



Chapter 1

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 some important Sketcher terms.*
- *Draw sketches using some of the tools in the Sketcher workbench.*
- *Use some of the drawing display tools.*

THE SKETCHER WORKBENCH

Most of the 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 1-1. The base sketch to create this solid model is shown in Figure 1-2.

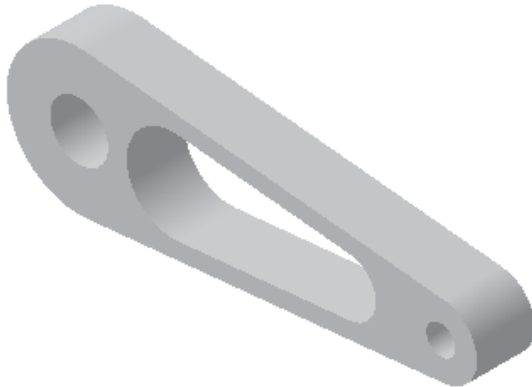


Figure 1-1 Solid Model of a Link.

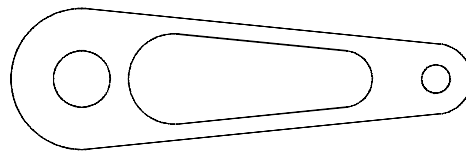


Figure 1-2 Base sketch for the solid model.

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 conversant 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** button from the **Sketcher** toolbar. Next, select a plane to draw the sketch. Draw the sketch and proceed to the **Part Design** or the **Wireframe and Surface Design** workbench to convert it into a solid model or surface model.

STARTING A NEW FILE

When you start CATIA V5R14, a new **Product** file, with the name **Product1**, is displayed on the screen, as shown in Figure 1-3. Close this file and start a new one in the **Part Design** workbench. You will learn more about the **Product** files in the later chapters.



When you choose **File > Close** from the menu bar, the start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Part Design** to make sure that you are in the **Part Design** workbench. The **Part name** dialog box is displayed, as shown in Figure 1-4. Enter the file name in the **Enter part name** edit box and choose **OK** to start a new file in the **Part Design** workbench. Alternatively, you can choose the **File** option from the menu bar; the **New** dialog box is displayed, as shown in Figure 1-5. Select **Part** from the **List of Types** list

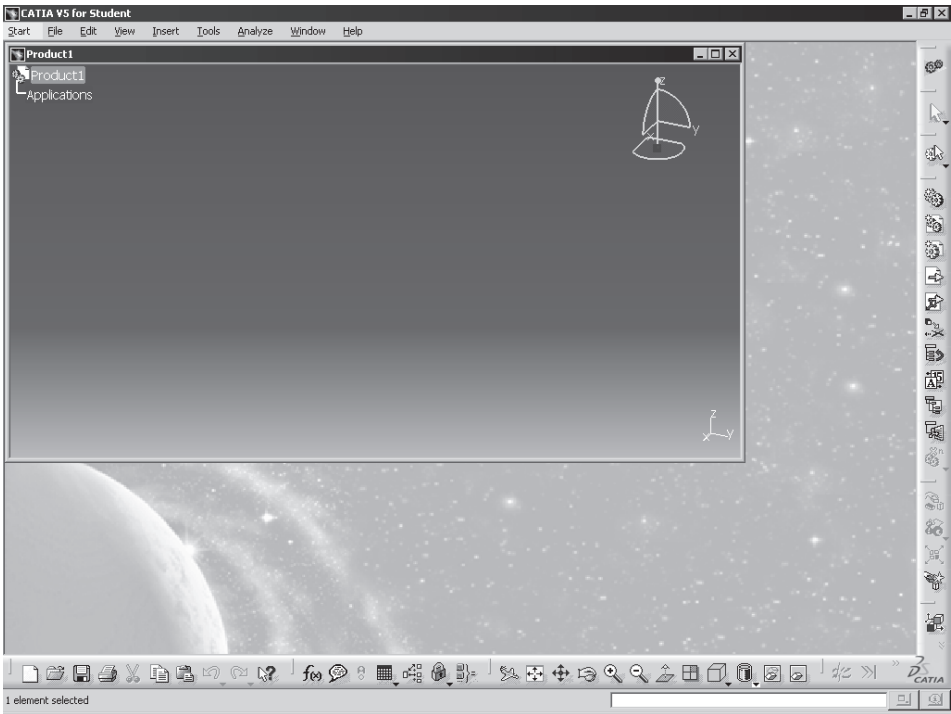


Figure 1-3 Initial screen that appears after starting CATIA V5R14

box provided in the **New** dialog box. You can also write the word **Part** in the **Selection** edit box at the bottom of the **List of Types** list box. Choose the **OK** button; the **Part name** dialog box is displayed. Enter the file name in it and choose the **OK** button.

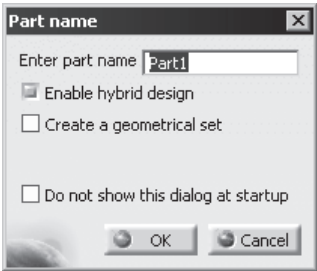


Figure 1-4 The **Part name** dialog box

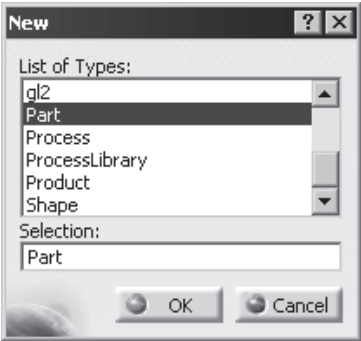


Figure 1-5 The **New** dialog box



Tip. If you clear the **Enable hybrid design** check box from the **Part name** dialog box, the new file will be started in the conventional design mode. In 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 is used. Therefore, it is recommended that you keep the **Enable hybrid design** check box selected, everytime you start a new file.

A new file in the **Part Design** workbench is displayed on the screen, as shown in Figure 1-6. The standard tools like the specification tree, **Compass**, **Geometry Axis** 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 Axis** is displayed on the bottom right corner of the screen.

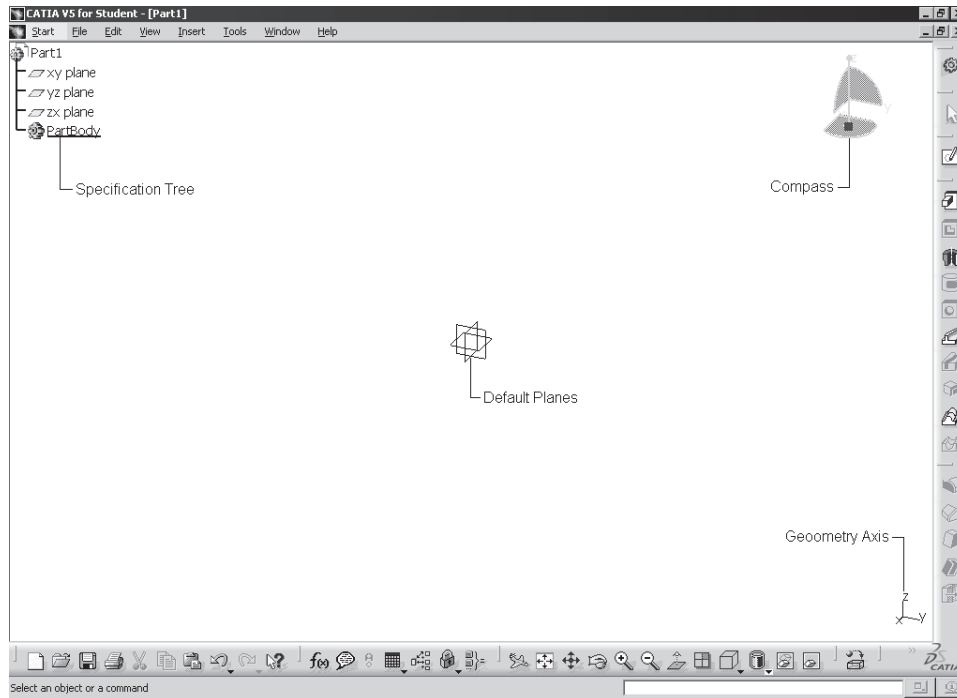


Figure 1-6 A new **Part Design** workbench file



Note

You can hide the **Compass**, specification tree, or the **Geometry Axis** by using the **View** menu. By default, check marks are displayed on the left of **Geometry**, **Specifications**, and **Compass** in the menu bar. This suggests 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 not to hide these standard tools. You can also use the **F3** key to toggle the display of the specification tree.

INVOKING THE SKETCHER WORKBENCH

The sketch is the basic requirement to create the base feature of any solid model. In CATIA V5, a sketch is drawn in the **Sketcher** workbench. To invoke the **Sketcher** workbench, choose the down arrow on the right of the **Sketcher** button in the **Sketcher** toolbar; a flyout appears. Press and hold the left mouse button on the horizontal line at the top of the flyout and drag it. The flyout is detached from its parent toolbar and becomes an independent toolbar. Figure 1-7 shows the **Sketcher** flyout as an independent toolbar. The two buttons in the **Sketcher** toolbar are, **Sketcher** and **Sketch with Absolute Axis definition**. The next section focuses on invoking the **Sketcher** workbench, using these two buttons.

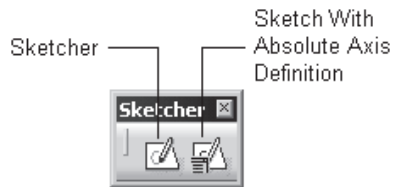


Figure 1-7 The *Sketcher* toolbar

Invoking the Sketcher Workbench Using the Sketcher Button



To invoke the **Sketcher** workbench, using the **Sketcher** tool, choose the **Sketcher** button from the **Sketcher** toolbar. You are 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 **Sketcher** workbench, that appears after on selecting the YZ plane as the sketching plane, is shown in Figure 1-8. The selected plane is parallel to the screen. You are prompted to select an object or a command. The sketching components that are displayed in the geometry area are discussed later in this chapter.

