



Chapter 2

Sketching, Dimensioning, and Creating Base Features and Drawings

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the Sketcher workbench of CATIA V5-6R2025*
- *Start a new file in the Part workbench*
- *Invoke the Sketcher workbench*
- *Set up the Sketcher workbench*
- *Understand the terms used in the Sketcher workbench*
- *Draw sketches using the tools in the Sketcher workbench*
- *Use some of the drawing display tools*
- *Dimension a sketch*
- *Extrude a sketch*
- *Generate drawing views*

THE SKETCHER WORKBENCH

Most components designed using CATIA V5-6R2025 are a combination of sketched features, placed features, and derived features. Placed features are created without drawing a sketch whereas 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 a chain link shown in Figure 2-1. The base sketch to create this solid model is shown in Figure 2-2.

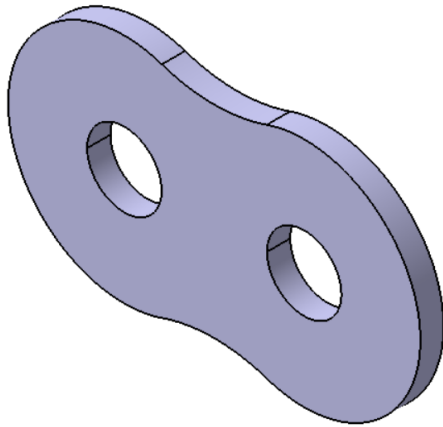


Figure 2-1 Solid model of the chain link

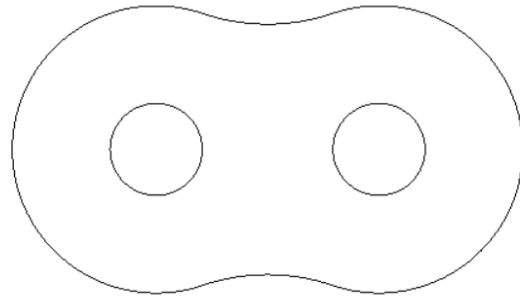


Figure 2-2 Base sketch for the solid model

The **Sketcher** workbench provides the space and tools to draw the 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 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 V5-6R2025, the initial interface of CATIA V5-6R2025 is displayed, as shown in Figure 2-3. Start a new file in the **Part Design** workbench.

Choose **Part Design** from **Start > Mechanical Design**; the **New Part** dialog box will be displayed, as shown in Figure 2-4. Enter part name in the **Enter part name** edit box and select the **Enable hybrid design** radio button if not selected by default. Choose the **OK** button; the **Part Design** workbench will be displayed. 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 available in this dialog box.

