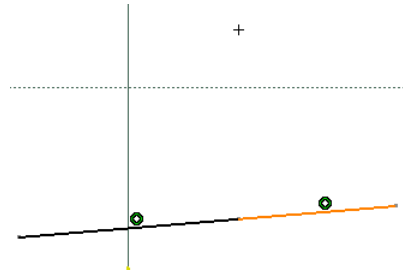


Figure 3-29 Break point with coincident constraint created

Closing Elements

Menu: Insert > Operation > Relimitations > Close arc
Toolbar: Operation > Relimitations sub-toolbar > Close arc



The **Close arc** tool is used to close trimmed circles, ellipses, or splines. To close elements, choose this tool from the **Relimitations** sub-toolbar in the **Operation** toolbar and then select one or more elements to be closed. In case of an arc, the resultant entity will be a circle and in case of a trimmed spline, the resultant entity will be the original spline.

**Tip**

You can close an arc or a trimmed circle to form a complete circle using the **Close arc** tool in the **Relimitations** sub-toolbar. Choose the **Close arc** tool from the **Relimitations** sub-toolbar and select the arc or trimmed circle to be closed. You can also close an arc or a trimmed circle by using the shortcut menu. To do so, right click to invoke the shortcut menu and then 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 into a standard element and remove its existing portion. To do so, choose the **Complement** tool from the **Relimitations** sub-toolbar and select the element. You can also use the shortcut menu to convert the complementary portion into an element, as discussed above.

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.

Filleting Sketched Elements

Menubar: Insert > Operation > Corner > Corner
Toolbar: Operation > Corner sub-toolbar > 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** tool, both the selected elements will be trimmed beyond the fillet region. This button is chosen by default. 
- the **Trim First Element** tool 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** tool, the resulting fillet will be created by retaining both the selected elements. 
- the **Standard Lines Trim** tool, 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** tool, 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** tool, the lines that extend beyond the corner will be retained as the construction elements. 

Figure 3-30 shows the elements to be selected and the fillet created using the **Trim All Elements** button. Figure 3-31 shows the fillet created using the **Trim First Element** tool and the **No Trim** tool.

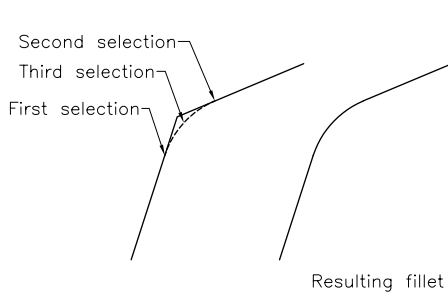


Figure 3-30 Fillet created using the **Trim All Elements** button

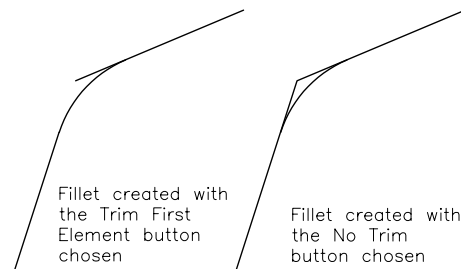


Figure 3-31 Fillets created using the **Trim First Element** and **No Trim** buttons

Creating a Tangent Arc

Menubar: Insert > Operation > Corner > Tangent Arc
Toolbar: Operation > Corner sub-toolbar > Tangent Arc



The **Tangent Arc** tool is used to create an arc tangent to a line. To create a tangent arc, choose this tool from the **Corner** sub-toolbar in the **Operation** toolbar; you will be prompted to select a curve. Select a line or curve; you will be prompted to select the end point of the corner. Select the end point of the curve; you will be prompted to locate the corner radius. Move the cursor to the desired location in the geometry area to locate the corner and click to create the tangent arc. As the tangent arc is created, rest of the line will disappear if the **Trim First Element** tool is chosen in the **Sketch tools** toolbar, as shown in Figure 3-32. To retain the rest of the line, choose the **No Trim** tool in the **Sketch tools** toolbar. If you want to keep the

same radius value while creating other corners, click **Keep as default radius for next** in the **Sketch tools** toolbar. To create an arc tangent to a line/curve upto the end point of another line/curve, select a line/curve and then select the end point of the other line/curve; the tangent arc will be created between the selected entities, as shown in Figure 3-33. Note that the options in the **Sketch tools** toolbar will be available on selecting an element.

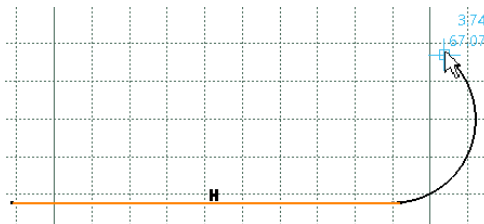


Figure 3-32 Arc created tangent to a line

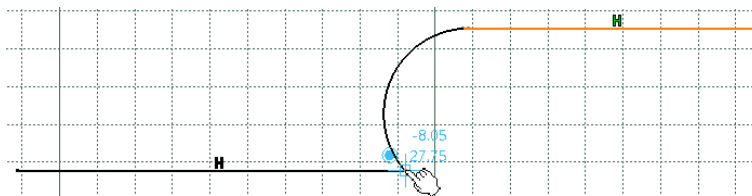


Figure 3-33 Tangent arc created upto the end point of another curve

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 3-34 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.

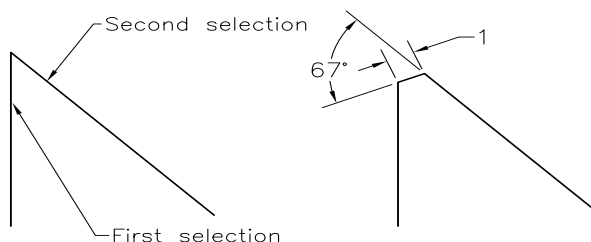


Figure 3-34 Elements to be selected and the resulting chamfer

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



If you choose the **First Length and Angle** tool, 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 tools 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** tool provided in the **Operation** toolbar; the **Transformation** sub-toolbar will be displayed, as shown in Figure 3-35. The tools in this sub-toolbar are known as the transformation tools.

