



Chapter 2

Drawing Sketches in the Sketcher Workbench-I

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the Sketcher workbench of CATIA V5.*
- *Start a new file in the Part workbench and invoke the Sketcher workbench within it.*
- *Set up the Sketcher workbench.*
- *Understand Sketcher terms.*
- *Draw sketches using tools in the Sketcher workbench.*
- *Use some of the drawing display tools.*

THE SKETCHER WORKBENCH

Most components designed using CATIA V5 are a combination of sketched features, placed features, and derived features. The placed features are created without drawing a sketch, whereas the sketched features require a sketch that defines its shape. Generally, the base feature of any design is a sketched feature. For example, refer to the solid model of the Link shown in Figure 2-1. The base sketch to create this solid model is shown in Figure 2-2.

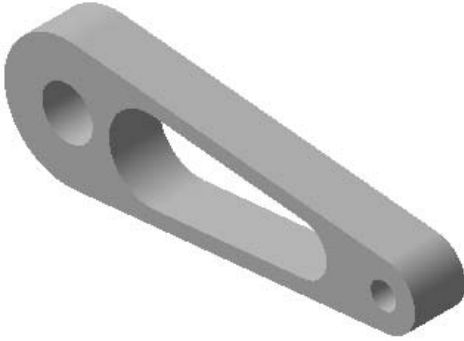


Figure 2-1 Solid model of the Link

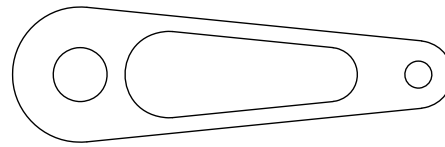


Figure 2-2 Base sketch for the solid model

The **Sketcher** workbench provides the space and tools to draw sketches of the solid model. Generally, the first sketch drawn to start the design is called the base sketch, which is then converted into a base feature. However, once you get familiar with the advanced options of CATIA V5, you will also be able to use a derived feature or a derived part as the base feature. In this chapter, you will learn more about the sketching tools in the **Sketcher** workbench that are used for drawing and displaying the sketches.

To draw a sketch, invoke the **Sketcher** workbench in the **Part Design** workbench or in the **Assembly Design** workbench by choosing the **Sketch** tool from the **Sketcher** toolbar. Next, select a plane to draw the sketch. Draw the sketch and proceed to the **Part Design** or **Wireframe and Surface Design** workbench to convert it into a solid model or a surface model.

STARTING A NEW FILE

When you start CATIA V5R20, a new **Product** file with the name **Product1** is displayed on the screen, as shown in Figure 2-3. Close this file and start a new one in the **Part Design** workbench. You will learn more about the **Product** file in the later chapters.

When you choose **Close** from the **File** menu, the start screen of CATIA V5 is displayed. Choose **Part Design** from **Start > Mechanical Design**; you will enter into the **Part Design** workbench and the **New Part** dialog box will be displayed, as shown in Figure 2-4. Enter the part name in the **Enter part name** edit box and select the **Enable hybrid design** radio button if it is not already selected. Choose **OK** to start a new file in the **Part Design** workbench. Alternatively, choose **New** from the **File** menu; the **New** dialog box will be displayed, as shown in Figure 2-5. Select **Part** from the **List of Types** list box in the **New** dialog box or write the word **Part** in the **Selection** edit box at the bottom of the **List of Types** list box. Next, choose the **OK** button; the

New Part dialog box will be displayed. Enter the file name in it and choose the **OK** button; a new file in the **Part Design** workbench will be displayed on the screen, as shown in Figure 2-6. The standard tools like the Specification tree, **Compass**, and Geometry Axes will help you complete the design. The Specification tree is displayed on the top left corner of the screen. The **Compass** is displayed on the top right corner while the Geometry Axes is displayed on the bottom right corner of the screen.

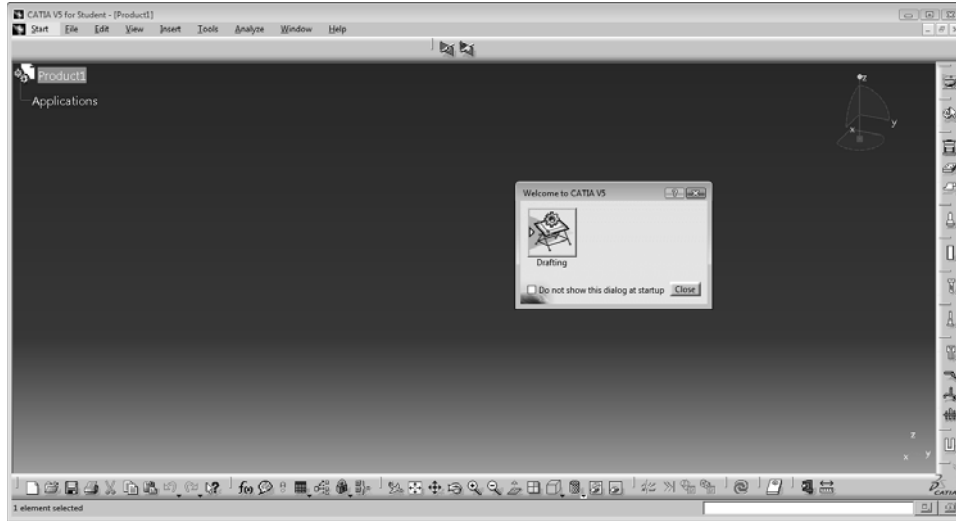


Figure 2-3 Initial screen after starting CATIA V5R20

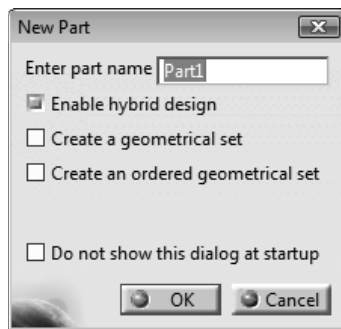


Figure 2-4 The **New Part** dialog box

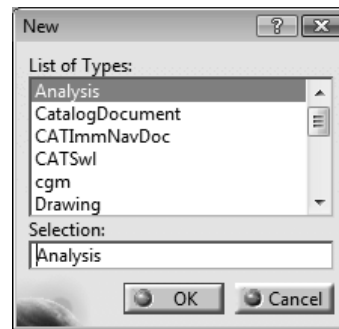


Figure 2-5 The **New** dialog box



Tip. If you clear the **Enable hybrid design** check box from the **New Part** dialog box, the new file will be started in the conventional design mode. In the earlier releases of CATIA V5, the parts were created in the conventional design mode of the **Part Design** workbench. In this textbook, the hybrid design mode has been used. Therefore, it is recommended that you keep the **Enable hybrid design** check box selected each time you start a new file.

**Note**

You can hide the **Compass**, the **Specification tree**, or the **Geometry Axes** by using the **View** menu. By default, check marks are displayed on the left of **Geometry**, **Specifications**, and **Compass** in the menu bar. This indicates that their display is turned on. Choose these options again to turn off their display. The display of these tools should be turned off only when the geometry area is too small to view the model, else it is recommended that you do not hide these standard tools. You can also use the **F3** key to toggle the display of the **Specification tree**.

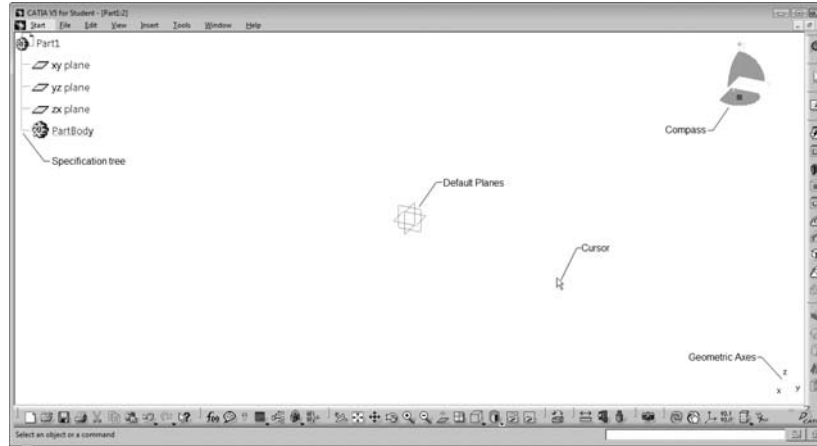


Figure 2-6 A new file opened in the **Part Design** workbench

INVOKING THE SKETCHER WORKBENCH

Sketch is the basic requirement to create the base feature of a solid model. In CATIA V5, a sketch is drawn in the **Sketcher** workbench. To invoke the **Sketcher** workbench, choose the down arrow on the lower right corner of the **Sketcher** toolbar; the **Sketcher** drop-down will appear. Figure 2-7 shows the **Sketcher** drop-down. The two buttons in the **Sketcher** drop-down are **Sketch** and **Positioned Sketch**. The next section focuses on invoking the **Sketcher** workbench using these two buttons.

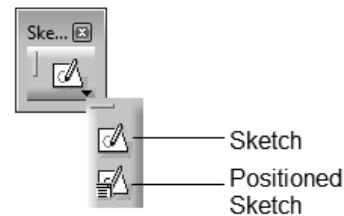


Figure 2-7 The **Sketcher** drop-down

Invoking the Sketcher Workbench Using the Sketch Tool



To invoke the **Sketcher** workbench using this method, choose the **Sketch** button from the **Sketcher** drop-down; you will be prompted to select a plane, planar face, or sketch. Select a plane from the three default planes in the **Specification tree** or from the geometry area; the selected plane will be invoked in the **Sketcher** workbench and oriented parallel to the screen, as shown in Figure 2-8. Also, you will be prompted to select an object or a command. The sketching components that are displayed in the geometry area are discussed later in this chapter.

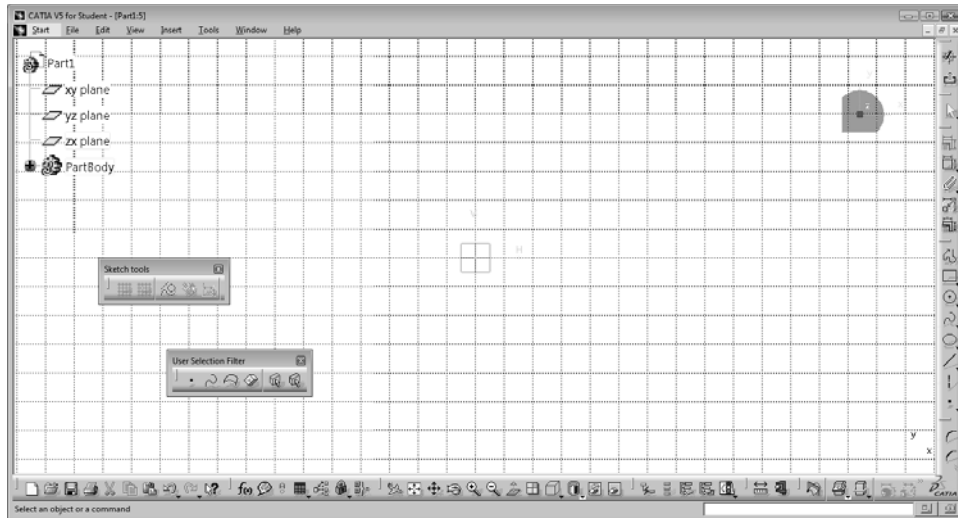


Figure 2-8 The **Sketcher** workbench invoked on selecting the **yz plane** as the sketching plane



Note

Remember that on invoking the **Sketcher** workbench, you will always be in the **Select** mode and therefore, prompted to select an object or a command. To exit the **Sketcher** workbench, choose the **Exit workbench** tool from the **Workbench** toolbar.



Invoking the Sketcher Workbench Using the Positioned Sketch Tool



In CATIA V5, you can also define a user-defined absolute axis while invoking the **Sketcher** workbench by using the **Positioned Sketch** option. To invoke the **Sketcher** workbench using this option, choose the **Positioned Sketch** tool from the **Sketcher** drop-down; the **Sketch Positioning** dialog box will be displayed, as shown in Figure 2-9. Also, you will be prompted to select a plane, a planar face, a sketch, an axis system, or two lines. You can set the absolute axis by using the options in this dialog box.

SETTING THE SKETCHER WORKBENCH

After invoking the **Sketcher** workbench, you need to set the workbench as per the sketching or drawing requirements. These requirements include modifying units, grid settings, and so on. The next section focuses on setting these parameters.

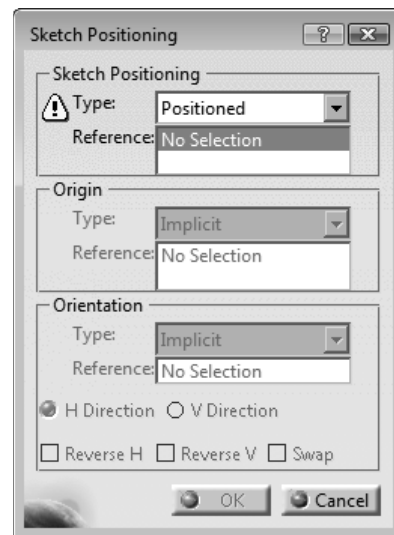


Figure 2-9 The **Sketch Positioning** dialog box

Modifying Units

To modify units, invoke the **Options** dialog box by choosing **Options** from **Tools** menu. Next, click on the + sign on the left of the **General** option to expand the tree, if it is not already expanded. Choose the **Parameters and Measure** option; the tabs corresponding to this selection appear on the right in the **Options** dialog box. Next, choose the **Units** tab. The **Options** dialog box, after choosing the **Units** tab, is shown in Figure 2-10.