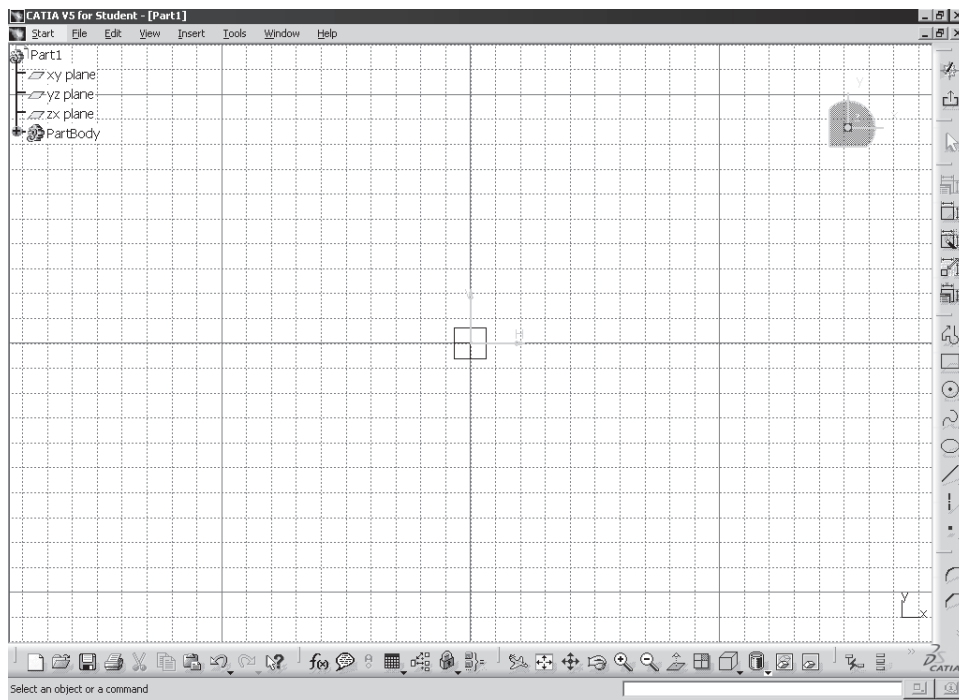


Figure 1-8 The *Sketcher* workbench invoked using the YZ plane as the sketching plane



Note

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

Invoking the Sketcher Workbench using the Sketch with Absolute Axis Definition Button



In CATIA V5, you can also define a user-defined absolute axis while invoking the **Sketcher** workbench, by using the **Sketch With Absolute Axis Definition** option. To invoke the **Sketcher** workbench using this option, choose the **Sketch With Absolute Axis Definition** button from the **Sketcher** toolbar. The **Sketch Positioning** dialog box will appear, as shown in Figure 1-9. You are then prompted to select a plane, planar face, sketch, an axis system, or two lines. You can set the absolute axis by using the options in this dialog box.

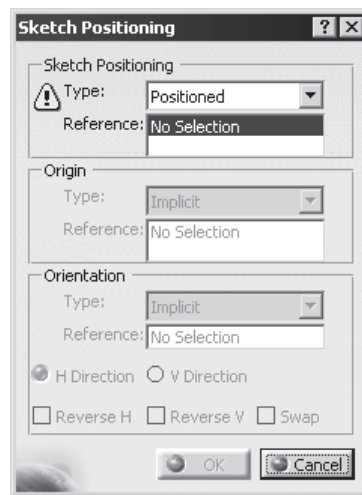


Figure 1-9 The Sketch Positioning dialog box

SETTING UP 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.

Modifying Units

To modify units, invoke the **Options** dialog box by choosing **Tools > Options** from the menu bar. Click on the + sign on the left of the **General** option to expand the tree. Choose the **Parameters and Measure** option; the tabs corresponding to this selection appear on the right side of the **Options** dialog box. Next, choose the **Units** tab. The **Options** dialog box, after invoking the **Units** tab, is shown in Figure 1-10.

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

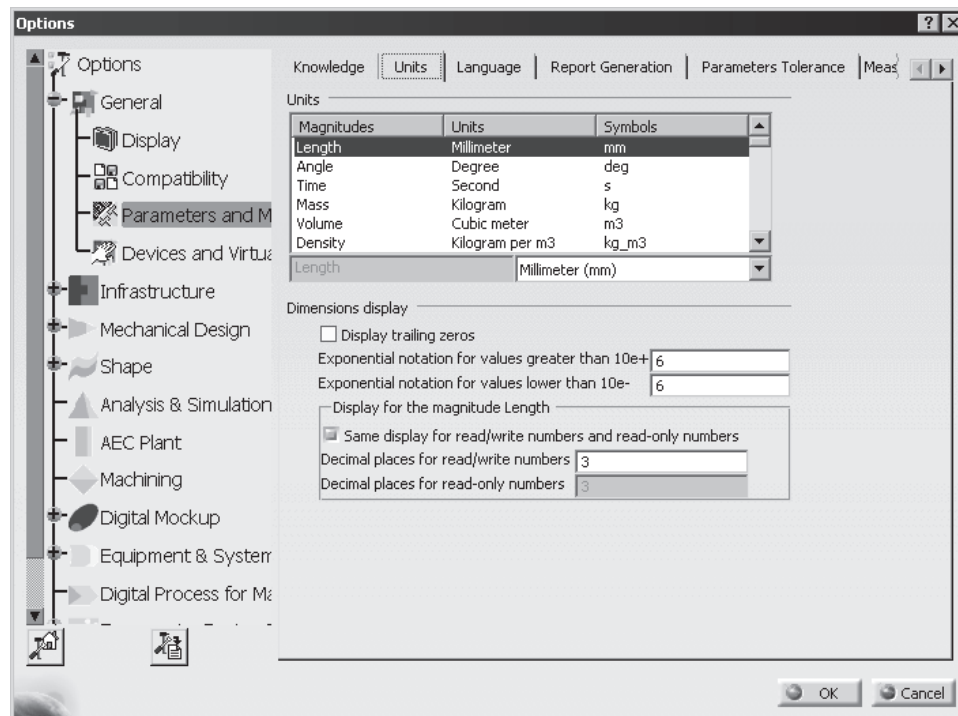


Figure 1-10 The Options dialog box with the Units tab chosen

Modifying the Grid Settings

On invoking the **Sketcher** workbench, you will observe two types of lines in the geometry area flowing in the horizontal and vertical directions. These are continuous lines and dotted black lines. The spacing between the two dotted lines is called graduation, while the spacing between the two continuous black lines is called primary spacing. The mesh that is formed because of the intersection of these lines in the vertical and horizontal direction 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 the horizontal and vertical direction. 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 direction 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** is said to be distorted. To change the values of **Primary Spacing** and **Graduations**, choose **Tools > Options** from the menu bar; the **Options** dialog box will be displayed. Choose the **Mechanical Design** option from the tree on the left of the dialog box. Next, choose the **Sketcher** option to display the **Sketcher** tab on the right of the **Options** dialog box, as shown in Figure 1-11.

The edit boxes of **Primary Spacing** and **Graduations** under the **H** row are already enabled. Here, **H** refers to the horizontal direction. To enable the edit boxes of **Primary Spacing**

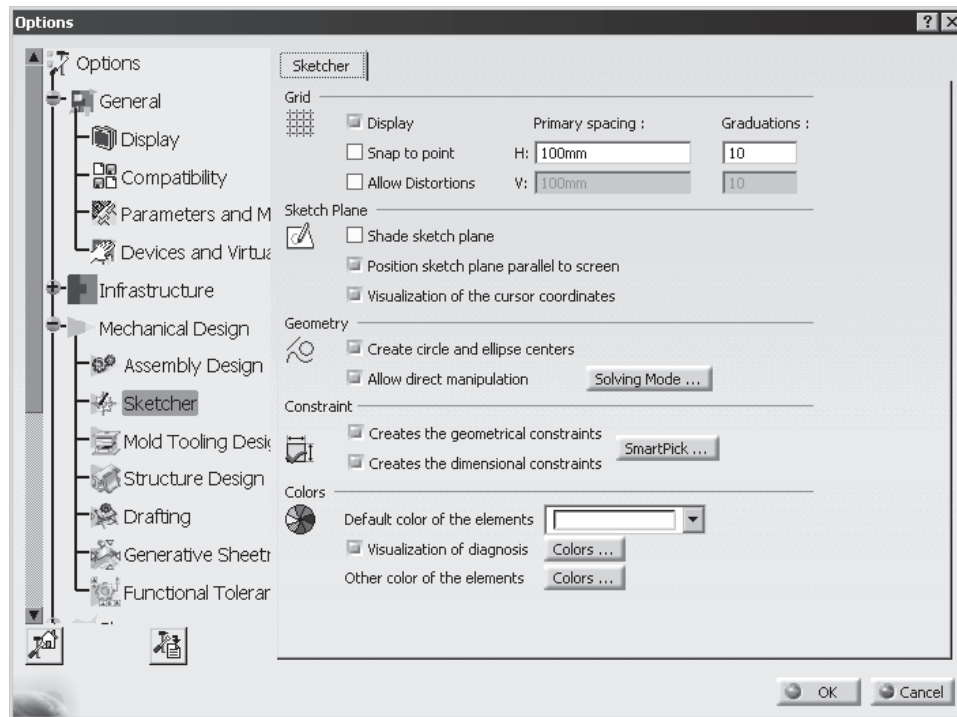


Figure 1-11 The **Options** dialog box with the **Sketcher** option selected

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. Choose **OK** to apply the newly formed **Grid** to the **Sketcher** workbench. Note, all the files that you open or start in the **Sketcher** workbench, henceforth, will use these values for the **Grid**.

UNDERSTANDING THE SKETCHER TERMS

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

Specification Tree

The specification tree is a manager, that keeps a track of all the 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 1-12 shows the specification tree in the expanded form.

The various levels under **Sketch.1** in the **Tree** are discussed next.

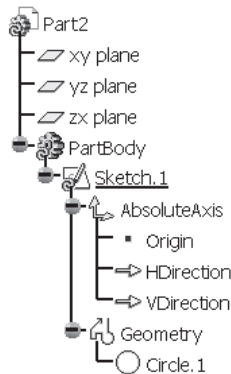


Figure 1-12 The expanded form of the specification tree



Tip. While expanding the branch of the specification tree, you may accidentally click on the branch lines. This will make the specification tree active and consequently the geometry area will be frozen. Note, that the color of the default planes will turn grey. Now, zooming and panning will, respectively, resize or reposition the specification tree, instead of the geometry view. The geometry area can be reactivated by clicking on the branch line again.

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 axis will be highlighted in the geometry area, when **AbsoluteAxis** is selected from the specification tree. A + sign is available on the left of the **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.

Origin

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

HDirection

The direction that is parallel to the horizontal axis is represented by **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 that are associated with the specification tree in the **Sketcher** workbench, while drawing and constraining sketches.