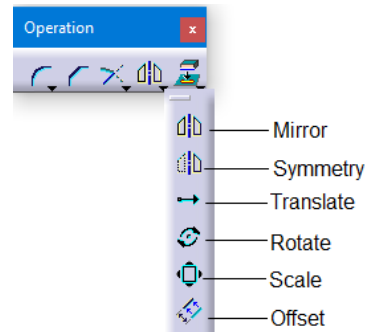


Figure 3-35 The **Operation** toolbar with the **Transformation** sub-toolbar

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** tool from the **Transformation** sub-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 3-36 shows the elements selected to be mirrored and the mirror line to be selected. Figure 3-37 shows the resulting mirrored sketch.

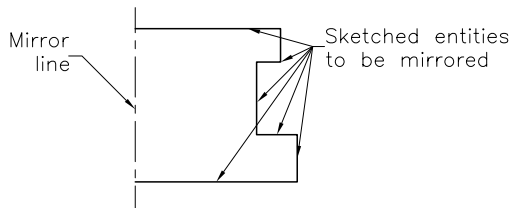


Figure 3-36 Elements selected to be mirrored and the mirror line to be selected

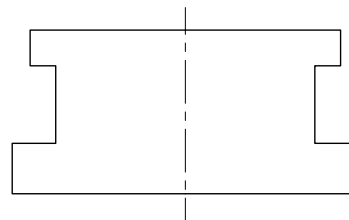


Figure 3-37 The resulting mirrored sketch

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** tool from the **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 with 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** tool 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 3-38. 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. On selecting the base point, the **Value** spinner becomes active. 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 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.

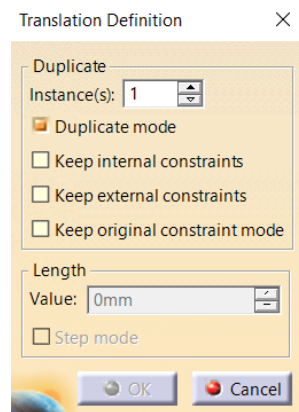


Figure 3-38 The **Translation Definition** dialog box

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** check box to keep the constraint in original mode. You will learn more about them in the later chapters.



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** tool from the **Transformation** sub-toolbar in the **Operation** toolbar; the cursor will be replaced by a point cursor and the **Rotation Definition** dialog box will be displayed, as shown in Figure 3-39. Also, you will be prompted to select the rotation center point. Click to 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. Click to specify a point; you will be prompted to select a point to define angle. As you move the cursor to specify the third point, a preview of the rotated selected elements will also be displayed. Select a point to specify the rotation angle.

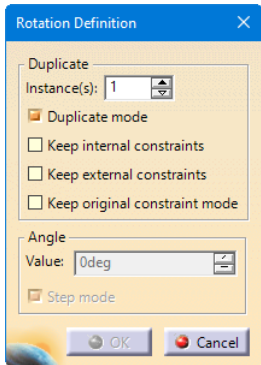


Figure 3-39 The **Rotation Definition** dialog box

Since the **Duplicate mode** check box is selected by default, another copy of the rotated element will be created. Figure 3-40 shows the points to be selected and the preview of the rotated instance of the selected elements. 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.

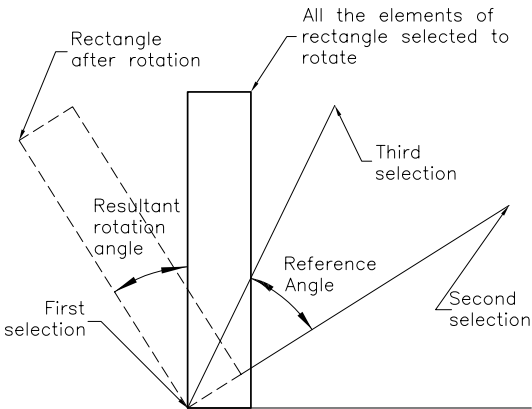


Figure 3-40 Points to be selected and the preview of the rotated elements

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** tool from **Transformation** sub-toolbar in the **Operation** toolbar; the **Scale Definition** dialog box will be displayed, as shown in Figure 3-41, 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.

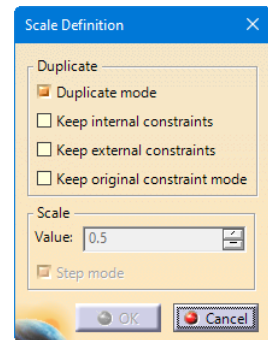


Figure 3-41 The **Scale Definition** dialog box

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** tool from the **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 to the side on 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 tools in the expanded **Sketch tools** toolbar, as shown in Figure 3-42: **No Propagation**, **Tangent Propagation**, **Point Propagation**, and **Both Side Offset** tools. These tools are used to define the elements that will be selected to offset. By default, the **No Propagation** tool is chosen. As a result, only the selected element will be offset. If you choose the **Tangent Propagation** tool, all elements that are tangent to the selected element will be automatically selected. If you choose the **Point Propagation** tool, all elements connected end to end with the selected element and forming a closed loop will be selected automatically.

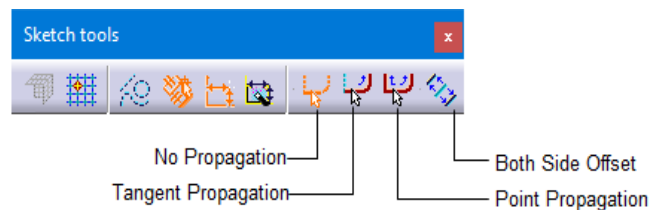


Figure 3-42 The options in the expanded **Sketch tools** toolbar

If you choose the **Both Side Offset** tool, the offset elements will be created on both sides of the selected element. Figure 3-43 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** tool is chosen. As a result, the entire closed loop is selected. Now click to create the offset of the closed loop.

Modifying the Offset Elements

You can modify the value of the offset elements using the **Profile Offset Definition** dialog box. To do so, expand the **Constraints** sub node of the Sketch node from the specification tree. Now double-click on the Profile Offset to be modified; the **Profile Offset Definition** dialog box will be displayed, as shown in Figure 3-44. You can modify the value of global offset distance in the **Value** spinner and the offset value of any particular element in the **Value** spinner from the **Local** area. The modified value of the element can be seen in the **Value** column next to the **Base** column. After modifying the parameters, choose the **OK** button from the **Profile Offset Definition** dialog box.