You can set the units for length, angle, time, mass, and so on, by using the options in the **Units** area. After specifying the value of the units, choose the **OK** button from the **Options** dialog box.

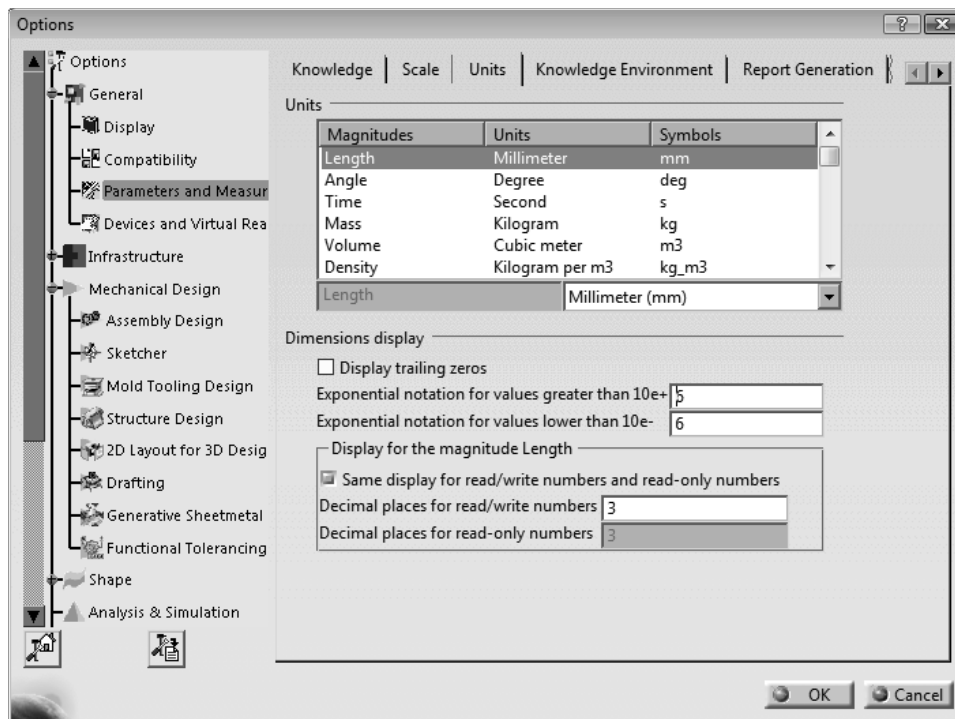


Figure 2-10 The **Options** dialog box with the **Units** tab chosen

Modifying Grid Settings

When you invoke the **Sketcher** workbench, two types of lines are displayed in the geometry area, one in the horizontal direction and the other in the vertical direction. These are continuous lines and dotted lines. The spacing between two dotted lines is called graduation and the spacing between two continuous black lines is called primary spacing. The mesh that is formed due to the intersection of these lines in the vertical and horizontal directions is called grid. In other words, primary spacing and graduation define the grid.

By default, the value of the **Graduations** parameter is set to 10 in both horizontal and vertical directions. The default value of the **Primary Spacing** parameter is 100mm. Though you can change the **Primary Spacing** and **Graduations** values in the horizontal and vertical directions individually, yet it is recommended not to change them. If the values of **Primary Spacing**

or **Graduations** in the horizontal direction are different from those in the vertical direction, then the **Grid** will be distorted. To change the values of **Primary Spacing** and **Graduations**, choose **Options** from the **Tools** menu; the **Options** dialog box will be displayed. Choose the **Mechanical Design** node from the tree on the left of the dialog box. Next, choose the **Sketcher** option to display the **Sketcher** tab on the right in the **Options** dialog box, as shown in Figure 2-11.

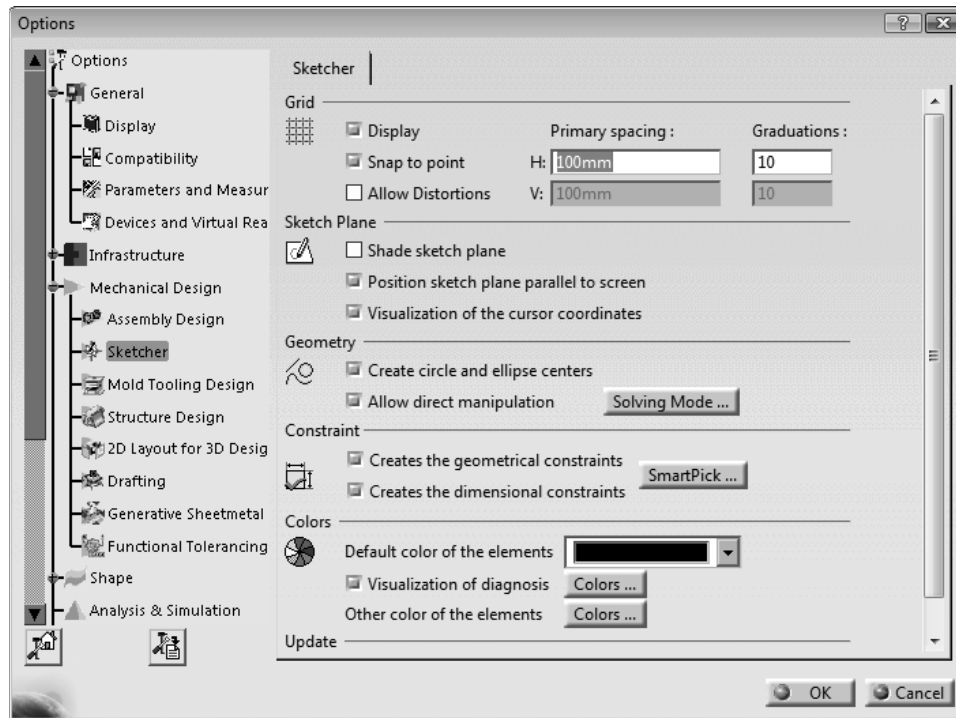


Figure 2-11 The **Options** dialog box with the **Sketcher** option chosen

The edit boxes of **Primary Spacing** and **Graduations** on the right of the **H** row are already enabled. Here, **H** refers to the horizontal direction. To enable the edit boxes of **Primary Spacing** and **Graduations** under the **V** row, select the **Allow Distortions** check box. Here, **V** refers to the vertical direction. Next, enter the values in the edit boxes corresponding to the **H** and **V** directions and then choose the **OK** button; the newly formed **Grid** will be applied to the **Sketcher** workbench. Note, all the files that you open or start in the **Sketcher** workbench, henceforth, will use these values for **Grid**.

UNDERSTANDING SKETCHER TERMS

Before learning about the sketching tools, it is important for you to understand some of the terms used in the **Sketcher** workbench. These terms are discussed next.

Specification Tree

The Specification tree is a manager that keeps track of all operations performed on the model. When you invoke the **Sketcher** workbench, a new member or branch, **Sketch.1**, is added to the Specification tree. Click on the + sign on the right of the **PartBody** to expand it; you can

view the **Sketch.1** member of the Specification tree. A + sign is associated with the **Sketch.1** on the branch. Click on this sign once to further expand the branch. Figure 2-12 shows the expanded Specification tree.

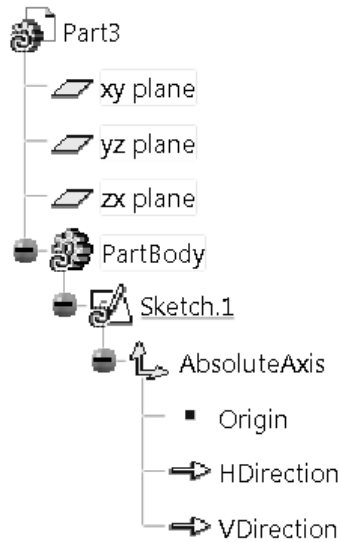


Figure 2-12 The expanded Specification tree

Various levels under **Sketch.1** in the Specification tree are discussed next.

AbsoluteAxis

In the **Sketcher** workbench, the default horizontal and vertical axes passing from the origin (0,0), to infinity are referred to as **AbsoluteAxis**. These axes will be highlighted in the geometry area, when **AbsoluteAxis** is selected from the Specification tree. Note that the + sign available on the left of **AbsoluteAxis** in the Specification tree. Click this + sign once to expand the branch by one level. The levels associated with this branch are discussed next.



Tip

While expanding the branch of the specification tree, you may accidentally click on the branch lines. This will activate the specification tree and consequently, the geometry area will be frozen. Note that the color of the default planes will turn gray. Now, zooming and panning will resize or reposition the specification tree instead of the geometry view. The geometry area can be reactivated by clicking on the branch line again or on the Geometry Axis available on the bottom right corner of the geometry area.

Origin

The **Origin** in the **Sketcher** workbench is the point where the absolute horizontal axis intersects the absolute vertical axis. The coordinates for **Origin** are (0,0). **Origin** is widely used while applying dimensional constraints to the sketches. You will learn more about dimensional constraints in later chapters.

HDirection

The direction that is parallel to the horizontal axis is represented by the **H** icon in the drawing window and is displayed as **HDirection** in the Specification tree. The **HDirection** is mostly used to constrain a sketch.

VDirection

The direction that is parallel to the vertical axis is referred to as the **VDirection** and is mostly used to constrain a sketch.

The branches of the Specification tree will increase as the design process continues. You will learn more about the branches associated with the Specification tree in the **Sketcher** workbench while drawing and constraining sketches.

Grid

This option is used to display or hide the Graduations and Primary Spacing lines from the graphic area. To activate or deactivate it, choose the **Grid** button from the **Sketch tools** toolbar, which appears only when you invoke the **Sketcher** workbench.

Snap to Point

This option is used to snap to the point of intersection of the primary spacing and the graduation lines while sketching. By default, the snap mode is active. To activate or deactivate it, choose the **Snap to Point** button from the **Sketch tools** toolbar, which appears only when you invoke the **Sketcher** workbench.

Construction/Standard Element

An element that is not a part of the profile while creating features and is used only as a reference, or to constrain the elements of the sketch in the **Sketcher** workbench, is called a **Construction** element. This element can be used only in the **Sketcher** workbench. A **Standard** element is one that takes part in the feature creation. Depending on the requirement of the design, you can convert a standard element into a construction element, or vice-versa, using the **Construction/Standard Element** button.

Select Toolbar

While drawing a sketch, you often need to select some elements. The tools that are required to make a selection are available in the **Select** toolbar, as shown in Figure 2-13. Various tools such as **Select**, **Rectangle Selection Trap**, and so on are available in this toolbar. By default, the **Select** tool is activated in the sketcher workbench unless any other tool or object is selected.

The tools in the **Select** toolbar can be invoked by choosing the down arrow on the lower-right of the **Select** toolbar. When you click on the down arrow, the **Select** drop-down will be displayed. The **Select** drop-down is shown in Figure 2-14. The methods of selecting an entity using the tools in the **Select** toolbar are discussed next.



Figure 2-13 The **Select** toolbar

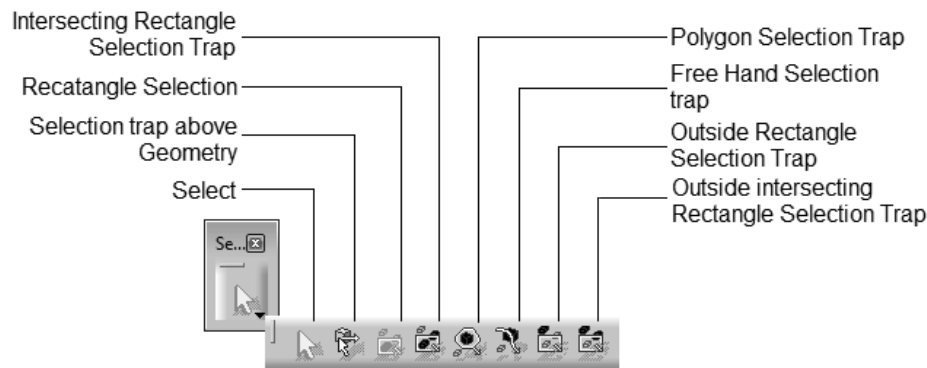


Figure 2-14 The **Select** drop-down



Note

You can detach a drop-down by dragging the vertical/horizontal line displayed at its top and place it in the drawing area. On doing so, it will act as a toolbar. Therefore, in this textbook, the drop-down will be referred by the name of the toolbar that will be obtained by detaching from the parent toolbar.

Select



This tool allows you to make a selection of the elements. As you move the arrow cursor near the element, with the **Select** tool activated, the arrow cursor will be replaced with a hand cursor. Left click on the element to select it.