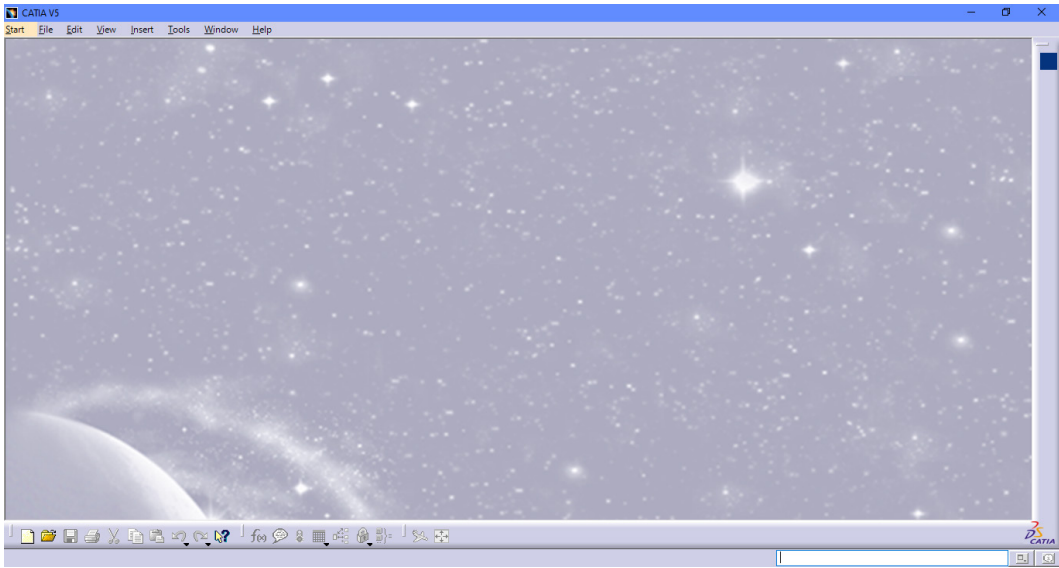


Figure 2-3 Initial interface of CATIA V5-6R2025

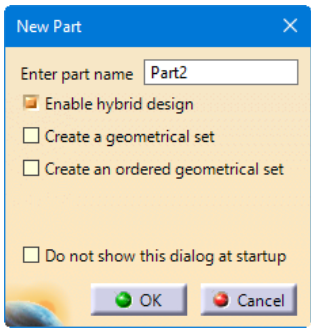


Figure 2-4 The New Part dialog box

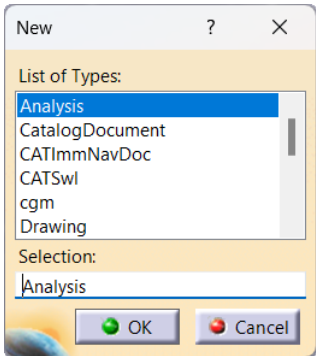


Figure 2-5 The New dialog box

Next, choose **OK** button; the **New Part** dialog box will be displayed. Enter the file name in the **Enter part name** edit box 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.

If you select the **Enable hybrid design** check box in the **New Part** dialog box, you will be able to work in a hybrid design mode. In this design mode, part body includes solid, wireframe, and surface elements. The color of the **PartBody** node icon will be green in both hybrid and non-hybrid design modes. But, when both design modes are opened together, the color of the **PartBody** node icon will be displayed as green in the hybrid design mode and as gray in the non hybrid design mode.

**Note**

The color of the **PartBody** node icon will be gray only when the **Enable hybrid design inside part bodies and bodies** check box is selected in the **Hybrid design** area of the **Part Document** tab of the **Part Infrastructure** sub-node of the **Infrastructure** node in the **Options** dialog box.

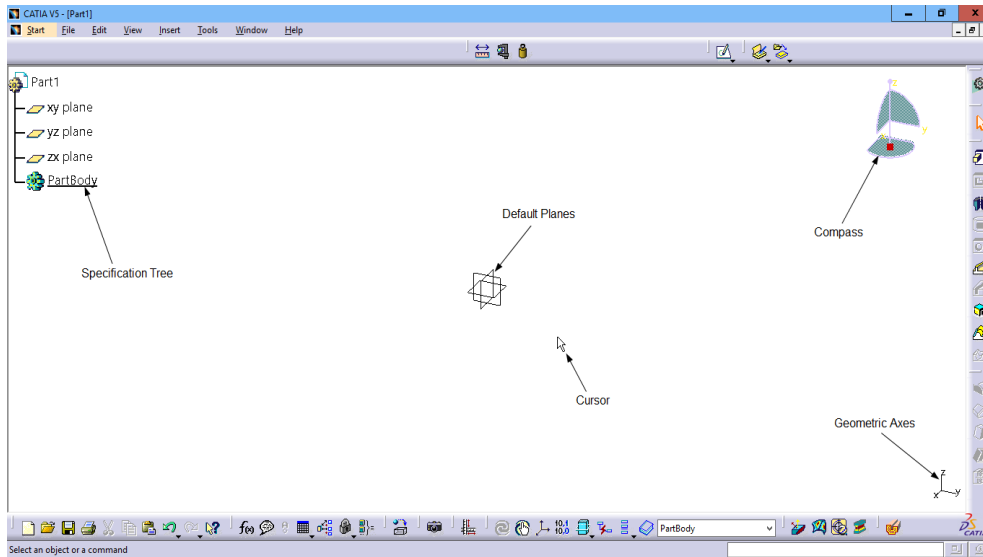


Figure 2-6 A new file opened in the **Part Design** workbench

If you select the **Create a geometrical set** check box, you will be able to create a geometrical set as well as a new part. Geometrical set created with this option is located above the Part Body node in the specification tree.

If you select the **Create an ordered geometrical set** check box, you will be able to create an ordered geometrical set as well as a new part. The ordered geometrical set created using this option are located under the PartBody node in the specification tree.

If you clear the **Enable hybrid design** check box from the **New Part** dialog box, the new file will start in the conventional design mode.

**Note**

1. In this textbook, the hybrid design mode has been used. Therefore, it is recommended that you keep the **Enable hybrid design** check box selected whenever you start a new file.

2. You can hide the Compass, the 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 indicating 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.

INVOKING THE SKETCHER WORKBENCH

Sketch is the basic step 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** sub-toolbar will appear, as shown in Figure 2-7. The two buttons in the **Sketcher** sub-toolbar are **Sketch** and **Positioned Sketch**. The next section focuses on invoking the **Sketcher** workbench using these two buttons.