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 to a construction element, or vice versa, using the **Construction/Standard Element** button.

Sketcher 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 **Sketcher** toolbar, shown in Figure 1-13. Various tools such as **Select**, **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.



Figure 1-13 The Sketcher toolbar

The tools in the **Sketcher** toolbar can be invoked by choosing the down arrow on the right of the **Select** tool. When you choose the down arrow, the **Select** flyout is displayed. Detach the **Select** flyout from the **Sketcher** toolbar by holding it from the vertical/horizontal line and place it in the geometry area. The **Select** flyout will now become the **Select** toolbar, as shown in Figure 1-14. The tools in the **Select** toolbar are discussed next.

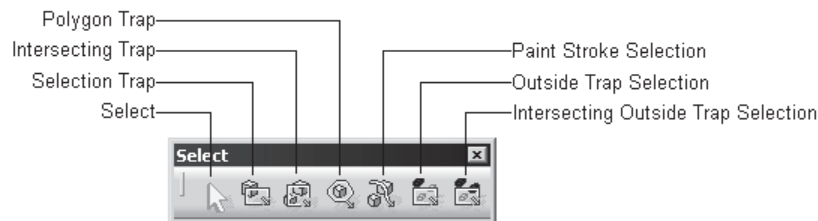


Figure 1-14 The Select toolbar



Note

For a better understanding and explanation of the buttons in the flyout, this book will refer to a flyout as a toolbar. This means, whenever you are asked to choose the down arrow on the right of any button, the flyout that appears will be called a toolbar. You can detach this flyout from the parent toolbar and place it in the geometry area.

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 is replaced with a hand cursor. Left click on the element to select it.

Selection Trap



This is a method of selecting elements by creating a selection trap. The trap is a rectangular box drawn by dragging the mouse to define the diagonally opposite corners. All the objects that lie fully inside the selection trap are selected. To create this trap, choose the **Selection Trap** button from the **Select** toolbar. Next, specify the first corner and then drag the mouse to specify the second corner.

Intersecting 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** button from the **Select** toolbar. Next, specify the first corner and then drag the mouse to specify the second corner.

Polygon Trap



This method includes selecting elements by drawing a closed polygon as the selection trap. You can select the elements of a sketch that are fully inside the polygon, by using this method. Choose the **Polygon Trap** button 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.

Paint Stroke Selection



This method includes selecting elements by dragging the mouse to draw a paint stroke across them. The elements that are intersected by the paint stroke are selected.

Outside Trap Selection



The elements that are outside the selection trap are selected by using this method. The elements that are intersected by the trap are not selected.

Intersecting Outside Trap Selection



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 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 end point 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 1-15 shows the use of the inferencing line to draw a line whose end point is tangent to an existing circle.

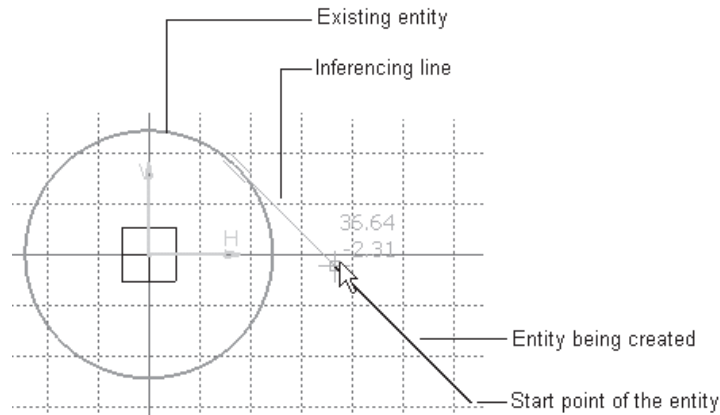


Figure 1-15 An example of an inferencing line

DRAWING SKETCHES USING THE SKETCHER TOOLS

The sketching tools that are used to draw the sketches in the **Sketcher** workbench are discussed next.

Drawing Lines

Menu: Insert > Profile > Line > Line
Toolbar: Profile > Line



The **Line** 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** button 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, invoke the **Line** tool by choosing the **Line** button 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.

After invoking the **Line** tool, you are prompted to select a point or click to locate the start point of the line. The prompt sequence is 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 are then prompted to specify its endpoint. Move the cursor away from the start point. You will notice a rubber band line attached to the cursor. Click anywhere in the geometry area to specify the endpoint of the line. Figure 1-16 shows a

line drawn by selecting the points from the display area. The orange color of the line specifies 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 suggests that it is a standard element.

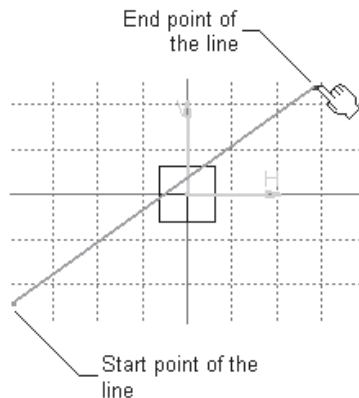


Figure 1-16 A line drawn, by selecting the start and endpoints from the geometry area



Note

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

Drawing Lines Using the Sketch tools Toolbar

Lines can also be drawn using the **Sketch tools** toolbar, which expands when you invoke the **Line** tool. Figure 1-17 shows the expanded **Sketch tools** toolbar, after you invoke the **Line** tool. The two methods to draw a line, using the **Sketch tools** toolbar, are discussed next.



Figure 1-17 The expanded form of the **Sketch tools** toolbar after invoking the **Line** tool

Drawing Lines by entering the Start and End point values

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 coordinates values of the start point, respectively, and then press ENTER. You will be prompted to enter the coordinates values for the endpoint. Specify the values in the **End Point H** and **V** edit boxes and press ENTER again. A line is drawn in the geometry area, corresponding to the entered values of the start and endpoints. Also, these dimensions of the start and endpoints are displayed from the origin. 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, suggesting 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 the chapters.

Similarly, you can draw a line by specifying the start point and entering the length and angle values.



Note

The specified dimension values for the start and endpoint are displayed, because the **Dimensional Constraint** button is chosen in the **Sketch tools** toolbar. 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 green. This suggests that the element is fully constrained. You will learn more about dimensioning and constraining of sketches in later chapters.



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.



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, 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** button from the **Profile** toolbar. The **Line** toolbar will appear, as shown in Figure 1-18. The types of lines that can be drawn using the **Line** toolbar are discussed next.

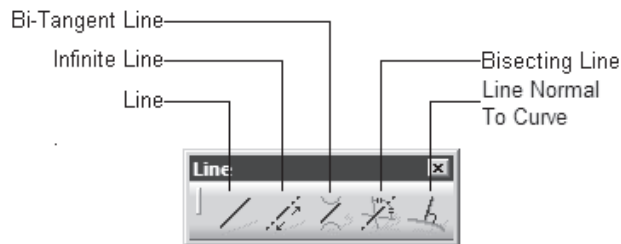


Figure 1-18 The **Line** toolbar

Drawing Infinite Lines

Menu: Insert > Profile > Line > Infinite Line
Toolbar: Profile > Line > Infinite Line



To draw an infinite line, invoke the **Infinite Line** tool from the **Line** toolbar. The **Sketch tools** toolbar is expanded. 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 > Bi-Tangent Line



Bi-Tangent lines are the lines that are tangent to two circles, arcs, ellipses, conics, or any curved geometry. 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 **Line** toolbar. Select the first element or the first curve geometry, and then, select the second element. A line will be drawn between the two selected curved elements. You will notice that the tangency symbol is visible on 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 > Bisecting Line



Bisecting lines are the lines that pass through two intersecting lines such that the angle formed between them is divided equally. To draw a bisecting line, invoke the **Bisecting Line** tool from the **Line** toolbar. Select the first line and then select the second line. A bisecting line of infinite length is drawn.

Drawing Lines Normal to a Curve

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



To draw a line normal to a curve, choose the **Line Normal To Curve** button from the **Line** toolbar. Specify the start point of the line anywhere in the geometry area; you are prompted to select the curve. After you select the curve, a line normal to it is drawn.



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

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 the later chapters. To draw an axis, invoke the **Axis** tool from the **Profile** toolbar. The **Sketch tools** toolbar is expanded. You are prompted to specify the start point of the axis. Click in the geometry area to specify it. You are then prompted to specify the endpoint of the axis. Move the cursor and click to specify it. An axis with the specified

points is displayed in the geometry area, as shown in Figure 1-19. You can also draw an axis by entering the parameters in the respective edit boxes of the expanded **Sketch tools** toolbar.



Note

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

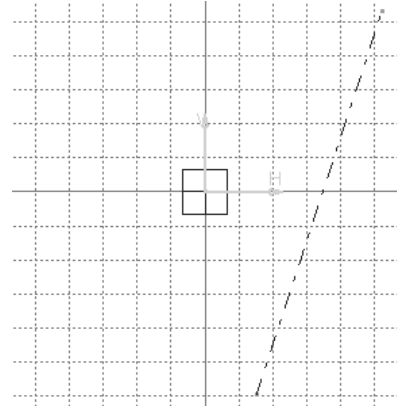


Figure 1-19 An axis drawn in the geometry area

Drawing Rectangles, Oriented Rectangles, and Parallelograms



The rectangle is a basic geometry that comprises of four sides. The adjacent sides are perpendicular to each other, while the opposite sides are equal in length. To draw a rectangle, choose the arrow on the right of the **Rectangle** button in the **Profile** toolbar. The **Predefined Profile** toolbar will be displayed, as shown in Figure 1-20. 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.

Drawing Rectangles

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



To draw a rectangle, invoke the **Rectangle** tool from the **Predefined Profile** toolbar, as shown in Figure 1-20.

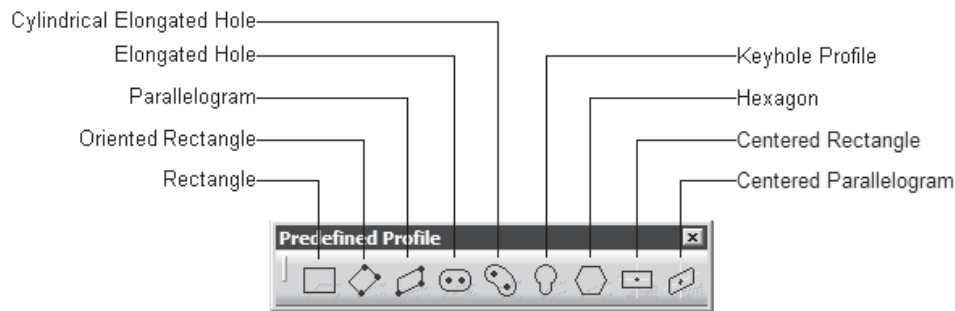


Figure 1-20 The **Predefined Profile** toolbar

When you invoke the **Rectangle** tool, the **Sketch tools** toolbar expands. You are 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. Next, you are 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.