Rectangle Selection Trap



This is a method of selecting entities by creating a selection trap. A trap is a rectangular box drawn by dragging the mouse to define the diagonally opposite corners. All the objects that lie completely inside the selection trap are selected. To work with this mode, choose the **Selection Trap** tool from the **Select** toolbar. Next, specify the first corner and then drag the mouse to specify the second corner.

Intersecting Rectangle Selection Trap



An intersecting trap is similar to the selection trap. The difference is that this method allows you to select elements of a sketch that are inside or are intersected by the trap. To create the intersecting trap, choose the **Intersecting Trap** tool from the **Select** toolbar. Next, specify the first corner and then drag the mouse to specify the second corner.

Polygon Selection Trap



This method includes selection of elements by drawing a closed polygon as the selection trap. You can select the elements of a sketch that are completely inside the polygon by using this method. Choose the **Polygon Trap** tool from the **Select** toolbar and draw a closed polygon by specifying its adjacent corners. The polygon creation can be terminated by double-clicking in the geometry area.

Free Hand Selection Trap



This method includes selection of elements by dragging the mouse to draw a free sketch across them. The elements intersected by the free sketch are selected.

Outside Rectangle Selection Trap



This method is used to select the elements that are outside the selection trap. The elements that are intersected by the trap are not selected.

Outside Intersecting Rectangle Selection Trap



The elements that are outside the selection trap or are intersected by the selection trap are selected by using this method.

Inferencing Lines

The inferencing lines are the temporary lines that are used to track a particular point on the screen. They are automatically displayed from the endpoints of the sketched elements or from the origin, when you select a sketching tool in the sketcher environment. Consider a case in which you want to draw a line such that its endpoint is tangent to the circle. Specify the start point of the line and then move the cursor in the direction tangent to the circle. You will note that the inference line is shown tangent to the existing circle. Next, specify the endpoint of the line. Figure 2-15 shows the use of the inferencing line to draw a tangent line. The inferencing lines are available only in the sketcher workbench.

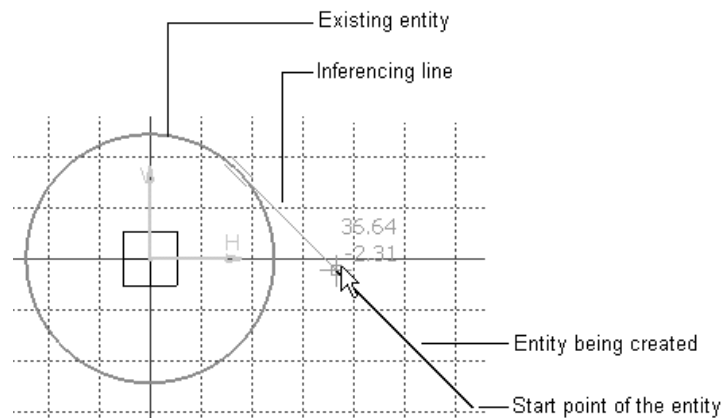


Figure 2-15 Using the inferencing line to draw a tangent line

DRAWING SKETCHES USING SKETCHER TOOLS

The tools to draw the sketches in the Sketcher workbench are available in the **Profile** toolbar. Most of the tools have a down arrow indicating that they have some more tools. To access these tools, click on the arrow and choose them from the drop-down. As mentioned earlier, the name of the drop-down will be the name of the toolbar that will be obtained by detaching it from the parent toolbar. All the sketch tools are discussed next.

Drawing Lines

Menu: Insert > Profile > Line > Line

Toolbar: Profile > Line drop-down > Line



The **Line** tool is one of the basic sketching tools in the **Sketcher** workbench. The general definition of a line is the shortest distance between two points. As CATIA V5 is parametric by nature, it allows the user to first draw a line of any length and at any angle, and then change it to the desired length and angle. To draw a line, choose the **Line** tool from the **Profile** toolbar. The two methods to draw a line in CATIA V5 are discussed next.

Drawing Lines by Specifying Points in the Geometry Area

To draw a line by specifying points in the geometry area, choose the **Line** tool from the **Profile** toolbar. You will observe that as you move the cursor in the geometry area, the coordinates corresponding to the current location of the cursor are displayed above it.

On invoking the **Line** tool, you will be prompted to select a point or click to locate the start point of the line. The prompt sequence will be displayed in the current information or dialog box area of the status bar below the geometry area. Click anywhere in the geometry area to specify the start point of the line; you will be prompted to specify the endpoint. Move the cursor away from the start point. On doing so, a rubber band line will be attached to the cursor. Click anywhere in the geometry area to specify the endpoint of the line. Figure 2-16 shows the line drawn by selecting points from the geometry area. The orange color of the line indicates that it is selected. Click anywhere on the screen to end the selection mode. You will notice that the color of the line changes to white. This indicates that it is a standard element.

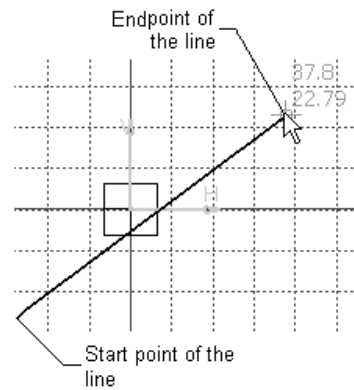


Figure 2-16 The line drawn by selecting the start and end points from the geometry area



Note

A line in CATIA V5 consists of three geometric elements: start point, line segment, and endpoint. The start point and endpoint are construction elements, while the line segment is a standard element.

Drawing Lines by Using the Sketch tools Toolbar

Lines can also be drawn using the **Sketch tools** toolbar, which expands when you invoke the **Line** tool. Figure 2-17 shows the expanded **Sketch tools** toolbar after invoking the **Line** tool. The two methods to draw a line using the **Sketch tools** toolbar are discussed next.

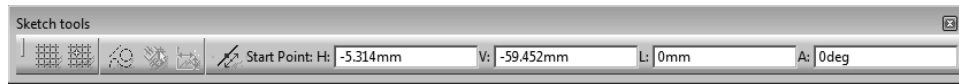


Figure 2-17 The expanded **Sketch tools** toolbar displayed after invoking the **Line** tool

Drawing Lines by Entering the Values of Start and End Points

To draw a line using the start and endpoint values, invoke the **Line** tool. The **Sketch tools** toolbar will expand. In the **Start Point H** and **V** edit boxes, specify the horizontal and vertical coordinate values of the start point, respectively, and then press ENTER; you will be prompted to enter the coordinate values for the endpoint. Specify the values in the **End Point H** and **V** edit boxes and press ENTER again; a line will be drawn in the geometry area corresponding to the value entered of the start point and endpoint. Also, the horizontal and vertical dimensions of the start point and endpoint are displayed from the origin. On completion of the line, you will observe that the **Sketch tools** toolbar is compressed to its original size after the line is drawn. The color of the created line is orange, indicating that it is selected. To end the selection mode, click anywhere in the geometry area. The line will appear green in color, which means that it is fully constrained. You will learn more about constraints in the later chapters.

Similarly, you can draw a line by specifying the start point, and entering the length and angle values. The positive angular value is measured in counterclockwise direction with respect to the H axis and the negative angular value is measured in clockwise direction with respect to the H axis.



Note

As the **Dimensional Constraint** button is chosen in the **Sketch tools** toolbar, the specified dimension values for the start and endpoint will be displayed. Let these values remain in the geometry area.

You will also notice that the color of the construction elements such as the start and endpoints of the line is gray. This suggests that the element is fully constrained.

Drawing Lines with a Symmetrical Extension



To draw a line with a symmetrical extension, invoke the **Line** tool and choose the **Symmetrical Extension** button from the expanded **Sketch tools** toolbar. When you draw the line using this option, its total length is double the distance you moved while specifying the start point and the endpoint.

In CATIA V5, a few more types of lines such as the infinite line, bisecting line, and bi-tangent line can be drawn. To draw these lines, choose the down arrow on the right of the **Line** tool from the **Profile** toolbar; the **Line** drop-down will appear, as shown in Figure 2-18. The types of lines that can be drawn using the tools available in the **Line** drop-down are discussed next.



Tip. The **Grid** button in the **Sketch tools** toolbar is used to toggle the display of the grid. While sketching, you can choose the **Grid** button any time to turn on or off the display of the grid.



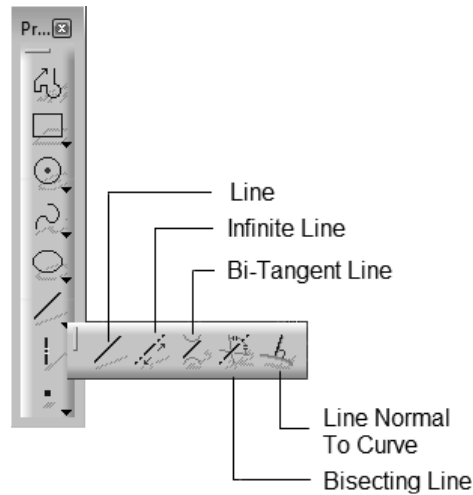


Figure 2-18 The **Line** drop-down

Drawing Infinite Lines

Menu: Insert > Profile > Line > Infinite Line

Toolbar: Profile > Line drop-down > Infinite Line



To draw an infinite line, invoke the **Infinite Line** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand. You can draw a horizontal infinite line, vertical infinite line, and infinite line passing through any two points using the options in this toolbar.

Drawing Bi-Tangent Lines

Menu: Insert > Profile > Line > Bi-Tangent Line

Toolbar: Profile > Line drop-down > Bi-Tangent Line



A line that is tangent to any two curved entities is called bi-tangent lines. The curved entities can be circle, ellipse, arc, conic and spline. You will learn more about these curved geometries later in this chapter. To draw a bi-tangent line, invoke the **Bi-Tangent Line** tool from the **Profile** toolbar. Next, select the first curved geometry and then select the second curved geometry; a line will be drawn between the two selected curved elements. Also, the tangency symbol will be displayed at the endpoints of the bi-tangent line. These are the tangent constraints. You will learn more about the tangent constraints in later chapters.

Drawing Bisecting Lines

Menu: Insert > Profile > Line > Bisecting Line

Toolbar: Profile > Line drop-down > Bisecting Line



A bisecting line pass through the intersection of two non-parallel lines such that the angle formed between them is divided equally. The intersection point of the non-parallel lines can be actual or apparent obtained by extending the lines virtually. To draw a bisecting line, invoke the **Bisecting Line** button from the **Profile** toolbar. Select the first line and then select the second line; a bisecting line of infinite length will be drawn.

**Note**

You can use the *Esc* key to exit a currently active tool.

Drawing Lines Normal to a Curve

Menu: Insert > Profile > Line > Line Normal To Curve

Toolbar: Profile > Line drop-down > Line Normal To Curve



To draw a line normal to a curve, choose the **Line Normal To Curve** tool from the **Profile** toolbar; you will be prompted to select the curve. Specify the start point of the line anywhere on the curve periphery; you will be prompted to select the other end point of the line. Select the curve; a line normal to it will be drawn.

Drawing Center Lines

Menu: Insert > Profile > Axis

Toolbar: Profile > Axis



You can draw a center line in CATIA V5 using the **Axis** tool. Generally, this tool is used to create the axis for the revolved feature. You will learn more about the revolved features in later chapters. To draw an axis, invoke the **Axis** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the start point of the axis. Click in the geometry area to specify the start point; you will be prompted to specify the endpoint of the axis. Move the cursor and click to specify the endpoint; an axis with the specified points will be displayed in the geometry area, as shown in Figure 2-19. You can also draw an axis by entering the parameters in the respective edit boxes of the expanded **Sketch tools** toolbar.

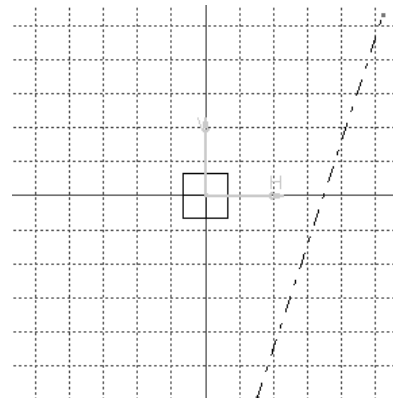


Figure 2-19 An axis drawn in the geometry area

**Note**

An axis is a construction element. Its applications are discussed in later chapters.

Drawing Rectangles, Oriented Rectangles, and Parallelograms

CATIA provides some set of tools that help you draw predefined profiles faster. These tools are grouped together in the **Predefined Profile** drop-down. To view this drop-down, choose the arrow on the right of the **Rectangle** tool in the **Profile** toolbar; the **Predefined Profile** drop-down will be displayed, as shown in Figure 2-20. The tools in this toolbar are **Rectangle**, **Oriented Rectangle**, **Parallelogram**, and so on. Some of these tools are discussed here and the remaining will be discussed in the next chapter.