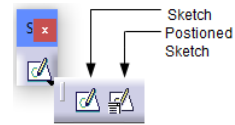


Figure 2-7 The *Sketcher* sub-toolbar

Invoking the Sketcher Workbench using the Sketch Tool



To invoke the **Sketcher** workbench using this method, choose the **Sketch** tool from the **Sketcher** sub-toolbar; you will be prompted to select a plane, a planar face, or a sketch.

Select a plane from the three default planes in the Specification tree or from the geometry area; the **Sketcher** workbench will be invoked and the selected plane will be oriented parallel to the screen, refer to 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.

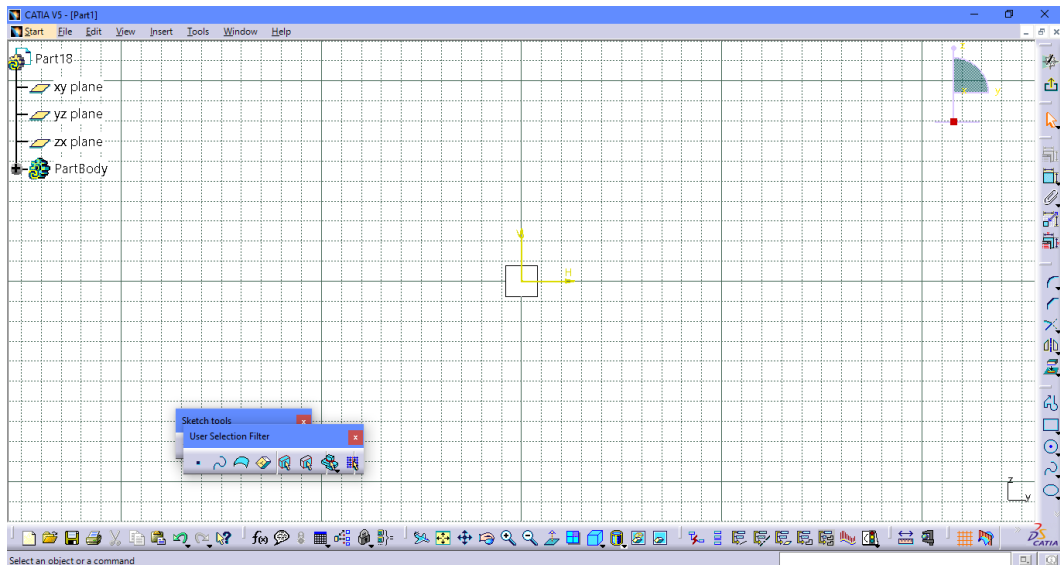



Figure 2-8 The *Sketcher* workbench invoked on selecting a plane as the sketching plane



Note

Remember once the **Sketcher** workbench is invoked, you will stay in the **Select** mode till you exit the **Sketcher** workbench. To exit, choose the **Exit workbench**  tool from the **Workbench** toolbar.

Invoking the Sketcher Workbench using the Positioned Sketch Tool



In CATIA V5, you can also create a user-defined absolute axis while invoking the **Sketcher** workbench by using the **Positioned Sketch** tool. To invoke the **Sketcher** workbench using this option, choose the **Positioned Sketch** tool from the **Sketcher** sub-toolbar; 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.

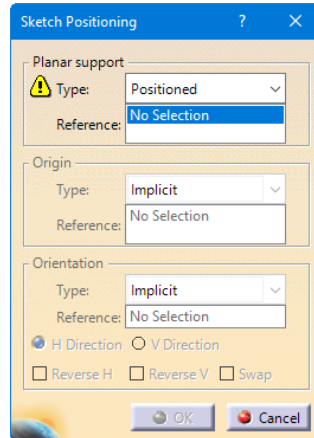


Figure 2-9 The Sketch Positioning dialog box

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.

Modifying Units

To modify units, invoke the **Options** dialog box by choosing **Options** from the **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.