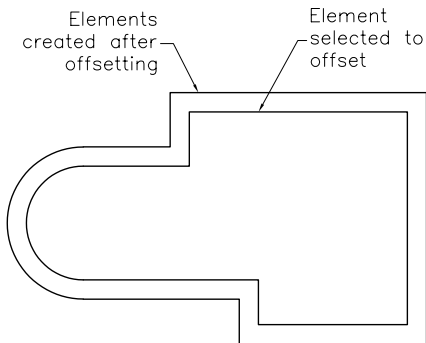


Figure 3-43 Elements created after offsetting the selected element

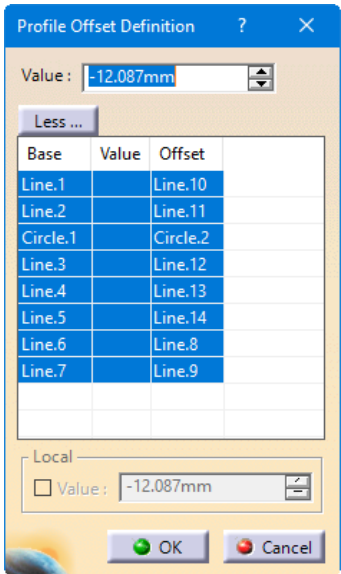


Figure 3-44 The Profile Offset Definition dialog box

Modifying Sketched Elements

In the **Sketcher** environment of CATIA V5, you can modify sketched elements by double-clicking on them. The process of modification of various 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 3-45. 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.

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 3-46. 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 into a construction element by selecting the **Construction element** check box.

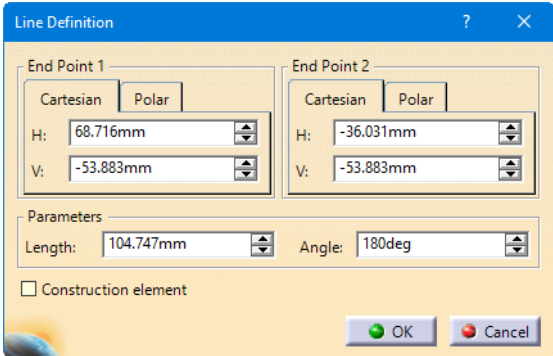


Figure 3-45 The Line Definition dialog box

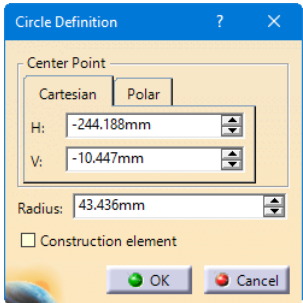


Figure 3-46 The Circle Definition dialog box

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 3-47. 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. Alternatively, in the selection area of the **Spline Definition** dialog box, select a control point after which the new control point is to be added.

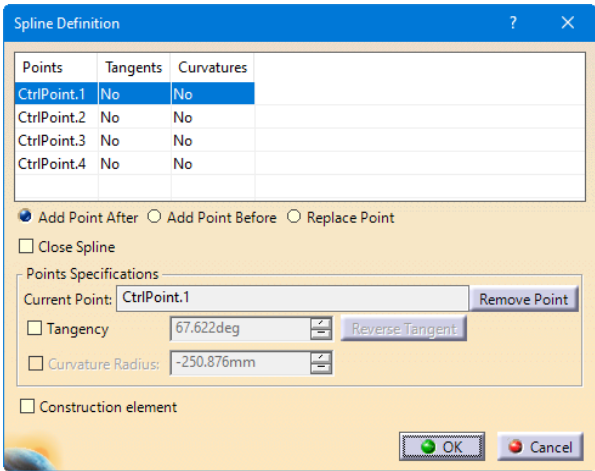


Figure 3-47 The Spline Definition dialog box

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 and curvature radius of the selected control point using the other options in this dialog box.

To change the control points of a spline, double-click on the control point that you want to edit; the **Control Point Definition** dialog box will be displayed, as shown in Figure 3-48. Also, the tangency and curvature radius manipulators will be displayed over the spline, as shown in Figure 3-49. Enter new coordinates in the **H** (horizontal) and **V** (vertical) boxes and select the **Tangency** check box to impose tangency on this control point. The tangency manipulator over the spline will be highlighted and the **Reverse Tangent** button and the **Curvature Radius** check box will get activated in the **Control Point Definition** dialog box. Specify the value of angle and radius in the **Tangency** and **Curvature Radius** edit boxes. Alternatively, you can use the manipulator to get the desired effect of angle and radius with respect to the H direction.

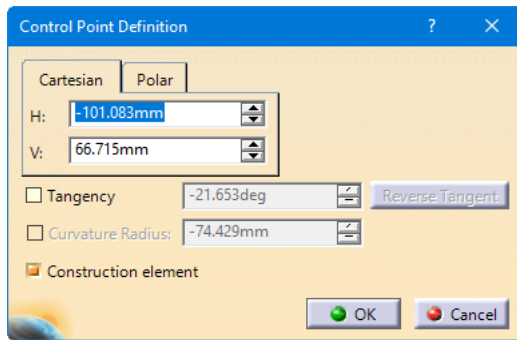


Figure 3-48 The **Control Point Definition** dialog box

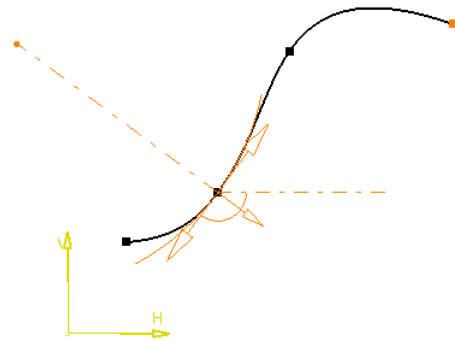


Figure 3-49 The spline with manipulators

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 3-50. You can modify the coordinates of the point using the options available in this dialog box.

Modifying the Sketched Ellipse

To modify a sketched ellipse, double-click on it; the **Ellipse Definition** dialog box will be displayed, as shown in Figure 3-51. 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.