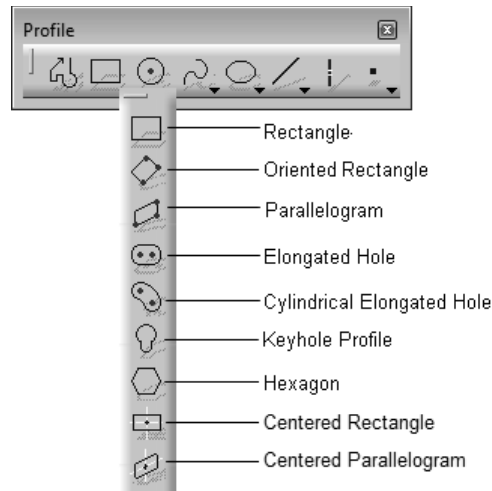


Figure 2-20 The *Predefined Profile* drop-down

Drawing Rectangles

Menu: Insert > Profile > Predefined Profile > Rectangle

Toolbar: Profile > Predefined Profile drop-down > Rectangle



To draw a rectangle, invoke the **Rectangle** tool from the **Profile** toolbar; refer to Figure 2-20. When you invoke the **Rectangle** tool, the **Sketch tools** toolbar will expand and you will be prompted to click the first point to create a rectangle. Click in the geometry area to specify the first point or the first corner of the rectangle; you will be prompted to specify the second point. Move the cursor away from the first point. You will notice that the preview of the rectangle is displayed as you move the cursor in the geometry area. Click to specify the diagonally opposite corner of the rectangle. You can also draw a rectangle by entering the values in the **Sketch tools** toolbar. On drawing a rectangle by this method, you will notice that dimensions and constraints are applied to the resulting rectangle. You will learn more about dimensioning and constraining in later chapters.



Note

The rectangle drawn in CATIA V5 is a combination of four lines and each line is an individual element.

Drawing Oriented Rectangles

Menu: Insert > Profile > Predefined Profile > Oriented Rectangle

Toolbar: Profile > Predefined Profile drop-down > Oriented Rectangle



To draw an oriented rectangle, invoke the **Oriented Rectangle** tool from the **Predefined Profile** toolbar; you will be prompted to locate the start point. Click in the geometry area to specify it; you will be prompted to locate the first side end point. Move the cursor away from the first point in any direction and specify the end point of the first side; you will be prompted to define the second side. The angle formed between the line and horizontal reference is called the orientation angle of the rectangle.

Also, the symbol of the perpendicular constraint will be displayed between the lines. You will learn more about constraints in later chapters. Click in the geometry area to specify the third corner of the rectangle. Figure 2-21 shows the oriented rectangle being drawn.

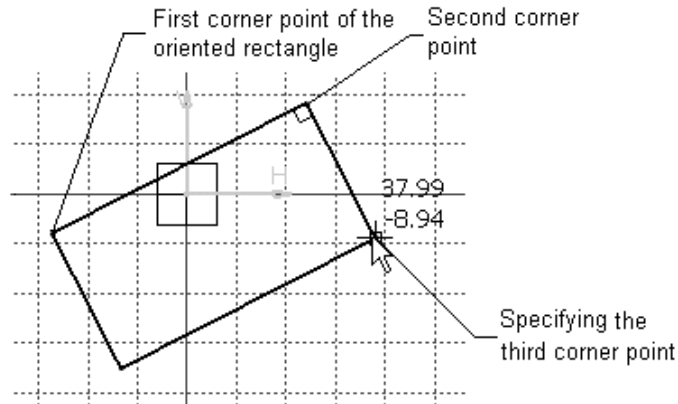


Figure 2-21 Selecting the corner points to draw the oriented rectangle



Note

You can also use the **Sketch tools** toolbar to enter the coordinate values for the first, second, and third corner in the respective edit boxes. To specify the orientation of the rectangle, enter the value of the orientation angle in the **A** edit box of the **Sketch tools** toolbar. You need to press the ENTER key, once you have entered the values.

Drawing Parallelograms

Menu: Insert > Profile > Predefined Profile > Parallelogram

Toolbar: Profile > Predefined Profile drop-down > Parallelogram



A parallelogram is a quadrilateral whose opposite sides are parallel to each other. To draw it, invoke the **Parallelogram** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the start point of the parallelogram. Click in the geometry area to specify its first corner; you will be then prompted to specify the end point of its first side. On moving the cursor away from the first corner, you will notice a line attached to the cursor. The line represents the first side of the parallelogram. Click in the geometry area to specify the endpoint of the line; you will be then prompted to specify the second side. Move the cursor away from the second corner; the preview of the parallelogram will be displayed. Click to specify the second side of the parallelogram; the parallelogram will be created, as shown in Figure 2-22.



Note

In CATIA V5, a parallelogram is a combination of four lines, where each line is an individual element. You can also use the **Sketch tools** toolbar to enter the coordinate values of the corner points of the parallelogram. You can enter the width, angle, and height values in the respective edit boxes of the expanded **Sketch tools** toolbar to specify its parameters.

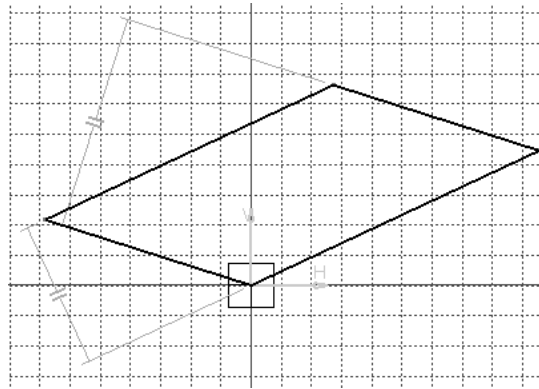


Figure 2-22 A parallelogram created by specifying the corner points

Creating Points



A point is defined as the geometrical element that has no magnitude of length, width, or thickness. It is only specified by its position. In CATIA V5, you can create points by clicking in the geometry area or by specifying the coordinates. You can also locate an intersection point or project a point on an element. To invoke any of the tools for creating a point, choose the down arrow on the right of the **Point** tool in the **Profile** toolbar; the **Point** drop-down will be displayed, as shown in Figure 2-23.

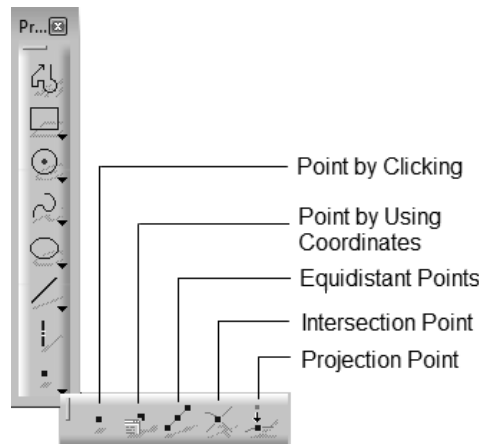


Figure 2-23 The **Point** drop-down

Creating Points by Clicking

Menu: Insert > Profile > Point > Point

Toolbar: Profile > Point drop-down > Point by Clicking



To draw points by clicking, invoke the **Point by Clicking** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to click to create the point. Click anywhere in the geometry area; a plus sign (+) will be displayed in the geometry area. You can also enter the horizontal and vertical coordinate values in the **H** and **V** edit boxes of the **Point Coordinates** area displayed in the expanded **Sketch tools** toolbar. You can create points by defining their coordinates using the other options in the **Point** toolbar. You can also create equidistant points, intersection points, and projection points using these options.

Drawing Circles

You can draw a circle in CATIA by defining its center and radius, 3 points on its periphery, by using co-ordinates and so on. All the tools to draw a circle in CATIA are grouped in the **Circle** drop-down. To view the **Circle** drop-down, choose the down arrow on the right of the **Circle**

tool in the **Profile** toolbar; the **Circle** drop-down will be displayed, as shown in Figure 2-24. The tools available in this drop-down will help you draw various types of circles and arcs. These tools are discussed next.

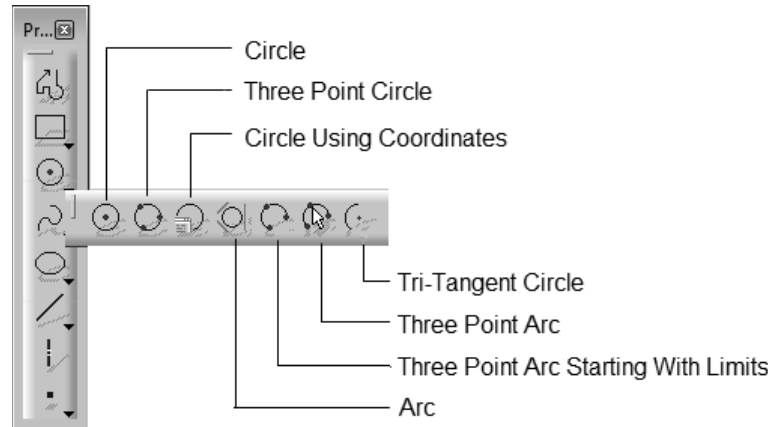


Figure 2-24 The **Circle** drop-down

Drawing Circles Using the Circle Tool

Menu: Insert > Profile > Circle > Circle

Toolbar: Profile > Circle drop-down > Circle



To draw a circle using this method, invoke the **Circle** tool from the **Circle** drop-down; you will be prompted to specify its center. Click anywhere in the geometry area to specify the center point; you will be prompted to specify a point that determines the radius of the circle. Move the cursor away from the center point to specify the radius; the preview of the circle will be displayed. Click in the geometry area to define its radius.



Note

You can also draw a circle by specifying the coordinates of its center point in the **Circle Center** edit box and the radius value in the **R** edit box of the expanded **Sketch tools** toolbar.

Drawing a Three Point Circle

Menu: Insert > Profile > Circle > Three Point Circle

Toolbar: Profile > Circle drop-down > Three Point Circle



In CATIA V5, a circle can also be drawn by specifying any three points that lie on its circumference. To draw a three point circle, invoke the **Three Point Circle** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand and you will be prompted to specify the start point of the circle. Click anywhere in the geometry area to specify the start point; you will be prompted to specify the second point through which the circle will pass. As you move the cursor away from the first point, a dotted line that originates from the first point and moves along with the cursor, will be displayed. This is the chord of the circle. Click in the geometry area to specify the second point on the circle; you will be then prompted to specify the last point. As you move the cursor to specify the third point, the preview of the circle will be displayed. Click to specify the third point to create the circle.

**Note**

*You can also enter the radius value in the **R** edit box of the expanded **Sketch tools** toolbar. Remember that when you enter the radius value, the other two points that lie on the circle should be specified within the reach of the radius value.*

Drawing Circles Using Coordinates

Menu: Insert > Profile > Circle > Circle Using Coordinates

Toolbar: Profile > Circle drop-down > Circle Using Coordinates



In CATIA V5, a circle can also be drawn by specifying the absolute coordinate values of the center and radius. To do so, invoke the **Circle Using Coordinates** tool from the **Profile** toolbar; the **Circle Definition** dialog box will be displayed. You can specify the coordinate values of the center point and radius using the options in this dialog box.

Drawing Tri-Tangent Circles

Menu: Insert > Profile > Circle > Tri-Tangent Circle

Toolbar: Profile > Circle drop-down > Tri-Tangent Circle



A tri-tangent circle is the one that is tangent to three sketched elements. The circle thus formed has a tangent relation with all the three elements. To draw it, you first need to draw three elements, which can be lines, circles, ellipses, arcs, or any geometrical element to which a circle can form a tangent relation. Next, invoke the **Tri-Tangent Circle** tool from the **Profile** toolbar and select the three elements one by one. A circle tangent to all these three elements will be displayed in the geometry area. Also, you will notice that some constraints are applied to the circle. You will learn more about them in later chapters.

**Note**

The location of the elements to be selected for creating a tri-tangent circle is important because its creation depends on the orientation of these selected elements. Also, the tangents are created as close as possible to the point where you click to select the elements. In case the element has to be extended to fulfill the need of the tangent relation, CATIA V5 will form a circle tangent at an apparent intersection.

Drawing Arcs

An arc is a geometric element that forms a sector of a circle or ellipse. Each arc must include at least two points. The tools to draw arcs are available in the **Circle** drop-down. In CATIA V5, there are three methods to draw arcs. These methods are discussed next.

Drawing Arcs by Defining the Center Point

Menu: Insert > Profile > Circle > Arc

Toolbar: Profile > Circle drop-down > Arc



To draw an arc by defining its center point, invoke the **Arc** tool from the **Profile** toolbar; you will be prompted to specify the center point. Click to specify the arc center. You are then prompted to define the radius and start point of the arc. Move the cursor away from the center point; the preview of the circle is displayed in the geometry

area. Click to specify the start point of the arc. You are then prompted to specify the endpoint of the arc. As you move the cursor, the preview of the arc is displayed. Click in the geometry area to specify the endpoint. Figure 2-25 shows an arc drawn using this method.