Note

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

*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 a few dimensions and constraints are applied to the resulting rectangle. You will learn more about dimensioning and constraining in later chapters.*

Drawing Oriented Rectangles

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



To draw an oriented rectangle, invoke the **Oriented Rectangle** tool from the **Predefined Profile** toolbar. You are prompted to specify the first point. Click in the geometry area to specify it. You are then prompted to specify the end point. Move the cursor away from the first point in any direction to specify the end point of the first side. You will notice that a line is attached to the cursor. Click in the geometry area to specify the end point. The angle formed between the line and horizontal reference is the orientation angle of the rectangle.

You are then prompted to define the second side. Move the cursor in the upward or downward direction of the line. You will notice the rectangle being drawn. Also, the symbol of the perpendicular constraint is displayed between the line drawn and the line attached to the cursor. You will learn more about constraints in later chapters. Click in the geometry area to specify the third corner of the rectangle. Figure 1-21 shows an oriented rectangle being drawn.

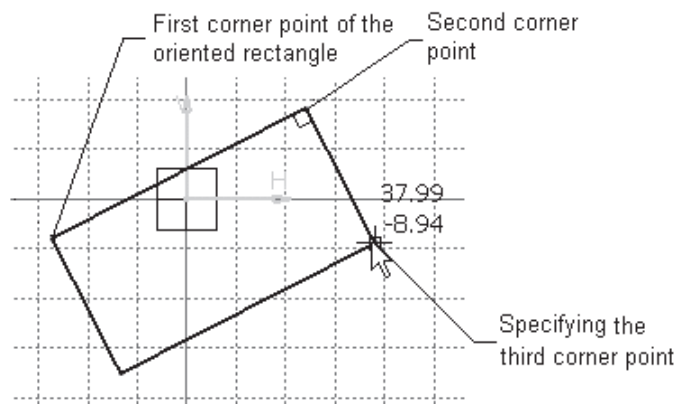


Figure 1-21 Selecting the corner points to draw an oriented rectangle

**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 typed the values.

Drawing Parallelograms

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



A parallelogram is a quadrilateral whose opposite sides are parallel to each other. To draw it, invoke the **Parallelogram** tool from the **Predefined Profile** toolbar; the **Sketch tools** toolbar expands. You are prompted to specify the start point of the parallelogram. Click in the geometry area to specify its first corner. You are 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 shows the first side of the parallelogram. Click in the geometry area to specify its end point. You are then prompted to specify its second side. Move the cursor away from the second corner; the preview of the parallelogram is displayed. Click to specify its second side. The parallelogram is created, as shown in Figure 1-22.

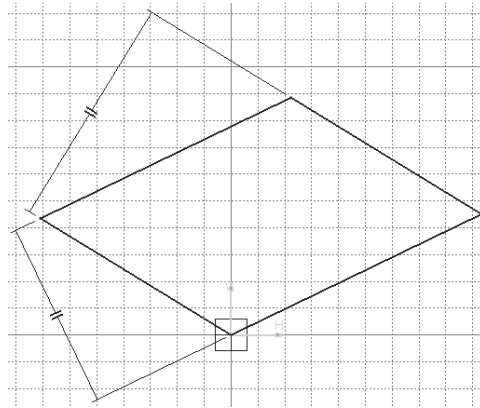


Figure 1-22 A parallelogram drawn by specifying the corner points

**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.

Drawing Points

A point is defined as the geometrical element that has no magnitude, length, width, or thickness. It is only specified by its position. In CATIA V5, you can draw 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 these methods,

choose the down arrow on the right of the **Point** button in the **Profile** toolbar. The **Point** toolbar is displayed, as shown in Figure 1-23.

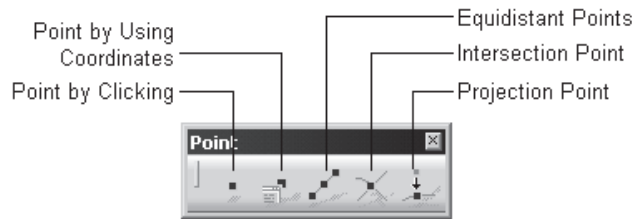


Figure 1-23 The **Point** toolbar

Drawing Points by Clicking

Menu: Insert > Profile > Point > Point
Toolbar: Profile > Point > Point by Clicking



To draw points by clicking, invoke the **Point by Clicking** tool from the **Point** toolbar; the **Sketch tools** toolbar expands. You are prompted to click to create the point, hence do so. You will notice a plus sign (+) is displayed in the geometry area. You can also enter the horizontal and vertical coordinates 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 its 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



A circle is defined as the path formed when a series of points called the locus move around a given point called the center, with a fixed distance called the radius. To draw a circle, choose the down arrow on the right of the **Circle** button in the **Profile** toolbar. The **Circle** toolbar is displayed, as shown in Figure 1-24. The available tools will help you to draw circles and arcs. These tools are discussed next.

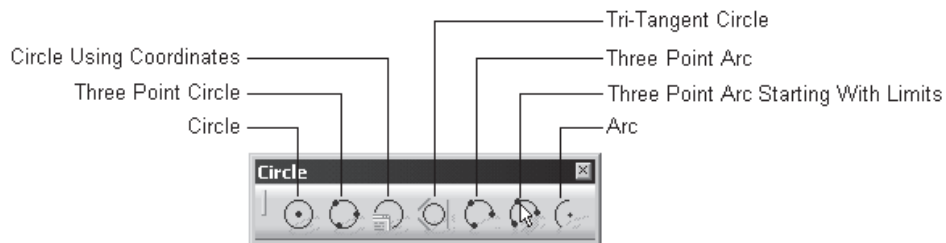


Figure 1-24 The **Circle** toolbar

Drawing Circles

Menu: Insert > Profile > Circle > Circle
Toolbar: Profile > Circle > Circle



To draw a circle, invoke the **Circle** tool from the **Circle** toolbar. You are prompted to specify its center. Click anywhere in the geometry area to specify the center point. Now, you are 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 is 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 boxes 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 > Three Point Circle



A circle can also be drawn by specifying any three points that lie on the circle. To draw a three point circle, invoke the **Three Point Circle** tool from the **Circle** toolbar; the **Sketch tools** toolbar expands. You are prompted to specify the start point of the circle. Click anywhere in the geometry area to specify it. You are then 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 is displayed. The dotted line originates from the first point and moves along with the cursor. This is the chord of the circle. Click in the geometry area to specify the second point on the circle. You are then prompted to specify the last point. As you move the cursor to specify the third point, the preview of the circle is displayed. Click to specify the third point.



Note

*You can also type 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
Toolbar: Profile > Circle > Circle Using Coordinates



In CATIA V5, circles 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 **Circle** toolbar; the **Circle Definition** dialog box is 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 > Tri-Tangent Circle



A tri-tangent circle is 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 the three elements, which can be lines, circles, ellipses, arcs, or any geometrical element, to which a circle can form a tangent relation. Then, invoke the **Tri-Tangent Circle** tool from the **Circle** toolbar. Select the three elements one by one. A circle tangent to all these three elements is displayed in the geometry area. 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, if 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** toolbar. 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 > Arc



To draw an arc by defining its center point, invoke the **Arc** tool from the **Circle** toolbar. You are 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. Now, 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 1-25 shows an arc drawn using this method.

Drawing Three Point Arcs

Menu: Insert > Profile > Circle > Three Point Arc
Toolbar: Profile > Circle > Three Point Arc



To draw a three point arc, invoke the **Three Point Arc** tool from the **Circle** toolbar. You are 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

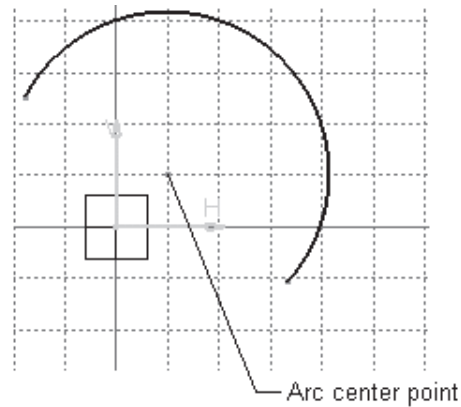


Figure 1-25 An arc

previous point. Click in the geometry area to specify its endpoint. Figure 1-26 shows selecting the first, second, and third point to draw a three point arc.

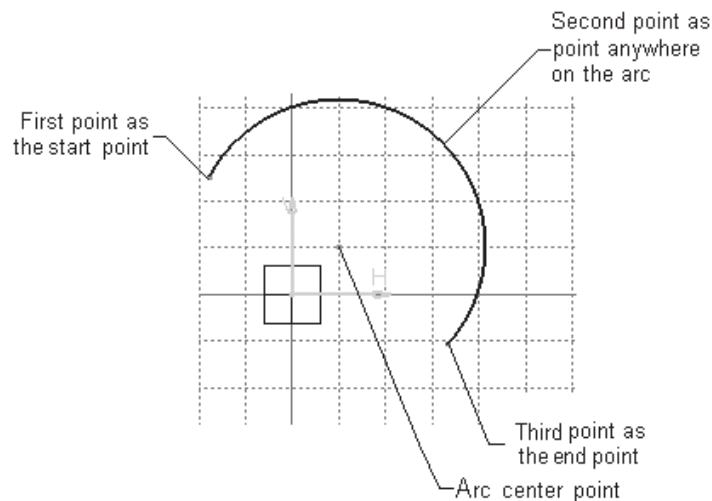


Figure 1-26 Selecting the points to draw a three point arc

Drawing Three Point Arcs Starting With Limits

Menu: Insert > Profile > Circle > Three Point Arc Starting With Limits
Toolbar: Profile > Circle > 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 type of arc, invoke the **Three Point Arc With Limits** tool from the **Circle** toolbar. You are then prompted to specify the start point of the arc. Click in the geometry area to specify the start point. You are then prompted to specify the endpoint of the arc. Move the cursor away

from the start point and click to specify the endpoint. You are then 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 is displayed. Click to specify the point on the arc. Figure 1-27 shows the selection of the first, second, and third point to draw an arc using this option.