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.

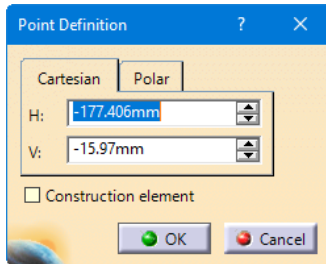


Figure 3-50 The Point Definition dialog box

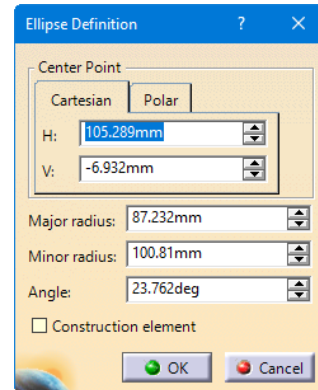


Figure 3-51 The Ellipse Definition dialog box

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.

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 3-52. Its sketch is shown in Figure 3-53. Do not dimension the sketch. The solid model and its dimensions are given only for reference. **(Expected time: 30 min)**

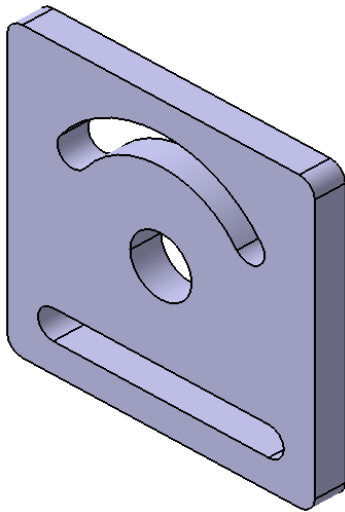


Figure 3-52 The model for Tutorial 1

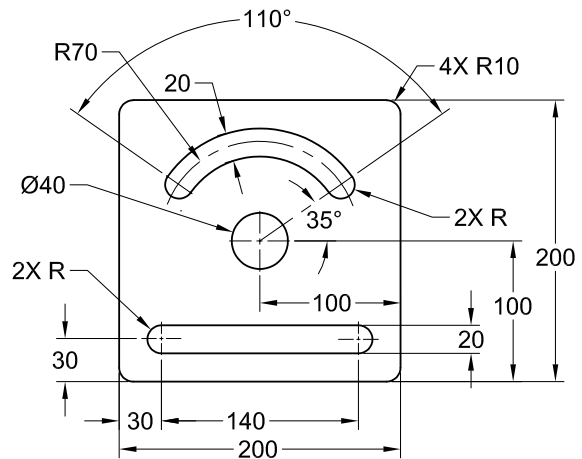




Figure 3-53 The sketch for Tutorial 1

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 fillet it using the **Corner** tool.
- Draw the inner loop of the sketch using the **Circle**, **Elongated Hole**, and **Cylindrical Elongated Hole** tools.
- Save and close the file.

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

- Choose the **New** button from the **Standard** toolbar to display the **New** dialog box. 
- Select the **Part** option from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
- 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 not already selected. Next, choose the **OK** button; a new file in the **Part** workbench is started.
- Choose the **Sketch** tool 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.

- Choose the **Zoom Out** tool from the **View** toolbar and make sure that the **Snap to Point** tool is chosen in the **Sketch tools** toolbar.
- Choose the **Centered Rectangle** tool from the **Predefined Profile** sub-toolbar in the **Profile** toolbar; you are prompted to specify a point to create the center of the rectangle.
- 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.
- 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 3-54. Now, click anywhere in the geometry area to make sure that the rectangle is no more selected.

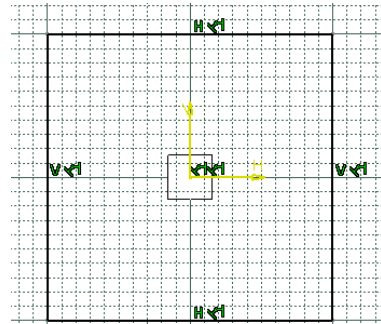


Figure 3-54 The sketch after drawing the centered rectangle

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

The vertices to be selected are shown in Figure 3-55.



Note

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

5. Choose the **Corner** tool 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. 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 the previous steps. The final outer loop of the sketch after filleting all vertices is shown in Figure 3-56.

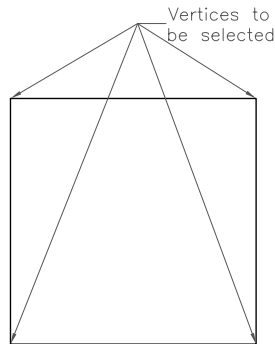


Figure 3-55 The vertices to be selected

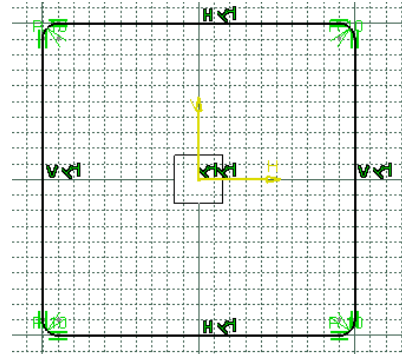




Figure 3-56 The final outer loop of the sketch


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** tool from the **Profile** toolbar; you are prompted to select a point to define the center of the circle. Click to 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 3-57. Next, you need to draw an elongated hole using the **Elongated Hole** tool.
3. Choose the **Elongated Hole** tool 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 when the radius value is displayed as **10** in the **Radius** edit box. Figure 3-58 shows 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** tool 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. Enter **70** in the **R** edit box to specify the radius at the start point of the elongated hole. Next, press the ENTER key.