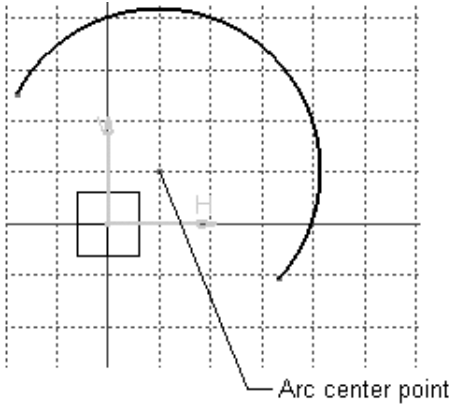


Figure 2-25 An arc

Drawing Three Point Arcs

Menu: Insert > Profile > Circle > Three Point Arc

Toolbar: Profile > Circle drop-down > Three Point Arc



To draw a three point arc, choose the **Three Point Arc** tool from the **Profile** toolbar; you will be prompted to specify the start point of the arc. Click anywhere in the geometry area to specify its start point. Next, you are prompted to select the second point through which the arc will pass. As you move the cursor away from the first point, a dotted chord is displayed. Click to specify the second point. You are prompted to specify the endpoint of the arc. The preview of the arc is displayed as you move away from the previous point. Click in the geometry area to specify its endpoint. Figure 2-26 shows the first, second, and third points being selected to draw a three point arc.

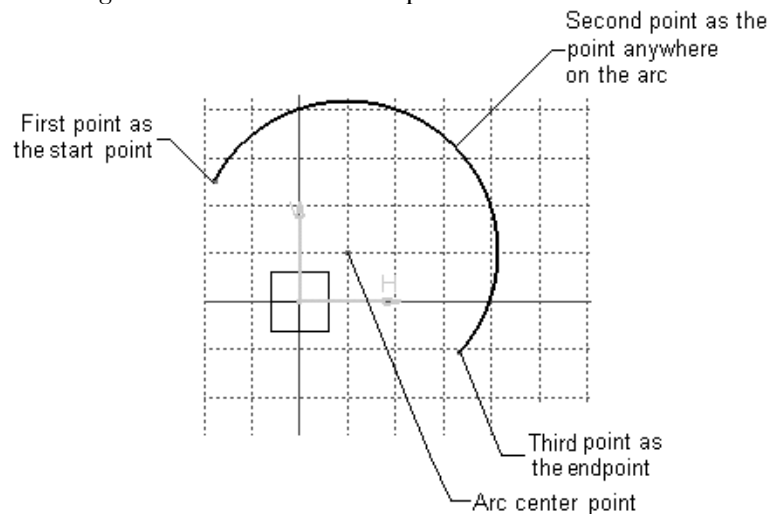


Figure 2-26 Selecting the points to draw a three point arc

Drawing Three Point Arc Starting With Limits

Menu: Insert > Profile > Circle > Three Point Arc Starting With Limits

Toolbar: Profile > Circle drop-down > Three Point Arc Starting With Limits



While drawing a three point arc starting with limits, you can specify the start and endpoint of the arc first and then the third point anywhere on it. To draw this arc, invoke the **Three Point Arc Starting With Limits** tool from the **Profile** toolbar; you will be prompted to specify the start point of the arc. Click in the geometry area to specify the start point; you will be then prompted to specify the endpoint of the arc. Move the cursor away from the start point and click to specify the endpoint; you will be prompted to specify the second point through which the arc will pass. As you move the cursor to specify this point, the preview of the arc will be displayed. Click to specify the point on the arc. Figure 2-27 shows the selection of the first, second, and third points to draw an arc using this option.

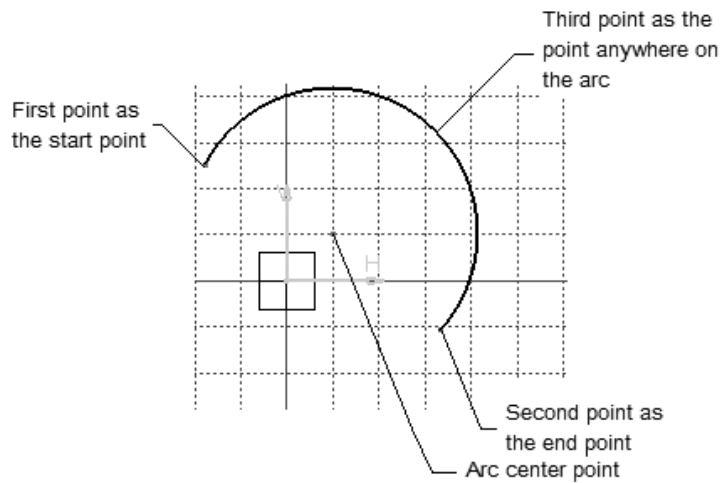


Figure 2-27 Selecting the points to draw a three point arc starting with limits

Drawing Profiles

Menu: Insert > Profile > Profile

Toolbar: Profile > Profile



In CATIA V5, a profile is defined as a combination of continuous lines and arcs. Drawing a continuous line means that the line automatically starts at the endpoint of the previous line. A profile can be an open or a closed contour. To draw the profile, invoke the **Profile** tool from the **Profile** toolbar; the **Sketch tools** toolbar will expand and the **Line** tool will be chosen in it. Also, you will be prompted to select the start point of the profile.

Click anywhere in the geometry area to specify the start point. Next, move the cursor away from the first point; a rubber-band line will be attached to the cursor with its first point fixed to the point you had specified. Click anywhere in the geometry area to specify the endpoint of the line or the second point of the profile. Move the cursor away from the second point to

draw the second line that is in continuation with the first line. You will notice that the second line originates from the endpoint of the first line. Click anywhere in the geometry area to specify the endpoint of the second line or the third point of the profile. To exit the **Profile** tool after drawing an open profile, choose the **Profile** tool again. If you draw a closed profile, you do not need to exit the **Profile** tool by choosing the **Profile** tool from the **Profile** toolbar. This tool will be automatically terminated when you specify the point to close the profile. Figure 2-28 shows an open profile.

You will notice that the expanded **Sketch tools** toolbar has three buttons: **Line**, **Tangent Arc**, and **Three Point Arc**, as shown in Figure 2-29. When you invoke the **Profile** tool, the **Line** button will be chosen by default. The profile that you have been drawing so far, using the **Profile** tool, is a combination of continuous lines. The process to draw an arc in continuation with the line using the **Profile** tool is discussed next.

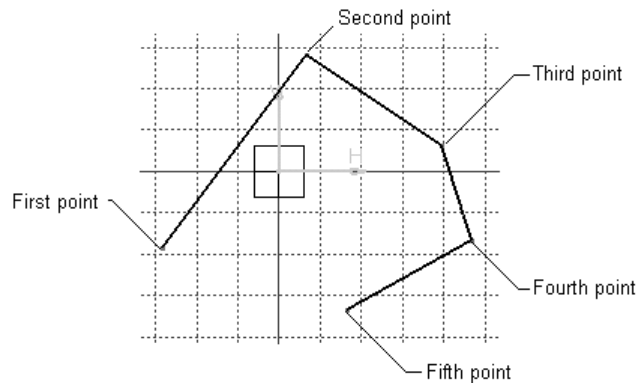


Figure 2-28 An open profile drawn using the **Profile** tool

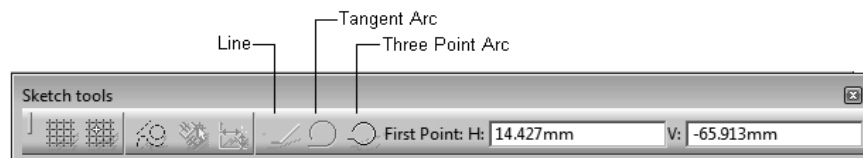


Figure 2-29 The **Sketch tools** toolbar

Drawing a Tangent Arc Using the Profile Tool

To draw a tangent arc in continuation with the line, invoke the **Profile** tool from the **Profile** toolbar. You will notice that the **Tangent Arc** button is disabled. This is because you first need to draw at least one line. After drawing the line, the **Tangent Arc** button will be enabled. Choose the **Tangent Arc** button from the expanded **Sketch tools** toolbar; the preview of the arc will be displayed in the geometry area and you will be prompted to specify the endpoint of the arc. Click in the geometry area to specify the endpoint. An arc, tangent to the line, will be drawn and displayed in the geometry area. Figure 2-30 shows a tangent arc being drawn using the **Profile** tool. After drawing the arc, the **Line** tool will again be chosen in the **Sketch tools** toolbar and you will be prompted to specify the endpoint of the current line.

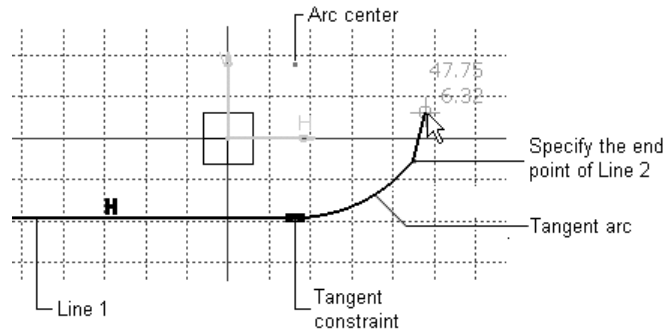


Figure 2-30 A tangent arc being drawn using the **Profile** tool



Note

You will notice a constraint applied between the line and arc. This is the tangent constraint. You will learn more about constraints in later chapters.

Drawing Three Point Arcs Using the Profile Tool

To draw a three point arc using the **Profile** tool, invoke it from the **Profile** toolbar. You will notice the **Three Point Arc** button available in the **Sketch tools** toolbar. You have two options. The first option is to draw the line and then draw the three point arc. The second option is to choose the **Three Point Arc** button first to draw a three point arc and then draw a line. Draw a line using the **Profile** tool. Now, instead of specifying the third point of the profile, choose the **Three Point Arc** button from the expanded **Sketch tools** toolbar; you will be prompted to specify the second point of the arc. Remember that the first point of the three point arc is the endpoint of the line you have drawn. Click in the geometry area to specify the second point of the arc; you will be prompted to specify the last point. Move the cursor and click to specify it; the three point arc will be displayed in the geometry area. Also, the **Profile** tool will be still active and you will be prompted to specify the endpoint of the current line. You can choose the **Profile** tool again to exit the **Profile** tool or continue with the tool by specifying more points in the geometry area.

DRAWING DISPLAY TOOLS

The drawing display tools for viewing drawing elements or geometries are available in the **View** toolbar shown in Figure 2-31. The basic tools such as **Zoom In**, **Zoom Out**, **Rotate**, **Pan**, **Normal View**, **Hide/Show**, and **Fit All In** will be discussed next. You will learn about the remaining tools in later chapters.

Fit All In

Menu: View > Fit All In

Toolbar: View > Fit All In



The **Fit All In** tool is used to increase the geometry area so that all sketched elements or geometries are included in the visible space. Note, if a drawing consists of dimensions that are beyond the visible space, invoking this tool will also include them in the visible space. You will learn more about dimensions in later chapters.

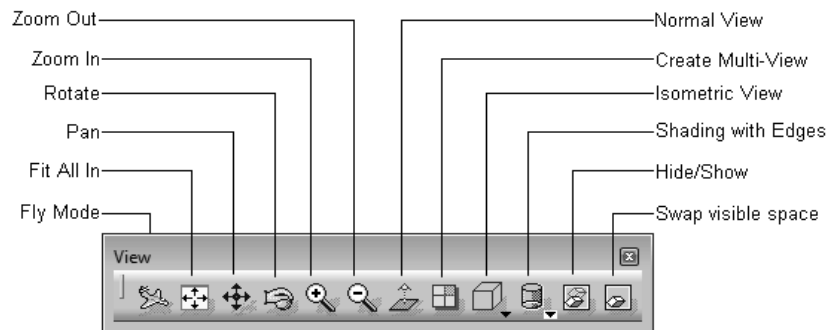


Figure 2-31 The View toolbar

Pan

Menu: View > Pan

Toolbar: View > Pan



The **Pan** tool is used to drag the current view in the geometry area. This option is generally used to display the elements or part of the elements that are outside the geometry area without actually changing the magnification of the current drawing. This is similar to holding a portion of the sketch and dragging it across the geometry area.

Zoom In

Menu: View > Modify > Zoom In

Toolbar: View > Zoom In



The **Zoom In** tool is used to zoom into the sketches in increments. Choose this button once to zoom into the sketch.

Zoom Out

Menu: View > Modify > Zoom Out

Toolbar: View > Zoom Out



The **Zoom Out** tool is used to zoom out of the sketch in increments. Choose this button once from the **View** toolbar to zoom out of the sketch.



Note

You can also dynamically zoom in or zoom out by selecting the **Zoom In Out** option from the **View** menu. To zoom in using this option, press and hold the left mouse button and then drag the mouse upward. To zoom out of the sketches, press and hold the left mouse button and then drag the mouse downward. The tool is automatically terminated once you release the mouse button.

Zoom Area

Menu: View > Zoom Area

The **Zoom Area** tool is used to define an area, which is to be magnified and viewed in the available geometry area. The area is defined using the two diagonal points of a rectangular box

in the geometry area. Press and hold the left mouse button to specify the first corner point. Then, drag the mouse to specify the other corner point of the box. The area that is enclosed inside the window will be magnified and displayed.

Normal View

Menu: View > Modify > Normal View

Toolbar: View > Normal View



The **Normal View** tool is used to orient the view normal to the sketch plane in the current **Sketcher** workbench, if its orientation is changed. If the current view is already normal to the screen, and you choose the **Normal View** button from the **View** toolbar, the viewing plane will also be reversed. In other words, on choosing this button, if the front plane is the current viewing plane, the back plane will become active for viewing.