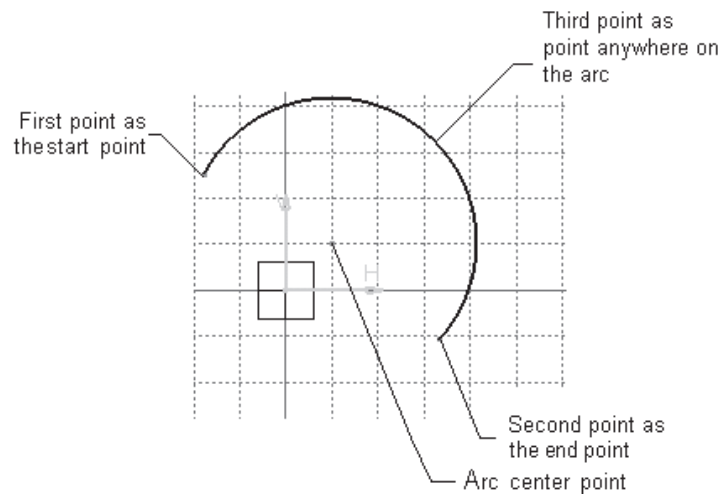


Figure 1-27 Selecting the points to draw a three point arc starting with limits

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 closed contour. To draw the profile, invoke the **Profile** tool from **Profile** toolbar. The **Sketch tools** toolbar expands and the **Line** tool is chosen in it. You are 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 is 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 the second line originating 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** button again. If you draw a closed profile, then you do not need to exit the **Profile** tool by choosing the **Profile** button from the **Profile** toolbar. This tool is automatically terminated when you specify the point to close the profile. Figure 1-28 shows an open profile.

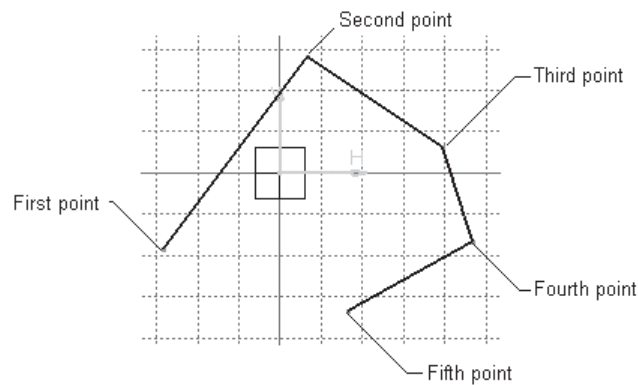


Figure 1-28 An open profile drawn using the **Profile** tool

You will notice that the expanded **Sketch tools** toolbar has three buttons: **Line**, **Tangent Arc**, and **Three Point Arc**, as shown in Figure 1-29. When you invoke the **Profile** tool, the **Line** button is 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.

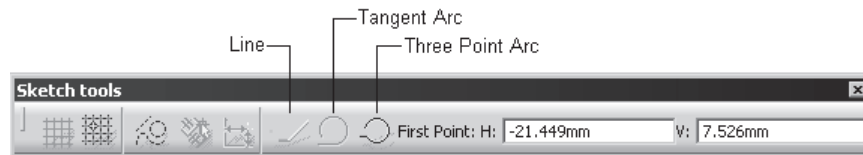


Figure 1-29 The **Sketch tools** toolbar

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 frozen. This is because you first need to draw at least one line. After drawing the line, the **Tangent Arc** button becomes available. Choose the **Tangent Arc** button from the expanded **Sketch tools** toolbar; the preview of the arc is displayed in the geometry area. You are prompted to specify the endpoint of the arc. Click in the geometry area to specify the endpoint. An arc, tangent to the line is drawn and displayed in the geometry area. Figure 1-30 shows a tangent arc being drawn, using the **Profile** tool. After drawing the arc, the line tool is again chosen in the **Sketch tools** toolbar and you are prompted to specify the endpoint of the current line.



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.

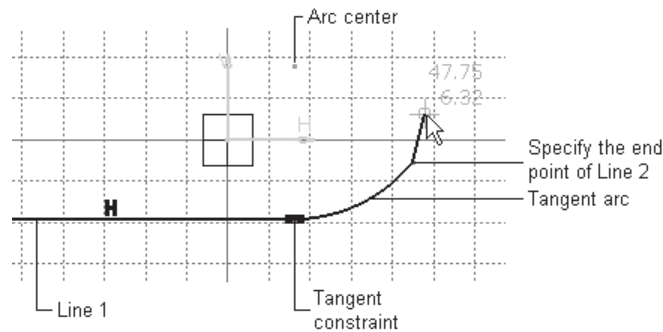


Figure 1-30 A tangent arc being drawn using the **Profile** tool

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 is available in the **Sketch tools** toolbar. You have two options. The first is to draw the line and then draw the three point arc. The second is to choose the **Three Point Arc** button first to draw a three point arc and then draw a line. Out of the two, you will learn to draw the line first and then the three point arc. Draw a line using the **Profile** tool. Without specifying the third point of the profile, choose the **Three Point Arc** button from the expanded **Sketch tools** toolbar; you are prompted to specify the second point of the arc. Remember, 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 are then prompted to specify its last point. Move the cursor and click to specify it. The three point arc is displayed in the geometry area. The **Profile** tool is still active. You are prompted to specify the endpoint of the current line. You can choose the **Profile** button again to end 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 1-31. The basic tools such as, **Zoom**, **Rotate**, **Pan**, **Normal View**, **Hide/Show**, and **Fit All In** will be discussed next. You will learn about the remaining tools in later chapters.

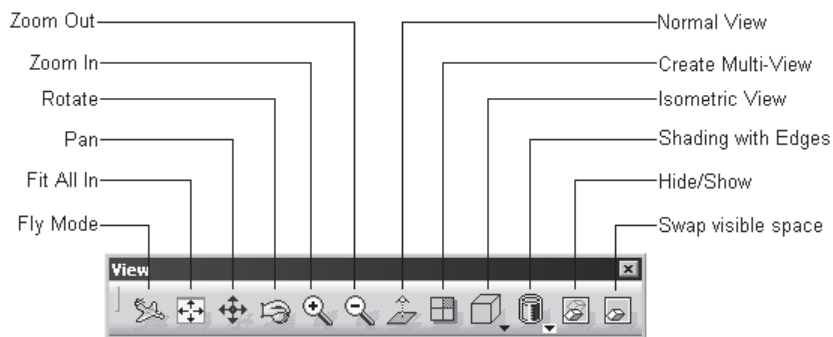


Figure 1-31 The **View** toolbar

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 the 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.

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 from the **View** toolbar once 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 into the sketches 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 is 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 rotates while working in the **Sketcher** workbench, you can choose the **Normal View** button to reorient it normal to the sketching plane.

Hide/Show

Menu: View > Hide/Show > Hide/Show
Toolbar: View > Hide/Show



To hide sketcher elements or geometric elements, invoke the **Hide/Show** tool, by choosing the **Hide/Show** button from the **View** toolbar. You are prompted to select an element. Click on the element to be hidden from the geometry area. You will notice that the particular element is no longer visible. The following tool explains how to redisplay the hidden element.

Swap Visible Space

Menu: View > Hide/Show > Swap visible space
Toolbar: View > Swap visible space



The hidden elements are stored in a space other than 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 return back 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 back to the visible geometry area.

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

Evaluation Chapter. Logon to www.cadcam.com for more details

TUTORIALS

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in Figure 1-32. The sketch is shown in Figure 1-33. You will not dimension the it. The solid model and dimensions are given for your reference. **(Expected time: 30 min)**

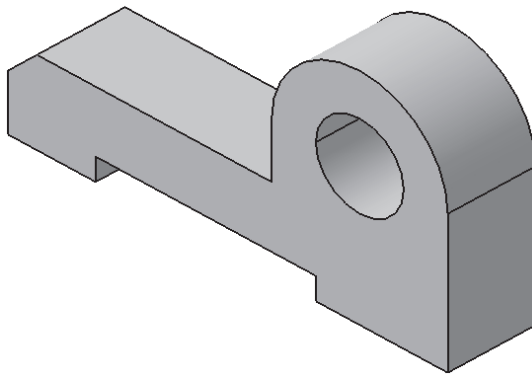


Figure 1-32 The solid model for Tutorial 1

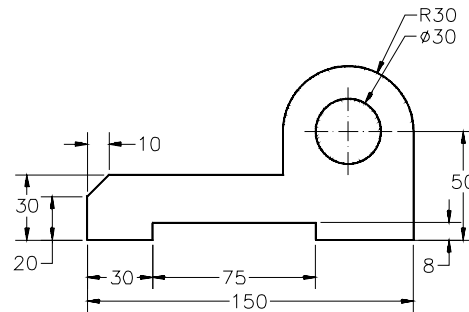


Figure 1-33 The sketch of the model

The following steps are required to complete this tutorial:

- Start CATIA V5 and then start a new CATPart file.
- Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, refer to Figures 1-36 and 1-37.
- Save and close the file.

Starting CATIA V5 and Starting a New Part File

- Start CATIA V5 by choosing **Start > Programs** (or **All Programs**, if you are working with Windows XP) > **CATIA > CATIA V5R14** or by double-clicking on the shortcut icon of CATIA V5R14 on the desktop of your computer.

A new **Product1** file is started.

- Choose **File > Close** from the menu bar; the start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Part Design** to make sure that you are in the **Part Design** workbench; the **Part name** dialog box is displayed, as shown in Figure 1-34.

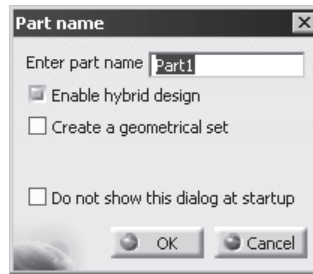


Figure 1-34 The **Part name** dialog box

3. Choose the **OK** button; a new file in the **Part Design** workbench is opened.
4. To invoke the **Sketcher** workbench, choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane. The displayed screen is shown in Figure 1-35.