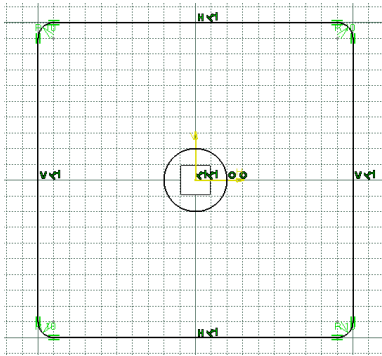


Figure 3-57 The sketch after drawing the circle

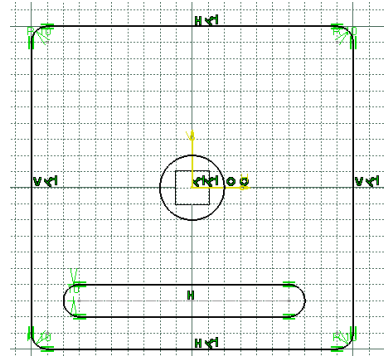


Figure 3-58 The sketch after drawing the elongated hole

10. Enter **35** in the **A** edit box and press the ENTER key.
11. Enter **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.



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.

12. Enter the value **10** in the **Radius** edit box. Next, press the ENTER key; the final sketch is created, as shown in Figure 3-59. Now, click anywhere in the geometry area to make sure that the cylindrical elongated hole is no more selected.

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.

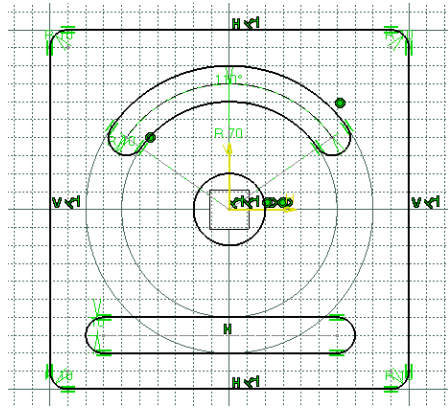


Figure 3-59 The final sketch

Tutorial 2

In this tutorial, you will draw the sketch of the model shown in Figure 3-60. The sketch is shown in Figure 3-61. Do not dimension the sketch. The solid model and its dimensions are given only for your reference.
(Expected time: 30 min)

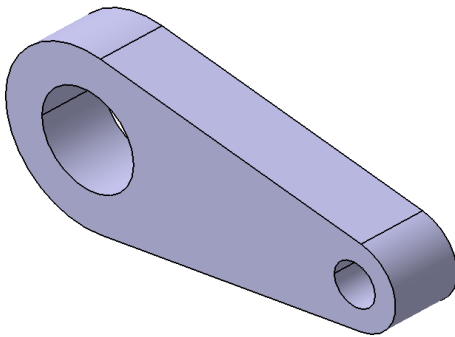


Figure 3-60 The model for Tutorial 2

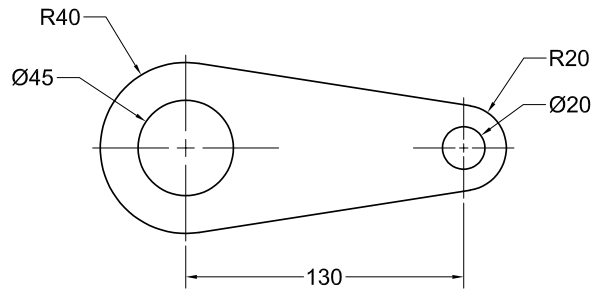




Figure 3-61 The sketch of the model for Tutorial 2

The following steps are required to complete this tutorial:

- a. 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.
- b. Trim the unwanted portion of the outer loop of the sketch using the **Quick Trim** tool.
- c. Draw the inner loops of the sketch using the **Circle** tool.
- 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 the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
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 not already selected, and then choose the **OK** button; a new **Part** file is started in the **Part** workbench.
4. Choose the **Sketch** tool 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** tool from the **Profile** toolbar; you are prompted to select a point to define the center of the circle. 
2. 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** tool is chosen in the **Sketch tools** toolbar.
3. 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 3-62.
4. Again, choose the **Circle** tool from the **Profile** toolbar. 
5. Move the cursor to a location whose coordinates are 130,0 and click to specify the center point of the circle.
6. Move the cursor toward the right and specify a 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 3-63.

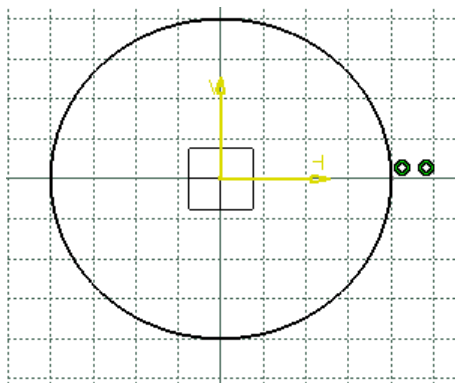


Figure 3-62 The sketch after drawing the first circle

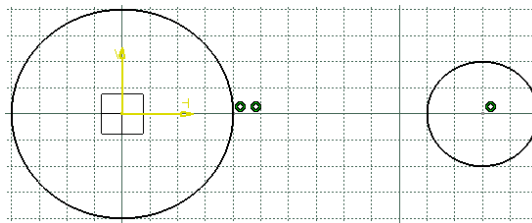


Figure 3-63 The sketch after drawing the second circle

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.

7. Choose the **Bi-Tangent Line** tool 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.

8. 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.
9. 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 3-64.
10. Similarly, draw a tangent line on the lower side of the sketch by selecting the fourth quadrants of the first and second circles. Figure 3-65 shows the sketch after drawing the second tangent line.

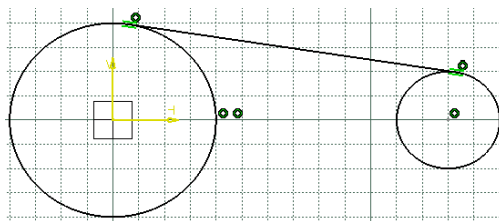


Figure 3-64 The sketch after drawing the first tangent line

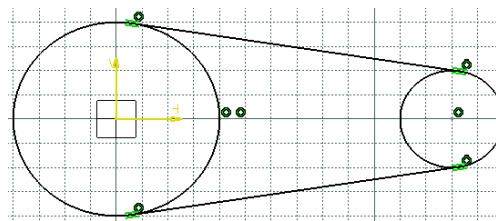



Figure 3-65 The sketch after drawing the second tangent line

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. 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. 
2. Click on the unwanted portion of the sketch, refer to Figure 3-66. The final sketch is shown in Figure 3-67.