Note

*By default, whenever you invoke the **Sketcher** workbench without defining any particular orientation, the positive horizontal reference direction points toward the right of the geometry area. Also, the positive vertical reference direction points toward its upper side. If you choose the **Normal View** button, the direction of the horizontal reference will be reversed by 180-degree. This means that the positive horizontal reference direction will point toward the left of the geometry area. Note that the vertical reference direction remains unchanged.*

*If accidentally the sketch view is rotated while working in the **Sketcher** workbench, you can choose the **Normal View** button to reorient it normal to the sketching plane.*

Splitting the Drawing Area into Multiple Viewports

Toolbar: View > Create Multi-View



The **Create Multi-View** tool is used to split the drawing area into four viewports. The model is displayed in different views in all viewports. To restore the single viewport configuration, choose this button again.

Hiding and Showing Geometric Elements

Menu: View > Hide/Show > Hide/Show

Toolbar: View > Hide/Show



This tool is used to hide sketcher elements or geometric elements from the current display. To do so, invoke the **Hide/Show** tool by choosing the **Hide/Show** button from the **View** toolbar; you will be prompted to select an element. Click on the element to be hidden from the geometry area. You will notice that the selected element is no longer visible.

Swapping Visible Space



The hidden elements are stored in a space different from the current display space. To view the space where the hidden elements are stored, invoke the **Swap visible space** tool from the **View** toolbar; you will notice that the background of the current geometry

area changes to green and only the hidden elements are visible. Invoke the **Hide/Show** tool and select the hidden elements to be redisplayed in the visible space. To return to the geometry area, choose the **Swap visible space** button again. Note that when you hide an element, only its display is turned off, but it still participates in the feature creation.

**Note**

Even if you draw a sketch in the space containing the hidden elements, it will not be visible there. It will only be displayed after you return to the visible geometry area.

You can change the standard element to a construction element in this space or vice-versa.

TUTORIALS

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 2-32. The sketch is shown in Figure 2-33. Do not dimension the sketch. The solid model and its dimensions are shown for your reference. **(Expected time: 20 min)**

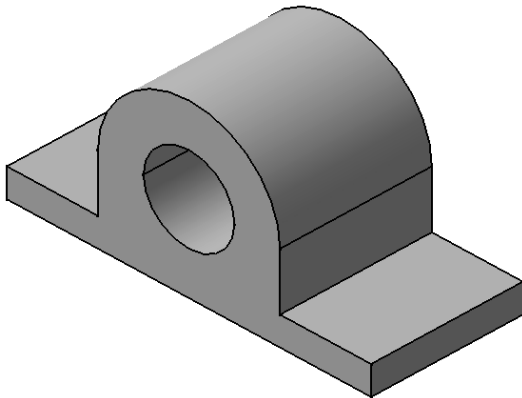


Figure 2-32 The solid model for Tutorial 1

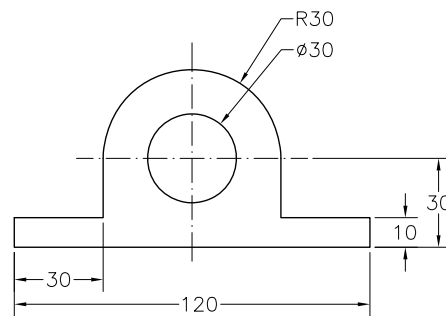


Figure 2-33 The sketch for the solid model

The following steps are required to complete this tutorial:

- Start CATIA V5 and then start a new Part file.
- Draw the sketch of the model using the **Profile** and **Circle** tools, refer to Figures 2-36 and 2-37.
- Save the sketch and close the file.

Starting a New Part File

- Start CATIA V5 by choosing **Start > All Programs > CATIA > CATIA V5R20** or by double-clicking on the shortcut icon of **CATIA V5R20** on the desktop of your computer; a new file, **Product1** is started.

**Note**

On starting CATIA V5, if the *Welcome to CATIA V5* dialog box is displayed, select the **Do not show this dialog box at startup** check box and then choose the **Close** button; a new file, *Product1* is started.

2. Choose **Close** from the **File** menu to close the **Product1** file. Next, choose **Start > Mechanical Design > Part Design** from the menu bar; the **New Part** dialog box is displayed, as shown in Figure 2-34.
3. Enter *c02tut1* as the name of the file in the **Enter part name** edit box.
4. Select the **Enable hybrid design** check box from the **New Part** dialog box, if it is not selected.
5. Choose the **OK** button; a new file in the **Part Design** workbench is started.
6. Choose the **Sketch** tool from the **Sketcher** toolbar and then select the *yz* plane from the Specification tree as the sketching plane; the **Sketcher** workbench screen is displayed, as shown in Figure 2-35.

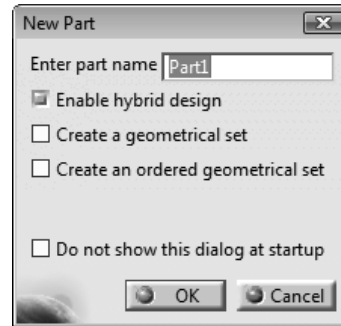


Figure 2-34 The *New Part* dialog box

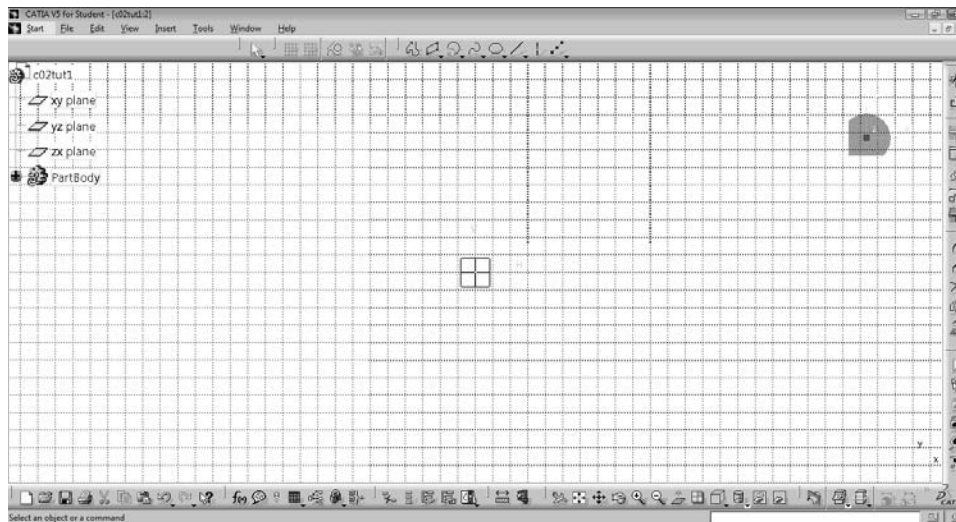




Figure 2-35 The *Sketcher* workbench screen

In this Tutorial, you need to draw the sketch in two parts: first as the outer loop and second as the inside circle in the **Sketcher** workbench.

Drawing the Outer Loop of the Sketch

In this section, first you need to draw the outer loop of the sketch using the **Profile** tool. In the sketch, the lower left corner of the sketch will be coincident with the origin of the **Sketcher** workbench. The resulting sketch will be in the first quadrant.

1. Choose the **Profile** tool from the **Profile** toolbar. 
2. Choose the **Snap to Point** button from the **Sketch tools** toolbar, if it is not chosen. 
3. Move the cursor to the location whose coordinates are 0,0 (at the origin) and specify the start point of the line. Note that the coordinates of the point are displayed on the cursor.
4. Move the cursor horizontally toward the right, the color of the line turns blue. Specify the endpoint of the line where the coordinates are 120,0.


**Note**

The change in the color of the line to blue implies that it is constrained. A constrained line may be horizontal or vertical, depending upon the direction in which the line is being drawn.

All constraints that are automatically applied to the drawn sketch will not be explained in this tutorial. You will learn about them in later chapters.

5. Move the cursor vertically upward and specify the endpoint of the line when the value of the coordinates is 120,10.
6. Move the cursor horizontally toward the left and specify the endpoint of the line when the value of the coordinates is 90,10.
7. Move the cursor vertically upward and specify the endpoint of the line when the value of the coordinates is 90,30.

After drawing these four lines, you need to draw the tangent arc using the **Tangent Arc** button.

8. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch to the **Tangent Arc** mode. 
9. Move the cursor to the location whose coordinates are 30, 30 and specify the endpoint of the tangent arc at that location. Note that, after specifying the endpoint of the tangent arc, the **Line** mode is activated and the line is attached to the cursor again.

**Note**

While drawing an arc, you will notice that the inferencing lines are displayed in the geometry area. These lines indicate the relations that they can have with other entities.

10. Move the cursor vertically downward and specify the endpoint of the line when the value of the coordinates is 30,10.
11. Move the cursor horizontally toward the left and specify the endpoint of the line when the value of the coordinates is 0,10.

12. Move the cursor vertically downward and specify the endpoint of the line such that the endpoint is coincident to the start point of the first line, and then press ESC.

The sketch after drawing the outer loop is shown in Figure 2-36. In this figure, the constraints are hidden for better visualization.

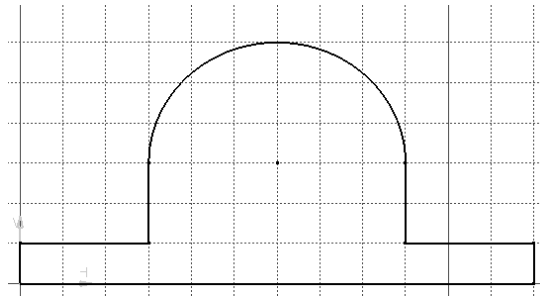


Figure 2-36 The sketch after drawing the outer loop

Drawing the Inner Loop of the Sketch

The inner loop of the sketch consists of a circle. You need to draw the circle using the **Circle** tool such that it is concentric to the arc of the outer loop.

1. Choose the **Circle** button from the **Profile** toolbar.
2. Move the cursor to the center point of the circular arc and specify the center point of the circle.
3. Enter **15** as the radius value of the circle in the **R** edit box provided in the **Sketch tools** toolbar and press ENTER.



The final sketch after drawing the inner loop is shown in Figure 2-37. Note that in this figure, the display of constraints is turned on.

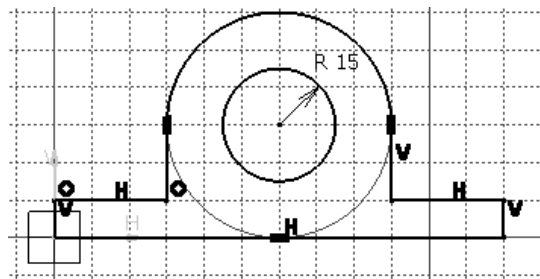


Figure 2-37 The final sketch for Tutorial 1

Saving the Sketch

After completing the sketch, you need to save it.

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



box. Using this dialog box, create the *CATIA* folder inside the *C:* drive. Then create the *c02* folder inside the *CATIA* folder.

2. Next, choose the **Save** button; the file is saved at *C: \CATIA\c02*.
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 2-38. The sketch is shown in Figure 2-39. Do not dimension the sketch. The solid model and its dimensions are shown for your reference. **(Expected time: 20 min)**

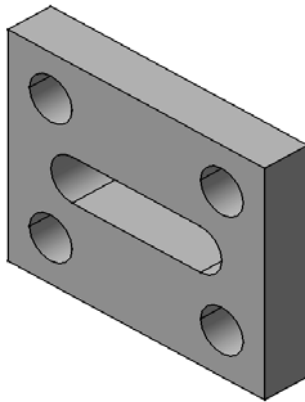


Figure 2-38 The solid model for Tutorial 2

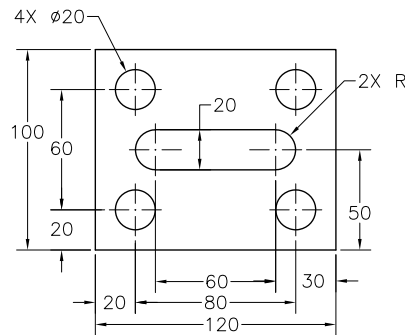


Figure 2-39 The sketch for the solid model

The following steps are required to complete this tutorial:

- a. Start a new Part file.
- b. Draw the sketch of the model using the **Rectangle**, **Profile**, and **Circle** tools, refer to Figures 2-41 through 2-43.
- c. Save the sketch and close the file.

Starting a New Part File

If you are starting a new session of CATIA, close the default **Product** file.

1. Choose **File > New** from the menu bar; the **New** dialog box is displayed. Alternatively, choose the **New** tool from the **Standard** toolbar to invoke the **New** dialog box, refer to Figure 2-40.
2. In this dialog box, select **Part** from the **List of Types** list box and then choose the **OK** button; the **New Part** dialog box is displayed.

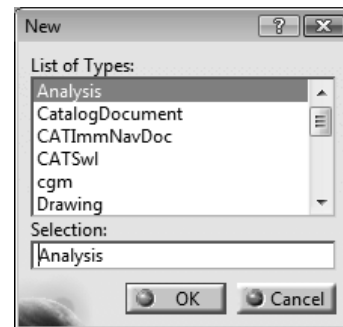


Figure 2-40 The New dialog box

3. Enter *c02tut2* as the name of the file in the **Enter part name** edit box and choose the **OK** button from the **New Part** dialog box; the new **Part** file opens in the **Part Design** workbench.
4. Choose the **Sketch** tool from the **Sketcher** toolbar and then select the *yz* plane from the Specification tree as the sketching plane; **Sketcher** workbench is invoked.