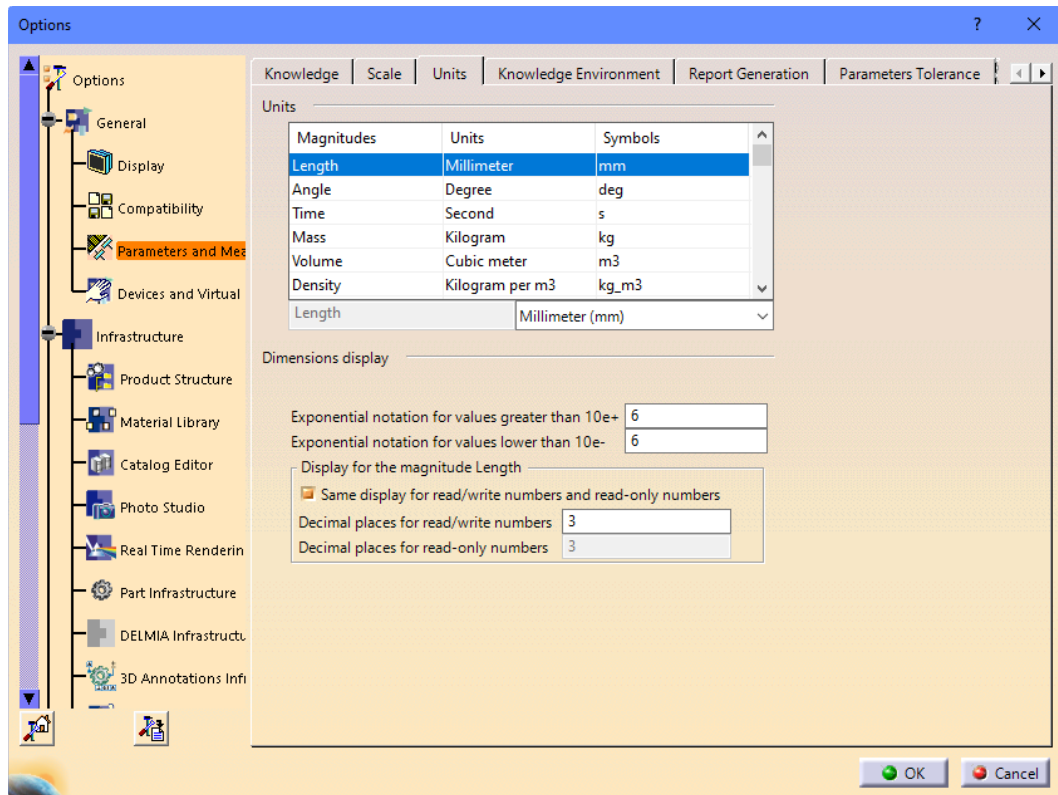


Figure 2-10 The Options dialog box with the Units tab chosen

Modifying the 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 horizontal and vertical lines are continuous and dotted lines. The spacing between two continuous lines is called primary spacing and the spacing between two dotted lines is called graduation. The mesh that is formed due to the intersection of these lines in the horizontal and vertical 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, refer to Figure 2-11.