Tip

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

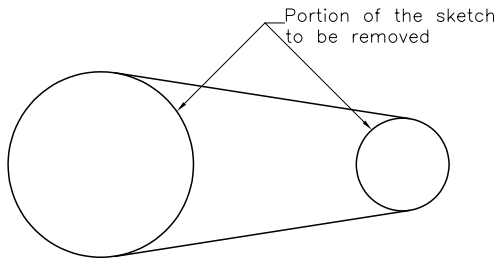


Figure 3-66 The unwanted portion of the sketch to be trimmed

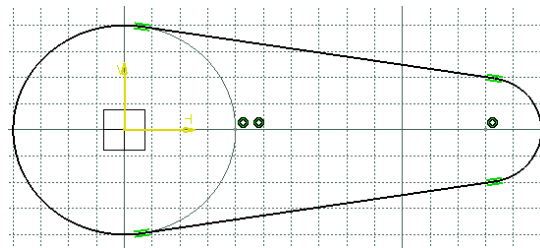



Figure 3-67 The sketch after trimming the unwanted portion

Drawing the Inner Loop of the Sketch

After drawing and trimming the outer loop of the sketch, you need to draw its inner loops consisting of two circles by using the **Circle** tool.

1. Double-click on the **Circle** tool from the **Profile** toolbar; you are prompted to select a point to define the center of the circle. 
2. 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.

3. Enter $45/2$ (radius of the circle) in the **R** edit box in the **Sketch tools** toolbar. Next, press the ENTER key; the inner circle is created.
4. As you double-clicked on the **Circle** tool, the **Circle** tool will still be 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 3-68. Press the ESC key to exit the selection set and the **Circle** tool.

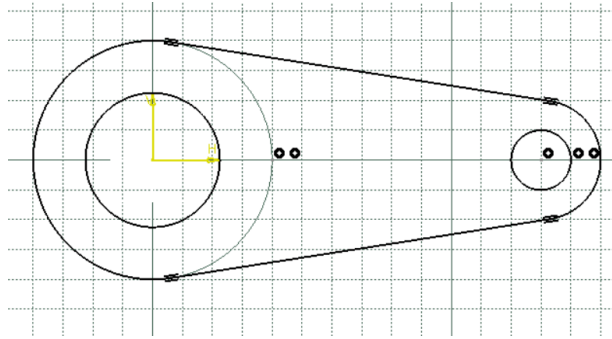


Figure 3-68 Final sketch after creating the outer and inner loops

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

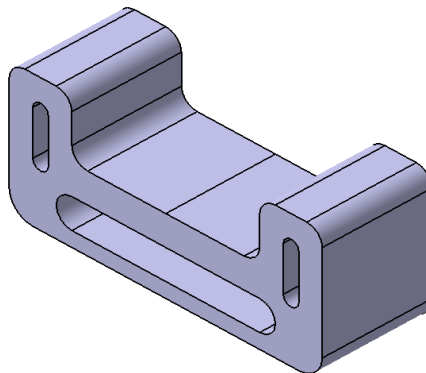


Figure 3-69 The model for Tutorial 3

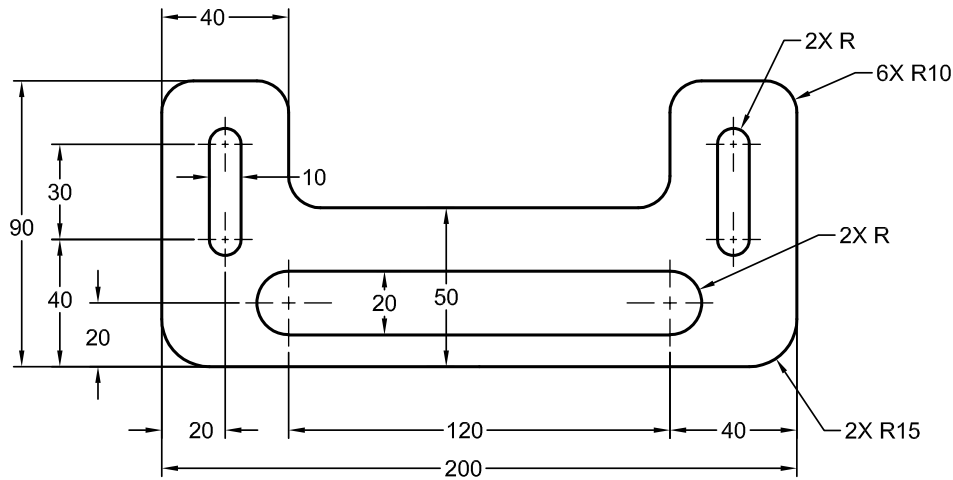


Figure 3-70 The sketch of the model for Tutorial 3

The following steps are required to complete this tutorial:

- Start a new part file in the **Part** workbench and draw the right half of the sketch using the **Profile** and **Elongated Hole** tools.
- Mirror the sketch along the vertical axis of origin.
- Draw the elongated hole in the lower portion of the sketch.
- 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 specification tree; 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; you are prompted to select the start point of the profile.
- Move the cursor to the origin and click to 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** tool 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** tool 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 3-70.
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 3-71.

Mirroring the Sketch

After drawing the right-half of the sketch, you need to mirror it using the **Mirror** tool about the vertical axis of the origin.

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** tool 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 3-72.
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 3-73. Press the ESC key to exit the selection set.