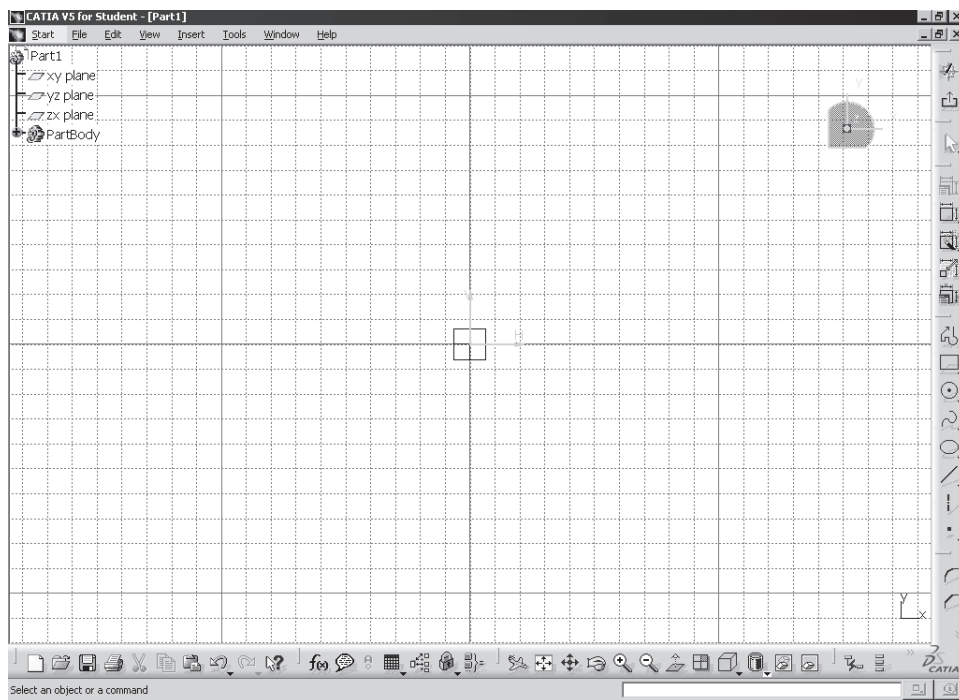


Figure 1-35 The **Sketcher** workbench screen

You will draw the sketch in two sections, first as the outer loop and second as the inside circle.

Drawing the Outer Loop of the Sketch

You should create the sketch symmetrically around the origin. This will reduce the time required to constrain and dimension it. The outer loop of the sketch can be drawn using the **Line** and the **Arc** tools. You will start drawing the outer loop from the lower left corner of the sketch.

1. Invoke the **Line** tool, by choosing the **Line** button 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 will be displayed, along with the cursor.
4. Click at the point whose coordinates are -50mm, -30mm. Next, move the cursor horizontally toward the right.



You will notice that the color of the line turns blue.



Note

A line that turns blue, while drawing, implies that it is constrained. The constraint may be horizontal or vertical, depending on the direction, in which the line is drawn.

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

Refer to Figure 1-32. The length of the first horizontal line at the lower left corner of the sketch is 30mm. Therefore, move the cursor until the length of the line is shown as 30mm in the **L** edit box of the **Sketch tools** toolbar.

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.

The first horizontal line is drawn. You will notice a **Horizontal** constraint is applied to it. After the line is drawn, it is still active and is displayed in orange. Click in the geometry area to remove it from the selection set.

As soon as you specify the endpoint of the line, the **Line** tool is terminated. Therefore, you need to choose this button again and again to draw multiple lines. You can avoid this by double-clicking on the **Line** button in the **Profile** toolbar. Now, the line tool will not be terminated until you press the ESC key twice.

6. Double-click on the **Line** button to invoke the **Line** tool and select the endpoint of the first horizontal line.
7. Press the TAB key thrice to highlight the value displayed in the **L** edit box of the **Sketch tools** toolbar. Type 8 in this edit box and press the ENTER key.

8. Now, move the cursor vertically upward and click when a vertical line is displayed.

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

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. Move the cursor horizontally toward the right and click when a horizontal line is displayed.

This draws the second horizontal line of length 75mm.

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 length of the line in the **L** edit box shows a value of 8mm.

This draws the second vertical line of length 8mm.

**Note**

You will notice that while drawing the second vertical line, the inferencing line is displayed in the geometry area. This line is often displayed, whenever the endpoint of a line is constrained, with an element already available in the sketch.

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 when the length of the line is 50mm.

This draws the third vertical line of length 50mm. Next, you will draw an arc.

13. To draw the arc, first invoke the **Circle** toolbar, by choosing the down arrow on the right of the **Circle** button, in the **Profile** toolbar. Choose the **Three Point Arc** button to invoke the **Three Point Arc** tool.

14. Select the start point of the arc as the endpoint of the previous vertical line and click on it.

15. Move the cursor to a point whose coordinates are 70mm, 50mm. These are displayed in the **Sketch tools** toolbar and also on top of the cursor. Click on this point to define 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 40mm, 20mm in the geometry area. The coordinate values are displayed on top of the cursor.

This draws the arc for the outer loop. The arc is in the selection mode; click anywhere in the geometry area to end the selection mode. Now, to continue drawing the outer loop, you need to invoke the **Line** tool again.

17. Double-click on the **Line** button from 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 when the value of the length line is 20mm in the **L** edit box of the **Sketch tools** toolbar.

This draws the fourth vertical line of length 20mm. The line is no longer in the selection mode. You are prompted to enter the start point of the next line.

19. Select the endpoint of the previous line as the start point of the fourth horizontal line. Move the cursor horizontally toward the left. Click when the length of the line in the **L** edit box of the **Sketch tools** toolbar shows a value of 80mm.

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

20. Select the endpoint of the previous line as the start point of the inclined line. Move the cursor such that the line is drawn at an angle of 225-degree. The current angle will be displayed in the **A** edit box of the **Sketch tools** toolbar. Click when a vertical inferencing line is displayed between the endpoint of the inclined line and the start point of the first horizontal line. This draws the inclined line of a horizontal length of 10mm.
21. Select the endpoint of the inclined line as the start point of the next line. Move the cursor vertically downward. Click when the length of the line in the **L** edit box shows a value of 20mm.

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

22. Choose the **Fit All In** button from the **View** toolbar to fit the current sketch on the screen.



The completed outer loop of the sketch is shown in Figure 1-36. The display of the constraints is turned off using the **Hide/Show** tool.

Drawing the Inner Circle

The circle will be drawn using the **Circle** tool.

1. Choose the **Circle** button from the **Circle** toolbar to invoke the **Circle** tool; you are prompted to define the center point of the circle.
2. Move the cursor to a point whose coordinates are 70mm, 20mm. Click, when the cursor snaps to this point.



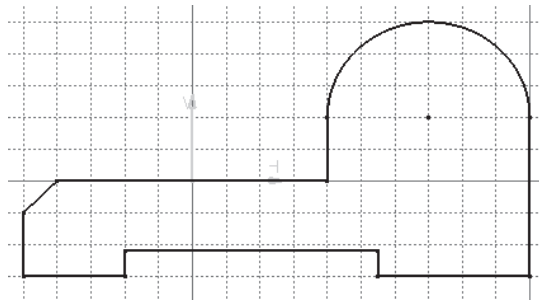


Figure 1-36 The outer loop of the sketch

3. Move the cursor horizontally toward the right and click when the radius of the circle in the **R** edit box of the **Sketch tools** toolbar shows a value of 15mm. Click anywhere to remove the circle from the selection.

This completes the sketch for Tutorial 1. The final completed sketch, with the display of constraints turned on, is shown in Figure 1-37.

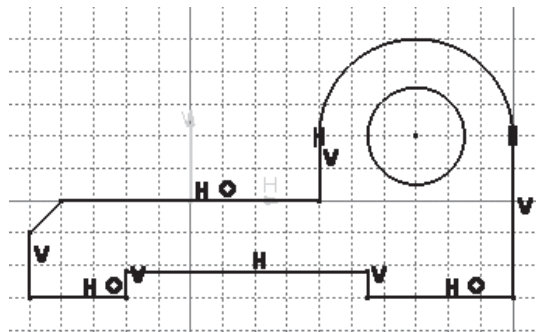



Figure 1-37 The final sketch for Tutorial 1

Saving and Closing the Sketch

After completing the sketch, you need to save it. Save each tutorial of this chapter in the *c01* folder, which is in the *CATIA* folder.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create the *CATIA* folder inside the *\My Documents* folder. Then create the *c01* folder inside the *CATIA* folder. 
2. Enter the name of the file as *c01tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.
3. Close the part file by choosing **File > Close** from the menu bar.



Tip. If you open a file that was saved in the **Sketcher** workbench, it will be opened in the **Sketcher** workbench only and not in the **Part Design** workbench.

Tutorial 2

In this tutorial, you will draw the sketch of the model shown in Figure 1-38. The sketch is shown in Figure 1-39. You will not dimension it. The solid model and dimensions are given for your reference.
(Expected time: 30 min)

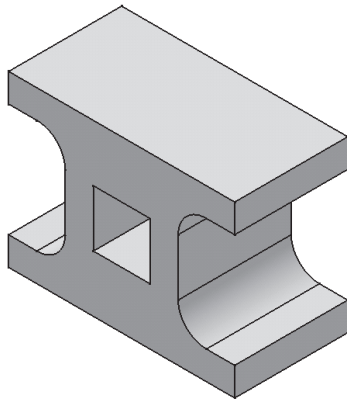


Figure 1-38 The solid model for Tutorial 2

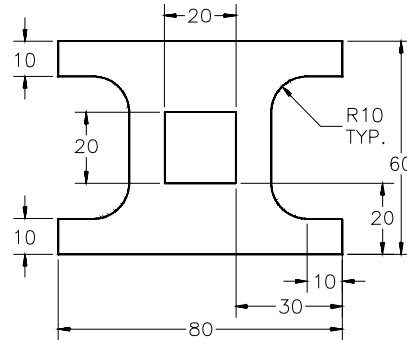


Figure 1-39 The sketch of the model

The following steps are required to complete this tutorial:

- Start a new CATPart file.
- Draw the sketch of the model using the **Profile** and **Rectangle** tool, refer to Figures 1-41 through 1-43.
- Save and close the file.

Starting a New Part File

- Choose **File > New** from the menu bar; the **New** dialog box is displayed, as shown in Figure 1-40.

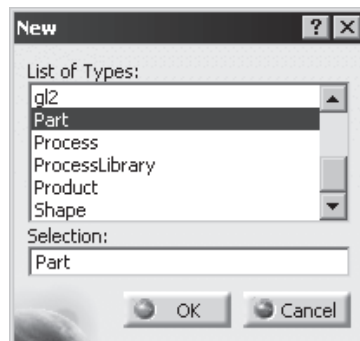



Figure 1-40 The New dialog box


- Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button; the **Part name** dialog box is displayed.