In this Tutorial, you need to draw the sketch in two parts. Initially, you need to draw the outer loop of the sketch, a rectangle and then the inner loops of the sketch, which consist of four holes and an elongated hole. First, draw an elongated hole using the **Profile** tool and then the four holes using the **Circle** tool.

Drawing the Outer Loop of the Sketch

In this section, you need to draw the outer loop of the sketch using the **Rectangle** tool.

1. Choose the **Rectangle** tool from the **Profile** toolbar.
2. Move the cursor to the location whose coordinates are -60,-50 and specify the lower left corner of the rectangle.
3. Move the cursor to the location whose coordinates are 60,50 and specify the upper right corner of the rectangle. Figure 2-41 shows the outer loop of the sketch drawn using the **Rectangle** tool.

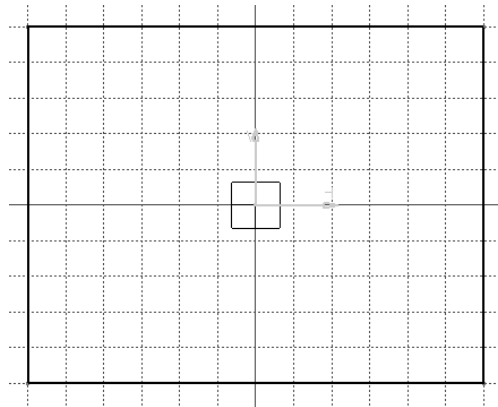


Figure 2-41 The outer loop of the sketch


Drawing the Inner Loop of the Sketch

You have drawn the outer loop of the sketch. Now, you need to draw its inner loop.

1. Choose the **Profile** tool from the **Profile** toolbar.
2. Move the cursor to the location whose coordinates are -30,10 and specify this point as the start point of the line.
3. Move the cursor horizontally toward the right and specify the endpoint of the line when the value of the coordinates is 30,10.



Next, you need to draw a tangent arc by switching over to the **Tangent Arc** option using the **Sketch tools** toolbar.

4. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch over to the **Tangent Arc** mode. 
5. Move the cursor to the location whose coordinates are 30,-10 and specify this point as the endpoint of the tangent arc.

Note that, after specifying the endpoint of the tangent arc, the **Line** mode is activated and the line is attached to the cursor again.

6. Move the cursor to the location whose coordinates are -30,-10 and specify the endpoint of the line.
7. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch to the **Tangent Arc** mode.
8. Move the cursor to the start point of the first horizontal line of the elongated hole and then specify the endpoint of the arc when it snaps to the start point.

The sketch after drawing the elongated hole is shown in Figure 2-42. In this figure, the constraints are hidden for better visualization.

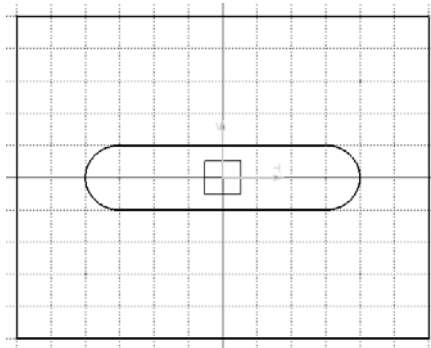


Figure 2-42 The sketch after drawing the elongated hole



Note

The elongated hole can also be created using the **Elongated Hole** tool and will be explained in the next chapter.

9. Choose the **Circle** tool from the **Profile** toolbar.
10. Move the cursor to the location whose coordinates are 40, 30 and specify the center point of the circle.
11. Enter **10** as the radius value of the circle in the **R** edit box of the **Sketch tools** toolbar and press ENTER.

You will notice that a radius dimension is displayed attached to the circle. This is because you have specified the radius value of the circle in the **R** edit box of the **Sketch tools** toolbar.

12. Choose the **Circle** tool from the **Profile** toolbar.
13. Move the cursor to the location whose coordinates are 40,-30 and specify the center point of the circle.
14. Enter **10** as the radius value of the circle in the **R** edit box of the **Sketch tools** toolbar and press ENTER.



Tip. If you double-click on the **Line** tool in the **Profile** toolbar, the **Line** tool will be invoked and will not exit after drawing a single line, until you press the ESC key twice. As a result, you can draw multiple lines using this tool.

15. Similarly, draw the other two circles. The coordinates of the center point of the other two circles are -40, 30 and -40, -30, respectively. The final sketch, with the display of constraints turned on, is shown in Figure 2-43.

Saving the Sketch

1. Choose the **Save** tool from the **Standard** toolbar to invoke the **Save As** dialog box. Browse to the folder named *c02* that you created in the first tutorial of this chapter.
2. Choose the **Save** button; the file is saved at *C:\CATIA\c02*.

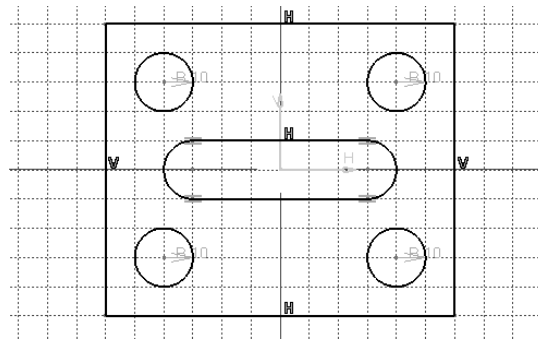


Figure 2-43 The final sketch

3. Next, 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 2-44. The sketch is shown in Figure 2-45. Do not dimension the sketch. The solid model and its dimensions are shown for your reference.
(Expected time: 20 min)

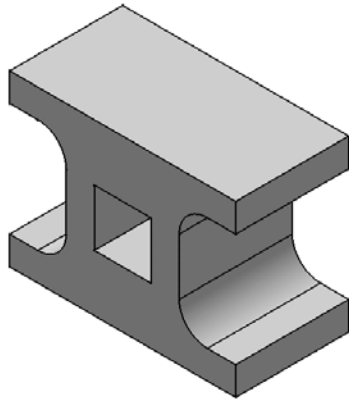


Figure 2-44 The solid model for Tutorial 3

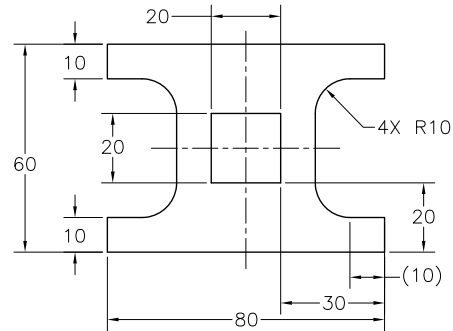


Figure 2-45 The sketch of the model

The following steps are required to complete this tutorial:

- Start a new Part file.
- Draw the sketch of the model using the **Profile** and **Rectangle** tools, refer to Figures 2-46 through 2-48.
- Save and close the file.

Starting a New Part File

- Choose **File > New** from the menu bar; the **New** dialog box is displayed.
- In **New** dialog box, select **Part** from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
- Enter *c02tut3* as the name of the file in the **Enter part name** edit box. Accept the rest of default options in the **New Part** dialog box and choose the **OK** button; a new **Part** file opens in the **Part Design** workbench.
- Choose the **Sketch** tool from the **Sketcher** toolbar and then select the *yz* plane as the sketching plane to invoke the **Sketcher** workbench.

Now, you need to draw the sketch in two parts: first the outer loop and then the inner cavity.

Drawing the Outer Loop of the Sketch

In this section, you need to draw the outer loop of the sketch using the **Profile** tool. Start drawing the outer loop from the lower left corner of the sketch. It is recommended that you keep the origin in the middle of the sketch drawn as it will reduce the time required for constraining and dimensioning the sketches. Also, it will help you capture the design intent easily.

- Choose the **Profile** tool from the **Profile** toolbar.



2. Move the cursor to the third quadrant; the coordinates of the point are displayed above the cursor.
3. Specify the start point of the line at the location whose coordinates are -40,-30 and then move the cursor horizontally toward the right.

On moving the cursor horizontally, you will notice that the line turns blue.

4. Move the cursor to the location whose coordinates are 40, -30. You can see the coordinates of the point on top of the cursor.
5. Specify the endpoint of the line. A rubber band line is attached to the cursor.
6. Move the cursor vertically upward and specify the endpoint of the second line on the point whose coordinates are 40,-20.
7. Move the cursor horizontally toward the left and specify the endpoint of the third line where the coordinates are 30, -20.

After drawing these three lines, you need to draw a tangent arc using the **Tangent Arc** button from the **Sketch tools** toolbar.

8. Choose the **Tangent Arc** button from the **Sketch tools** toolbar.
9. Move the cursor to the location whose coordinates are 20, -10 and specify the endpoint of the tangent arc; the **Line** mode is activated and the line gets attached to the cursor again. Figure 2-46 shows the sketch after drawing three lines and a tangent arc. In this figure, the constraints are hidden for better visualization.

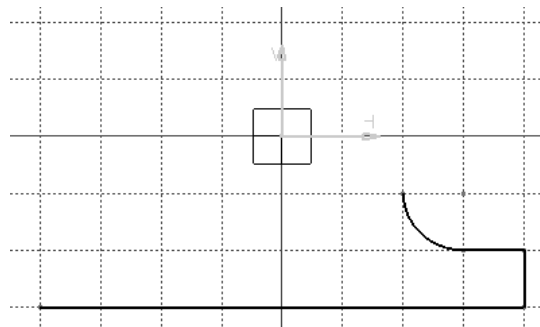


Figure 2-46 The sketch after drawing three lines and a tangent arc

10. Move the cursor vertically upward to the location whose coordinates are 20, 10 and then specify the endpoint of the line at this location.

Next, you need to draw a tangent arc using the **Tangent Arc** button from the **Sketch tools** toolbar.

11. Choose the **Tangent Arc** button from the **Sketch tools** toolbar; the **Tangent Arc** mode is activated.
12. Move the cursor to the location whose coordinates are 30, 20 and specify the endpoint of the tangent arc. As soon as you specify the end point of the tangent arc, the **Line** mode is activated again.
13. Move the cursor horizontally toward the right and specify the endpoint of the line, when the coordinates are 40,20.
14. Move the cursor vertically upward and specify the endpoint of the line, where the coordinates are 40,30.
15. Move the cursor horizontally toward the left and specify the endpoint of the line, where the coordinates are -40,30.
16. Move the cursor vertically downward and specify the endpoint of the line, where the coordinates are -40,20.
17. Move the cursor horizontally toward the right and specify the endpoint of the line, where the coordinates are -30,20.

Next, you need to draw a tangent arc by choosing the **Tangent Arc** button from the **Sketch tools** toolbar.

18. Choose the **Tangent Arc** button from the **Sketch tools** toolbar; the **Tangent Arc** mode is activated.
19. Move the cursor to the location whose coordinates are -20,10 and specify the endpoints of the arc at this location; the **Line** mode is activated and line is attached to the cursor.
20. Move the cursor vertically downward and specify the endpoint of the line, where the coordinates are -20,-10.
21. Switch to the **Tangent Arc** mode by choosing the **Tangent Arc** button from the **Sketch tools** toolbar and then move the cursor to the location whose coordinates are -30,-20. Next, specify the endpoint of the tangent arc at this location.
22. Move the cursor horizontally toward the left and specify the endpoint of the line, where the coordinates are -40,-20.
23. Move the cursor vertically downward and specify the endpoint of the line when it snaps to the start point of the outer loop. The sketch after drawing the outer loop and hiding the constraints is shown in Figure 2-47.

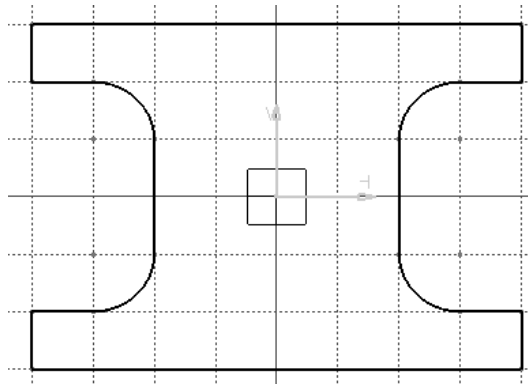


Figure 2-47 The sketch after drawing the outer loop and hiding the constraints

Drawing the Inner Cavity of the Sketch

After drawing the outer loop of the sketch, you need to draw its inner rectangular cavity using the **Rectangle** tool.

1. Choose the **Rectangle** tool from the **Profile** toolbar.
2. Move the cursor to the location whose coordinates are -10,10 and specify the upper-left corner of the rectangle at this location.
3. Move the cursor to the location whose coordinates are 10,-10 and specify the lower-right corner of the rectangle at this location.
4. Choose the **Fit All In** button from the **View** toolbar to fit the sketch into the geometry area.