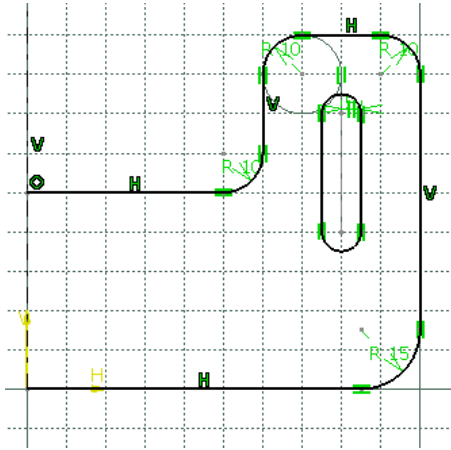


Figure 3-71 The sketch after drawing the elongated hole

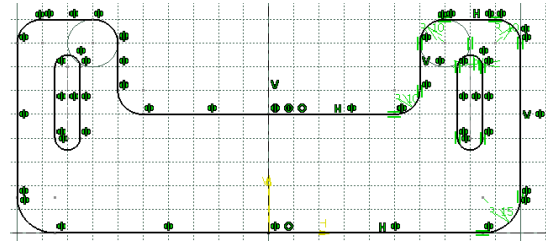


Figure 3-72 The sketch after mirroring

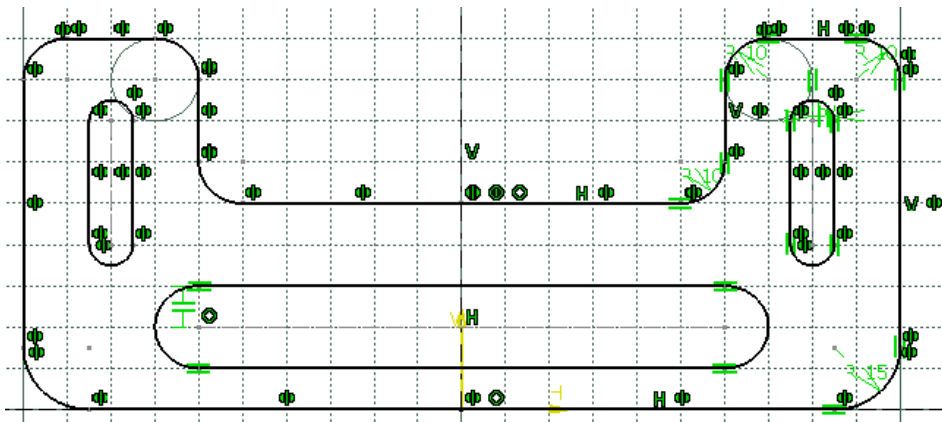


Figure 3-73 The sketch after creating the horizontal elongated hole

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. After invoking the **Quick Trim** tool if you choose the _____ button from the **Sketch tools** toolbar and then select the sketched element, the selected element will break at the intersection.
2. The _____ tool is also used to extend the sketched elements.
3. You can offset the sketched elements by selecting them and then choosing the _____ tool from the **Transformation** sub-toolbar.
4. You can modify a sketched arc using the _____ dialog box.
5. You can modify a sketched ellipse using the _____ dialog box.
6. You can create a parabola by using the **Parabola by Focus** tool from the **Conic** toolbar. (T/F)
7. In CATIA V5, you can draw a hexagon using the **Rectangle** tool. (T/F)
8. You cannot draw an n-sided polygon by using the **Profile** tool. (T/F)
9. In the **Sketcher** workbench of CATIA V5, you cannot trim the sketched elements. (T/F)
10. You can draw a key hole profile in the **Sketcher** workbench of CATIA V5. (T/F)

Review Questions

Answer the following questions:

1. 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
2. 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**
3. Which of the following tools is used to fillet the sketched elements?
 - (a) **Fillet**
 - (b) **Corner**
 - (c) **Chamfer**
 - (d) None of these

4. 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**
5. Which of the following drop-downs is used to invoke the **Keyhole Profile** tool?
 - (a) **Transformation**
 - (b) **Relimitations**
 - (c) **Operation**
 - (d) **Predefined Profile**
6. You can scale the sketched elements by selecting them and then choosing the **Translate** tool from the **Transformation** toolbar. (T/F)
7. The **Rotate** tool is used to rotate the sketched elements. (T/F)
8. You can create complementary portion of an arc or a trimmed circle by choosing the **Complement** tool from the **Relimitations** sub-toolbar and then by selecting the element. (T/F)
9. In the **Sketcher** environment of CATIA V5, you can modify a sketched element by double-clicking on it. (T/F)
10. You can create a cylindrical elongated hole by using the **Elongated Hole** tool. (T/F)

EXERCISES

Exercise 1

Draw the sketch of the model shown in Figure 3-74. The sketch to be drawn is shown in Figure 3-75. Do not dimension the sketch. The solid model and dimensions are given only for your reference.

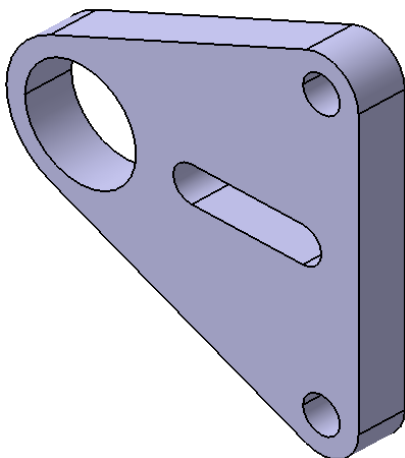


Figure 3-74 The model for Exercise 1

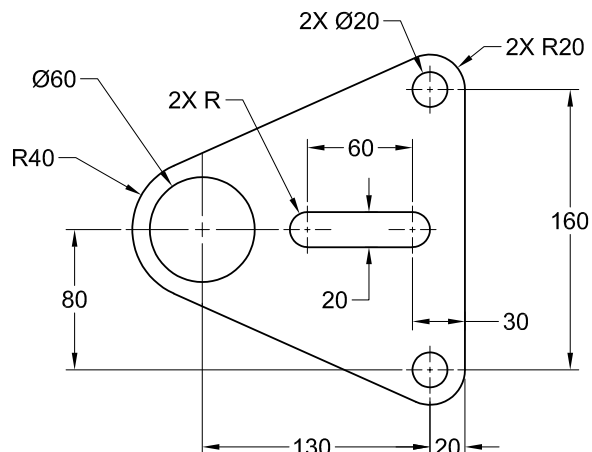


Figure 3-75 The sketch for Exercise 1

Exercise 2

Draw the sketch of the model shown in Figure 3-76. The sketch to be drawn is shown in Figure 3-77. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**