3. Accept the default options in the **Part name** dialog box and choose the **OK** button to open a new file in the **Part Design** workbench.
4. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane to invoke the **Sketcher** workbench. 

You will draw the sketch in two sections, first the outer loop and next the inner cavity.

Drawing the Outer Loop of the Sketch

You will draw the outer loop of the sketch using the **Line** and **Arc** tools. Start drawing the outer loop from the left corner of the sketch. It is recommended to keep the origin in the middle of the drawn sketch, as this will reduce the time required for constraining and dimensioning the sketches. This will also help you to capture the design intent easily.

1. Invoke the **Profile** tool from the **Profile** toolbar.
2. Move the cursor to the third quadrant. The coordinates of the point will be displayed above the cursor. 
3. Specify the start point of the line at the point whose coordinates are -40, -30 and then move the cursor horizontally toward the right.

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

4. Move the cursor to a location whose coordinates are 40, -30. The coordinates of the point can be seen on top of the cursor.
5. Specify the endpoint of the line at this location. A rubber band line is attached to the cursor. Move the cursor vertically upward.
6. Specify the endpoint of the second line on the point whose coordinates are 40mm, -20mm.

A rubber band line is attached to the cursor.

7. Move the cursor horizontally toward the left and specify the endpoint of the third line where the value of the coordinates is 30, -20.

After drawing these three lines, draw a tangent arc using the **Tangent Arc** option in the **Profile** tool.

8. Choose the **Tangent Arc** button in the **Sketch tools** toolbar.
9. Move the cursor to a location whose coordinates are 20, -10 and specify the endpoint of the tangent arc. Figure 1-41 shows the sketch, after drawing the three lines and tangent arc. The system switches back to the **Line** mode.
10. Move the cursor vertically upward to a location whose coordinates are 20, 10.

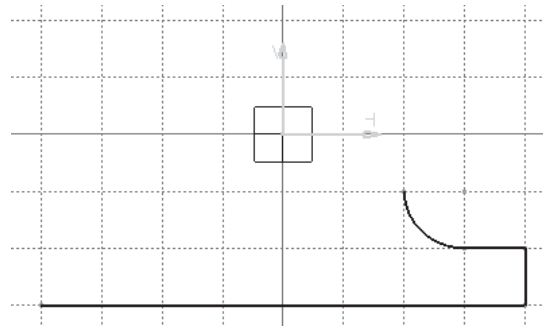


Figure 1-41 The sketch after drawing the three lines and a tangent arc

11. Specify the endpoint of the line at this location.

Next, you need to draw a tangent arc by switching to the arc mode using the **Tangent Arc** option in the **Profile** tool.

12. Choose the **Tangent Arc** button from the **Sketch tools** toolbar.



13. Move the cursor to a location whose coordinates are 30, 20 and specify the endpoint of the tangent arc.

The system switches back to the **Line** mode.

14. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is 40, 20.
15. Move the cursor vertically upward and specify the endpoint of the line, when the value of the coordinates is 40, 30.
16. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is -40, 30.
17. Move the cursor vertically downward and specify the endpoint of the line, when the value of the coordinates is -40, 20.
18. Move the cursor horizontally toward the right and specify the endpoint of the line, when the value of the coordinates is -30, 20.

Next, you need to draw a tangent arc by switching to the tangent arc mode.

19. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch to the tangent arc mode.



20. Move the cursor to a location whose coordinates are -20, 10 and specify the endpoint of arc at this location.

The system switches back to the Line mode.

21. Move the cursor vertically downward and specify the endpoint of the line, where the value of the coordinates is -20, -10.
22. Switch to the Tangent mode and move the cursor to a location whose coordinates are -30mm, -20mm. Specify the endpoints of the tangent arc at this location.
23. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of coordinates is -40, -20.
24. 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 completing the outer loop and hiding the constraints, is shown in Figure 1-42.

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 a location whose coordinates are -10, 10. Specify the upper left corner of the rectangle at this location.
3. Move the cursor to a location whose coordinates are 10, -10. 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 inside the geometry area.



The final sketch, after drawing the inner loop, is shown in Figure 1-43. Note that the display of the constraints has been turned on.

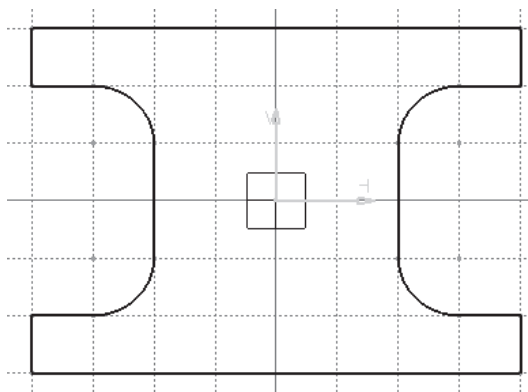


Figure 1-42 The sketch, after drawing the outer loop of the sketch

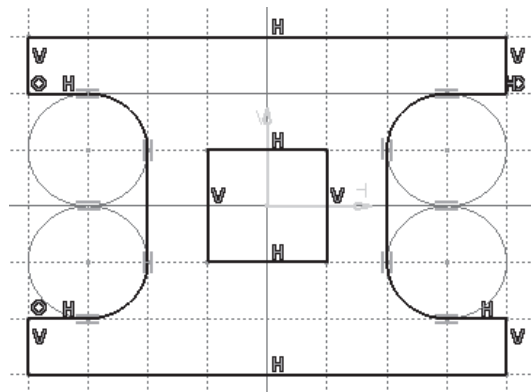



Figure 1-43 The final sketch after drawing the inner loop of the sketch

Saving the Sketch

After completing the sketch, you need to save it. As mentioned earlier, you need to save each tutorial of this chapter in the *c01* folder in the *CATIA* folder.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse the folder named *c01* that you created in the last tutorial. 
2. Enter the name of the file as *c01tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

Tutorial 3

In this tutorial, you will draw the sketch of the model shown in Figure 1-44. The sketch is shown in Figure 1-45. You will not dimension it. The solid model and dimensions are given (Expected time: 30 min)

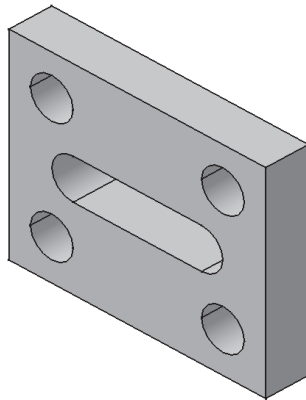


Figure 1-44 The solid model for Tutorial 3

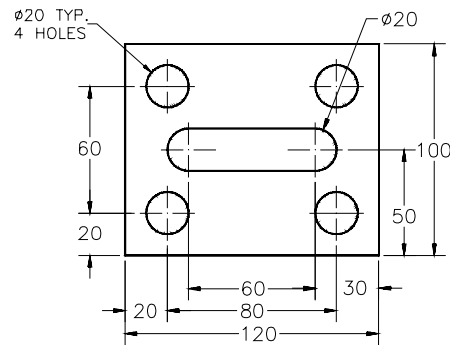


Figure 1-45 The sketch for the solid model

The following steps are required to complete this tutorial:

- a. Start a new CATPart file.
- b. Draw the sketch of the model using the **Rectangle**, **Profile**, and **Circle** tools, refer to Figures 1-46 through 1-48.
- c. Save the sketch and close the file.

Starting a New Part File

1. Choose **File > New** from the menu bar; the **New** dialog box is displayed.
2. Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button; the **Part name** dialog box is displayed.
3. Choose the **OK** button to open a new file in the **Part Design** workbench.

4. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane.



This sketch will be drawn in two parts. Initially, you will draw the outer loop of the sketch, that is, a rectangle. Next, you need to draw the inner loops of the sketch, which consists of four holes and an elongated hole. First, you will draw an elongated hole using the **Profile** tool and then the four holes using the **Circle** tool.

Drawing the Outer Loop of the Sketch

The outer loop of the sketch will be drawn using the **Rectangle** tool.

1. Choose the **Rectangle** button from the **Profile** toolbar.
2. Move the cursor to a 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 1-46 shows the outer loop of the sketch drawn using the **Rectangle** tool.

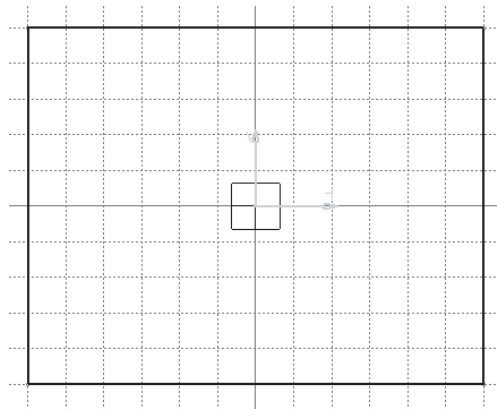


Figure 1-46 The outer loop of the sketch


Drawing the Inner Loop of the Sketch

After drawing the outer loop of the sketch, draw its inner loop.

1. Choose the **Profile** button from the **Profile** toolbar.
2. Move the cursor to a location whose coordinates are -30, 10 and specify 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 **Sketch tools** toolbar to switch over to the arc mode. 
5. Move the cursor to a location whose coordinates are 30, -10 and specify the endpoint of the tangent arc.

The system switches over to the **Line** mode.

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 over to the arc mode.
8. Move the cursor to the start point of the first horizontal line of the elongated hole. Specify the endpoint of the arc when it snaps to the start point.

The sketch, after drawing the elongated hole, is shown in Figure 1-47.

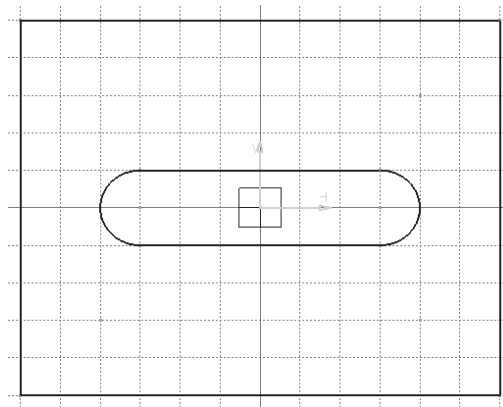


Figure 1-47 The sketch after drawing the elongated hole

9. Choose the **Circle** button from the **Circle** toolbar.




Note

*If you have closed the **Circle** toolbar, you can use the **Profile** toolbar to invoke the **Line** tool.*

10. Move the cursor to a location whose coordinates are 40, 30 and specify the center point of the circle.
11. Specify the value of **10** as the radius in the **Radius** edit box of the **Sketch tools** toolbar.