The final sketch after drawing the inner loop is shown in Figure 2-48. Note that in this figure, the display of constraints has been turned on.

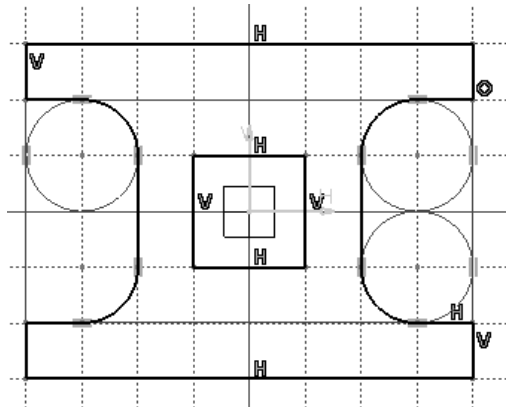



Figure 2-48 The final sketch after drawing the inner loop

Saving the Sketch

After completing the sketch, you need to save it.

1. Choose the **Save** tool from the **Standard** toolbar to invoke the **Save As** dialog box. Browse the *c02* folder that you created in the last tutorial. 
2. Next, choose the **Save** button from this dialog box; the file is saved at *C:\CATIA\c02*.
3. Close the part file by choosing **File > Close** from the menu bar.

Tutorial 4

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

The following steps are required to complete this tutorial:

- a. Start CATIA V5 and then start a new **Part** file.
- b. Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, refer to Figures 2-51 and 2-52.
- c. Save and close the file.

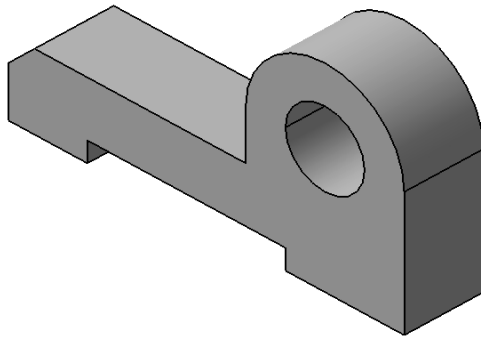


Figure 2-49 The solid model for Tutorial 4

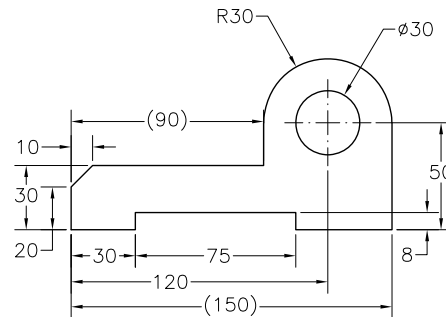


Figure 2-50 The sketch of the model

Starting CATIA V5 and Starting a New Part File

1. Choose **File > New** from the menu bar; the **New** dialog box is displayed.
2. In this dialog box, select **Part** from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
3. In the **New Part** dialog box, enter *c02tut4* as the name of the file in the **Enter part name** edit box. Accept the rest of the default options in the **New Part** dialog box and choose the **OK** button; a new **Part** file opens in the **Part Design** workbench.

4. Choose the **Sketch** tool from the **Sketcher** toolbar and then select the yz plane as the sketching plane to invoke the **Sketcher** workbench.



Now, you need to draw the sketch in two parts: first as the outer loop and second as the inner circle.

Drawing the Outer Loop of the Sketch

In this section, you need to draw the sketch symmetrically around the origin because it will reduce the time required for constraining and dimensioning it. You will draw the outer loop of the sketch using the **Line** and **Arc** tools.

1. Invoke the **Line** tool by choosing the **Line** tool from the **Profile** toolbar.
2. Choose the **Snap to Point** button from the **Sketch tools** toolbar, if it is not chosen.
3. Move the cursor in the third quadrant; the coordinates of the point are displayed above the cursor.
4. Specify the point whose coordinates are -50,-30. Next, move the cursor horizontally toward the right.



It is evident from Figure 2-50 that the length of the first horizontal line at the lower left corner of the sketch is 30mm. Therefore, you need to move the cursor until the length of the line is shown as 30mm in the **L** edit box of the **Sketch tools** toolbar and press left mouse button.

5. Press the left mouse button when the length of the line in the **L** edit box of the **Sketch tools** toolbar displays a value of 30mm.

After drawing the first horizontal line, you will notice that a **Horizontal** constraint is applied to it. Note that the line is still selected and displayed in orange. Click anywhere in the geometry area to remove it from the selection set.


As soon as you specify the endpoint of the line, the **Line** tool gets terminated. Therefore, you need to choose the **Line** tool again and again to draw multiple lines. You can avoid it by double-clicking on the **Line** tool in the **Profile** toolbar. On doing so, the **Line** tool will not terminate until you press the ESC key twice.

6. Double-click on the **Line** tool to invoke the **Line** tool and select the endpoint of the first horizontal line.
7. Press the TAB key four times to highlight the value displayed in the **L** edit box of the **Sketch tools** toolbar. Enter 8 in this edit box and then press the ENTER key.
8. Now, move the cursor vertically upward and click when a vertical line is displayed in blue.

A vertical line of length 8mm is drawn. You will notice that this line is no longer in the selection mode and you are prompted to select the start point of the next line. This happens because of double-clicking on the **Line** tool. It makes the **Line** tool active, until another tool is invoked.

9. Select the endpoint of the vertical line as the start point of the second horizontal line. Enter **75** in the **L** edit box of the **Sketch tools** toolbar and press ENTER. Now, move the cursor horizontally toward the right and click when a horizontal line is displayed; the second horizontal line of length 75mm is drawn.
10. Select the endpoint of the second horizontal line as the start point of the second vertical line and move the cursor vertically downward. Click when the **L** edit box displays a value of 8mm; the second vertical line of length 8mm is drawn.
11. Select the endpoint of the second vertical line as the start point of the third horizontal line and move the cursor horizontally toward the right. Click to draw the line, when the length in the **L** edit box shows a value of 45mm.
12. Select the endpoint of the previous line as the start point of the third vertical line and move the cursor vertically upward. Click to draw the line, when the **L** edit box displays a value of 50mm; the third vertical line of length 50mm is drawn.

Next, you need to draw a three point arc using the **Three Point Arc** tool.

13. To draw the three point arc, first invoke the **Circle** drop-down by choosing the down arrow on the right of the **Circle** tool in the **Profile** toolbar. Choose the **Three Point Arc** tool from the **Circle** drop-down to invoke the **Three Point Arc** tool. 
14. Select the start point of the arc as the endpoint of the previous vertical line.
15. Move the cursor to the point whose coordinates are 70mm, 50mm. These coordinates are displayed in the **Sketch tools** toolbar and also on top of the cursor. Now, click in the geometry area to specify the second point.
16. Move the cursor to specify the third point of the arc. Click on the point when the cursor snaps to a location 40, 20 in the geometry area. The coordinate values are displayed on top of the cursor.

This draws the arc of the outer loop. The arc is in the selection mode. Click anywhere in the geometry area to exit the selection mode. Now, to continue drawing the outer loop, you need to invoke the **Line** tool again.

17. Double-click on the **Line** tool in the **Profile** toolbar to invoke the **Line** tool.
18. Select the endpoint of the arc as the start point of the fourth vertical line. Move the cursor vertically downward to draw it. Click to draw the line, when the length in the **L** edit box shows a value of 20mm in the **Sketch tools** toolbar.

- This draws the fourth horizontal line of length 80mm. Note that the line is green in color, because it passes through the origin.

- This completes the sketch of the outer loop. It is recommended that you modify the geometry area such that the sketch fits inside the screen. This can be done by using the **Fit All In** tool.

-

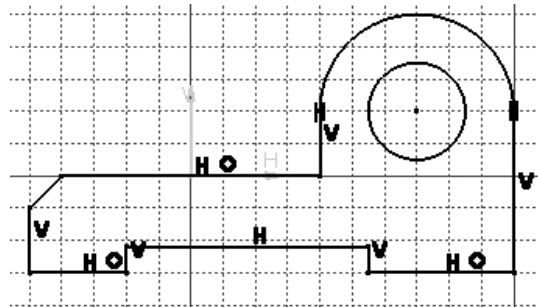
Figure 2-51 The completed outer loop of the sketch

Drawing the Circle

1. Choose the **Circle** tool from the **Circle** drop-down to invoke the **Circle** tool; you are prompted to define the center point of the circle.



- The final sketch with the display of geometrical constraints turned on is shown in Figure 2-52.



Saving and Closing the Sketch

- 

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

- Evaluation Copy. Do not reproduce. For information visit www.cadcim.com**

5. You can convert a sketched element into a construction element by using the _____ tool.
6. To draw a rectangle at an angle, you need to use the _____ tool.
7. The rectangle is considered as a combination of individual _____.
8. The _____ tool is used to draw continuous lines.
9. Using the _____ tool, you can create a circle by specifying the coordinates of its center point.
10. The _____ are the temporary lines that are used to track a particular point on the screen.

REVIEW QUESTIONS

Answer the following questions:

1. The 3 point arcs are the ones that are drawn by defining a start point, an endpoint, and a point on the arc. (T/F)
2. The **Parallelogram** tool is available in the **Predefined Profile** drop-down. (T/F)
3. The **Symmetrical Extension** button, when chosen from the **Sketch tools** toolbar, draws a simple line. (T/F)
4. In CATIA V5, circles are drawn by specifying the center point of the circle and then entering radius value in the dialog box that is displayed. (T/F)
5. When you start CATIA V5, a file in the **Product** workbench is started by default. (T/F)
6. In CATIA V5, a combination of which of the following elements is considered as a rectangle?
 - (a) **Lines**
 - (b) **Arcs**
 - (c) **Splines**
 - (d) None of these
7. Which of the following tools is not available in the **Predefined Profile** toolbar?
 - (a) **Rectangle**
 - (b) **Oriented Rectangle**
 - (c) **Parallelogram**
 - (d) **Circle**

8. Which one of the following elements will not be considered while converting a sketch into a feature?
- (a) **Sketched circles** (b) **Sketched lines**
(c) **Construction elements** (d) None of these
9. Which one of the following tools is available in the **Line** toolbar?
- (a) **Line** (b) **Infinite Line**
(c) **Bisecting Line** (d) All of these
10. In which workbench of CATIA V5, you can draw the sketches that can be used to create features?
- (a) **Part** (b) **Assembly**
(c) **Shape** (d) None of these

EXERCISES

Exercise 1

Draw the sketch of the model shown in Figure 2-53. The sketch to be drawn is shown in Figure 2-54. Do not dimension it. The solid model and dimensions are shown for your reference.

(Expected time: 30 min)

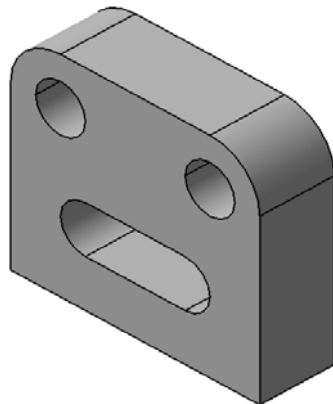


Figure 2-53 The solid model for Exercise 1

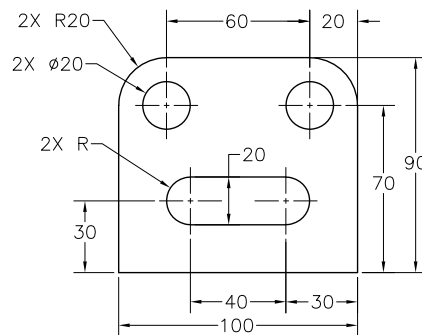


Figure 2-54 The sketch of the model

Exercise 2

Draw the sketch of the model shown in Figure 2-55. The sketch to be drawn is shown in Figure 2-56. Do not dimension it. The solid model and dimensions are shown for your reference.

(Expected time: 30 min)

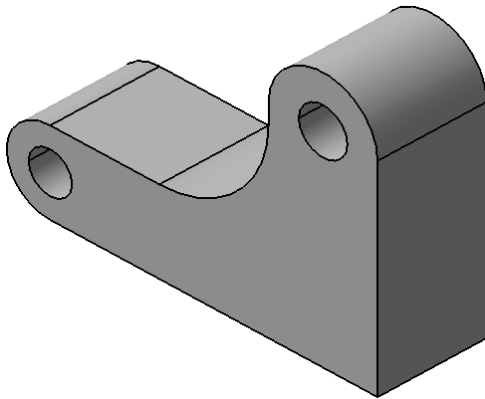


Figure 2-55 The solid model for Exercise 2

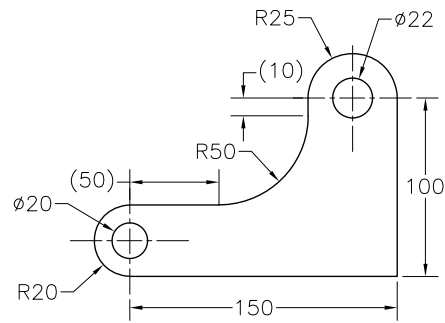


Figure 2-56 The sketch of the model

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. F, 5. Construction/Standard Element, 6. Oriented Rectangle, 7. lines,
8. Profile, 9. Circle Using Coordinates, 10. inferencing lines.