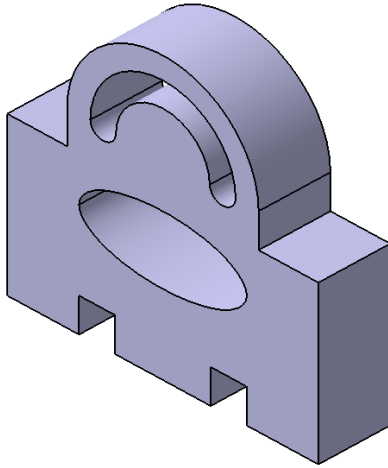


Figure 3-76 The model for Exercise 2

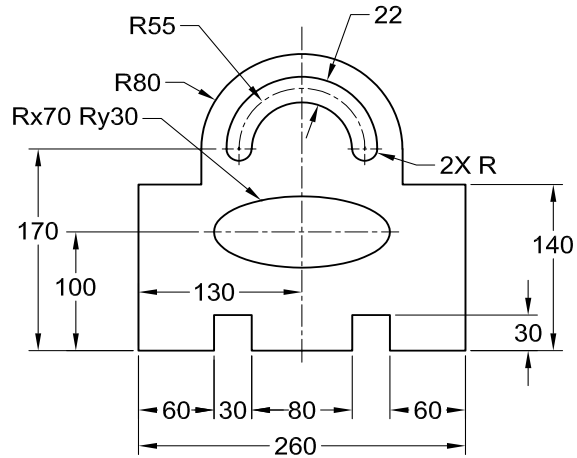


Figure 3-77 The sketch for Exercise 2

Exercise 3

Draw the sketch of the model shown in Figure 3-78. The sketch to be drawn is shown in Figure 3-79. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**

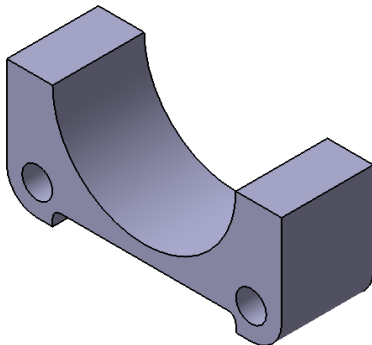


Figure 3-78 The model for Exercise 3

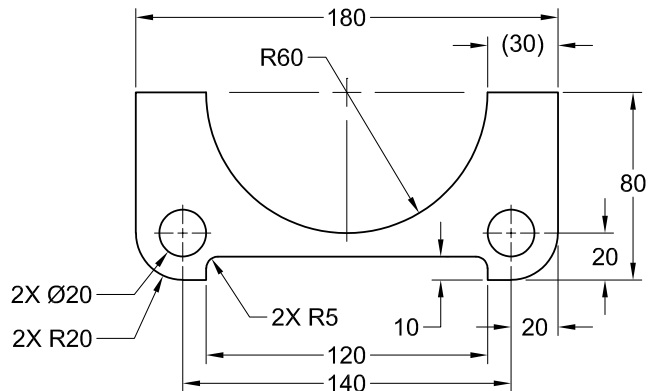


Figure 3-79 The sketch for Exercise 3

Exercise 4

Draw the sketch of the model shown in Figure 3-80. The sketch to be drawn is shown in Figure 3-81. Do not dimension the sketch. The solid model and dimensions are given only for your reference.
(Expected time: 30 min)

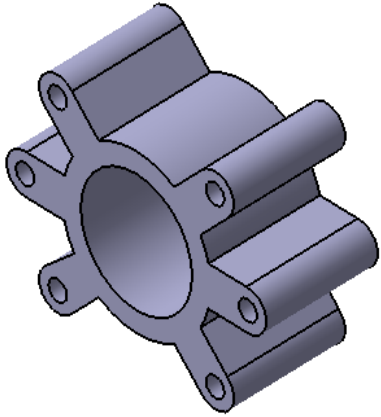


Figure 3-80 The model for Exercise 4

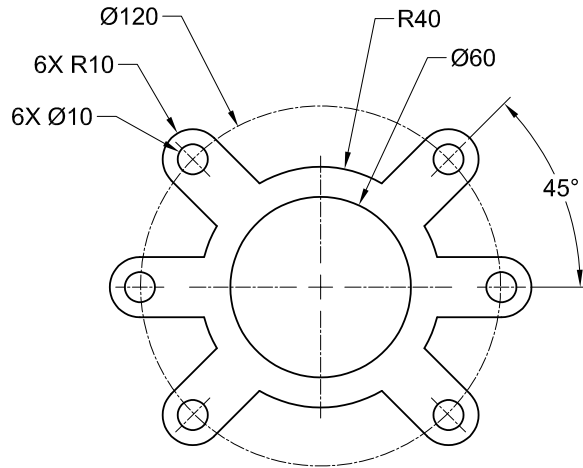


Figure 3-81 The sketch for Exercise 4

Exercise 5

Draw the sketch of the model shown in Figure 3-82. The sketch to be drawn is shown in Figure 3-83. Do not dimension the sketch. The solid model and dimensions are given only for your reference.
(Expected time: 30 min)

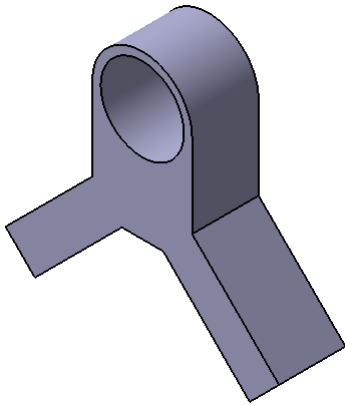


Figure 3-82 The model for Exercise 5

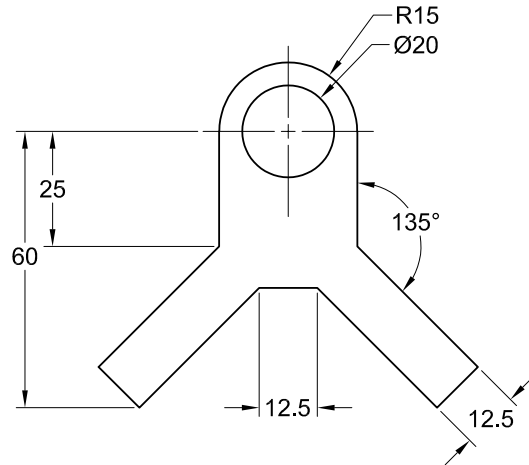


Figure 3-83 The sketch for Exercise 5

Answers to Self-Evaluation Test

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