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

12. Choose the **Circle** button from the **Circle** toolbar. 
13. Move the cursor to a location whose coordinates are 40, -30 and specify the center point of the circle.
14. Specify 10mm as the value of the radius in the **Radius** edit box of the **Sketch tools** toolbar.
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 1-48.

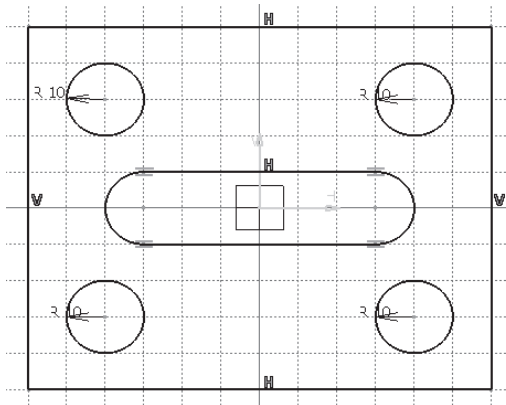



Figure 1-48 The final sketch

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Browse the folder named *c01* that you created in the first tutorial. 
2. Enter the name of the file as *c01tut3* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.
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 1-49. The sketch is shown in Figure 1-50. You will not dimension it. The solid model and dimensions are given for your reference. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Start a new CATPart file.
- b. Draw the sketch of the model using the **Profile** and the **Circle** tools, refer to Figures 1-51 and 1-52.
- c. Save the sketch and close the file.

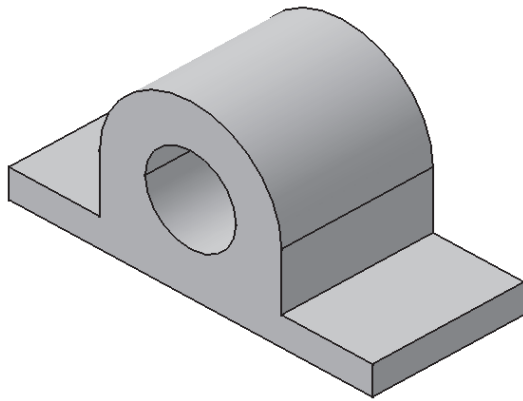


Figure 1-49 The solid model for Tutorial 4

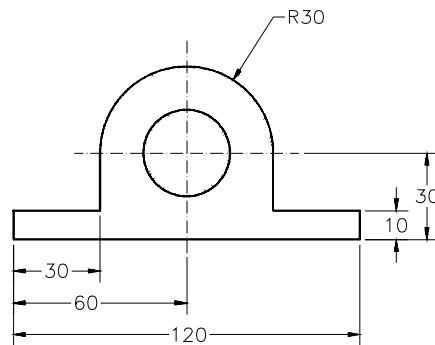



Figure 1-50 The sketch for the solid model


Starting a New Part File

1. Choose **New** from **File** menu; the **New** dialog box is displayed.
2. Select **Part** from the **List of Types** list box from this dialog box. Choose the **OK** button; the **Part name** dialog box is displayed.
3. Choose the **OK** button to open a new file in the **Part Design** workbench.
4. Choose the **Sketcher** button from the **Sketcher** toolbar and then select the YZ plane as the sketching plane to invoke the **Sketcher** workbench. 

This sketch will be drawn in two parts. Initially, you will draw the outer loop of the sketch using the **Profile** tool and then the inner loop of the sketch, which is a hole.


Drawing the Outer Loop of the Sketch

The outer loop of the sketch will be drawn using the **Profile** tool. In this sketch, the lower-left corner of the sketch is coincident to the origin of the **Sketcher** workbench. The resulting sketch will be drawn in the first quadrant.

1. Choose the **Profile** button from the **Profile** toolbar. 
2. Move the cursor to a location whose coordinates are 0, 0 and specify 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 120, 0.
4. Move the cursor vertically upward and specify the endpoint of the line, when the value of the coordinates is 120, 10.
5. Move the cursor horizontally toward the left and specify the endpoint of the line, when the value of the coordinates is 90, 10.

6. 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, draw the tangent arc using the **Tangent Arc** option from the **Sketch tools** toolbar.

7. Choose the **Tangent Arc** button from the **Sketch tools** toolbar to switch to the **Tangent Arc** mode. 
8. Move the cursor to a location whose coordinates are 30, 30 and specify the endpoint of the tangent arc at this location.

The system switches back to the **Line** mode.

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

The sketch, after drawing the outer loop, is shown in Figure 1-51.

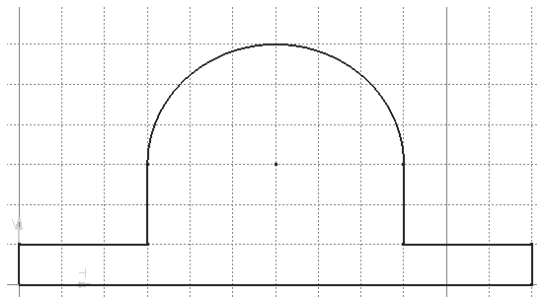



Figure 1-51 The sketch, after drawing the outer loop

Drawing the Inner Loop of the Sketch

The inner loop of the sketch consists of a circle that will be drawn, using the **Circle** tool, concentric to the arc of the outer loop.

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

The final sketch, after drawing the inner loop is shown in Figure 1-52. Note that the display of constraints is turned on in this figure.

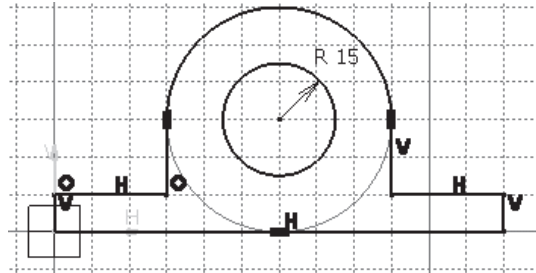



Figure 1-52 The final sketch

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box and browse the *c01* folder. 
2. Enter the name of the file as *c01tut4* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c01* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with those given at the end of this chapter.

1. The base feature of any design is a sketched feature, which is created by drawing the sketch. (T/F)
2. You can also draw an arc, while working with the **Profile** tool. (T/F)
3. To enter the **Sketcher** workbench you need to choose the **Sketcher** button. (T/F)
4. When you open a file that has been saved in the sketching environment, it opens in the part modeling environment. (T/F)
5. You can convert a sketched element into a construction element by using the _____ button.
6. To draw a rectangle at an angle, you need to use the _____ tool.
7. The _____ are the temporary lines that are used to track a particular point on the screen.
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 rectangle is considered as a combination of individual _____.

REVIEW QUESTIONS

Answer the following questions.

1. The 3 point arcs are the ones that are drawn by defining the start point, endpoint, and a point on the arc. (T/F)
2. The **Parallelogram** button is available in the **Predefined Profile** toolbar. (T/F)
3. The **Symmetrical Extension** button, when selected 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 the radius in the dialog box that is displayed. (T/F)
5. When you start CATIA V5, by default, a file in the **Product** workbench is started. (T/F)
6. In CATIA V5, a rectangle is considered as a combination of which of the following elements?
 - (a) Lines
 - (b) Arcs
 - (c) Splines
 - (d) None
7. Which tool 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
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, can you draw the sketches that can be used to create features?
 - (a) **Part**
 - (b) **Assembly**
 - (c) **Shape**
 - (d) None

EXERCISES

Exercise 1

Draw the sketch of the model shown in Figure 1-53. The sketch to be drawn is shown in Figure 1-54. Do not dimension it. The solid model and dimensions are given for your reference.
(Expected time: 30 min)

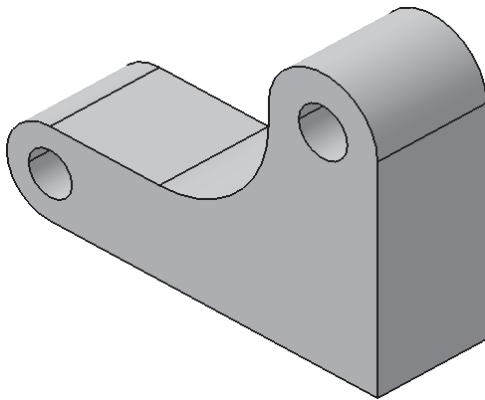


Figure 1-53 The solid model for Exercise 1

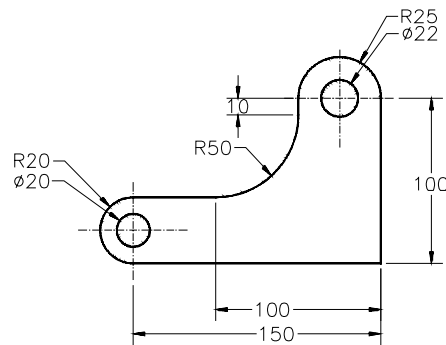


Figure 1-54 The sketch of the model

Exercise 2

Draw the sketch of the model shown in Figure 1-55. The sketch to be drawn is shown in Figure 1-56. Do not dimension it. The solid model and dimensions are given for your reference.
(Expected time: 30 min)

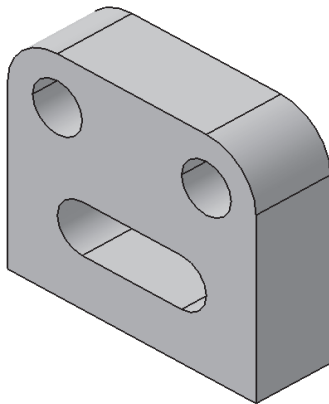


Figure 1-55 The solid model for Exercise 2

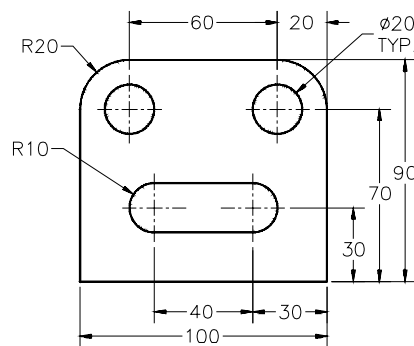


Figure 1-56 The sketch of the model

Answers to Self-Evaluation Test

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