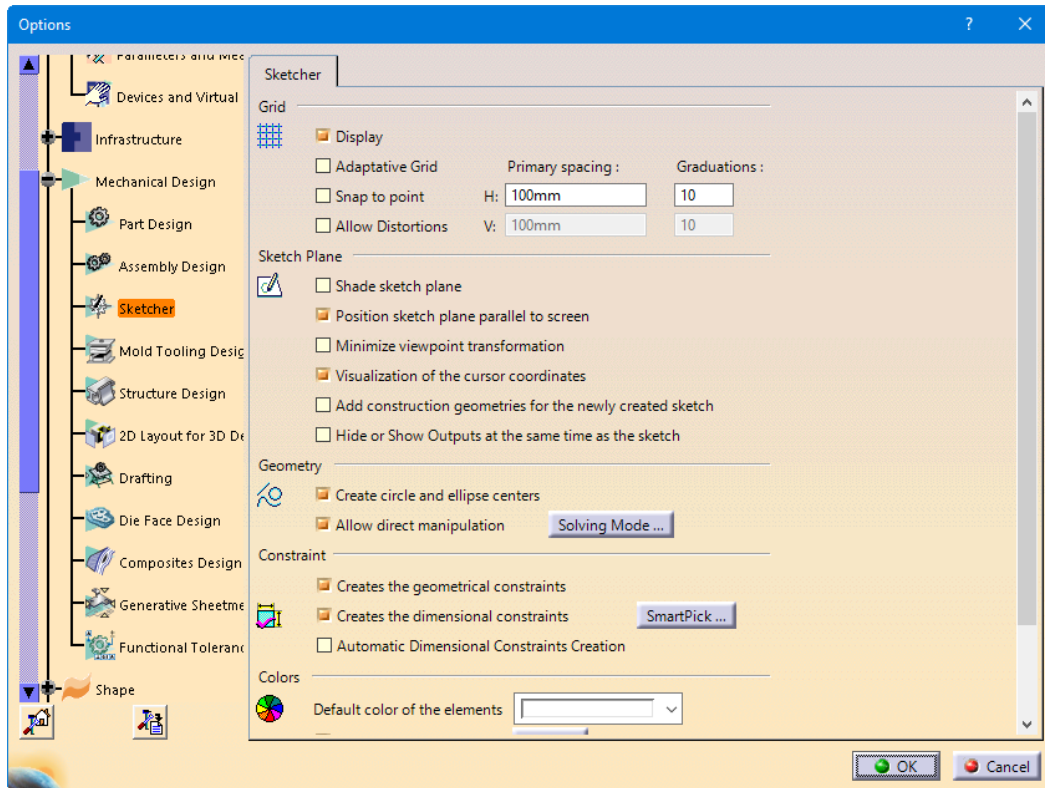


Figure 2-11 The Options dialog box with the Sketcher option chosen

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, henceforth all the files that you open or start in the **Sketcher** workbench 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 under the **PartBody** node. Click on the + sign on the left 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.

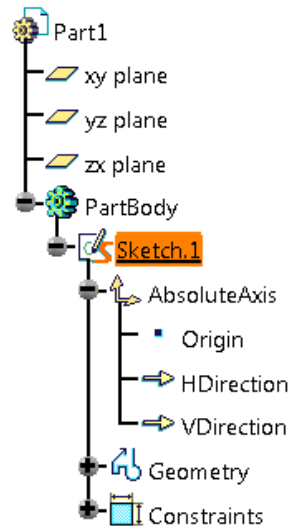


Figure 2-12 The expanded Specification tree

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. Notice the + sign available on the left of **AbsoluteAxis** in the Specification tree. Click on 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, if the branch lines are accidentally clicked then the Specification tree will get activated. Consequently, the geometry area will be frozen. Now, zooming and panning will resize or reposition the Specification tree instead of the geometry view. The geometry area can be activated again 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). It is widely used while applying dimensional constraints to the sketches. You will learn more about dimensional constraints in later chapters.

HDirection

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

VDirection

The direction parallel to vertical axis is referred to as **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 **Visualization** toolbar which appears only when you are in 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 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.



Figure 2-13 The **Select** toolbar

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** sub-toolbar will be displayed, as shown in Figure 2-14. The methods of selecting an entity using the tools in the **Select** toolbar are discussed next.

**Note**

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

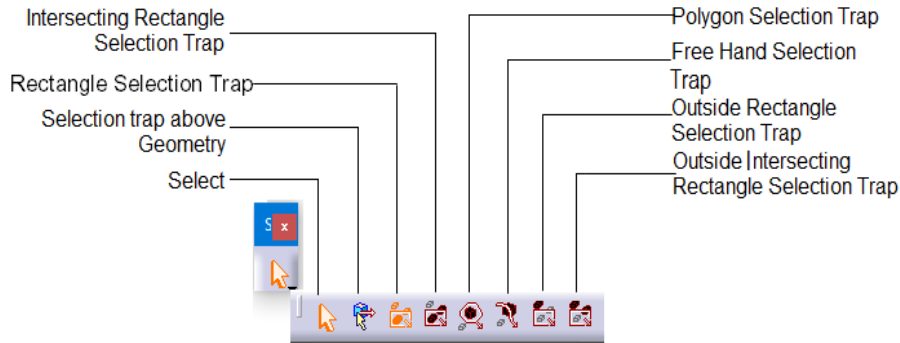


Figure 2-14 The *Select* sub-toolbar

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. Click on the element to select it.

Rectangle Selection Trap



This tool is used to select 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. This tool is active by default. If not, choose the **Rectangle Selection Trap** tool from the **Select** toolbar. Note that this method of selection will not allow you to start creating trap from the top of an entity.

Selection trap above Geometry



This tool works same as the **Rectangle Selection Trap** tool with the only difference that this tool allows you to start creating trap from the top of given sketched entity.

Intersecting Rectangle Selection Trap



The **Intersecting Rectangle Selection Trap** tool is similar to the **Selection trap above Geometry** tool. The difference between them is that this tool allows you to select elements of a sketch that are inside or are intersected by the trap. To create an intersecting trap, choose the **Intersecting Rectangle Selection 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. To use this method, choose the **Polygon Selection 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 temporary lines that are used to track a particular point on the screen. When a sketching tool is selected in the sketcher environment, the inferencing lines are automatically displayed from the endpoints of the sketched elements or from the origin. 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.

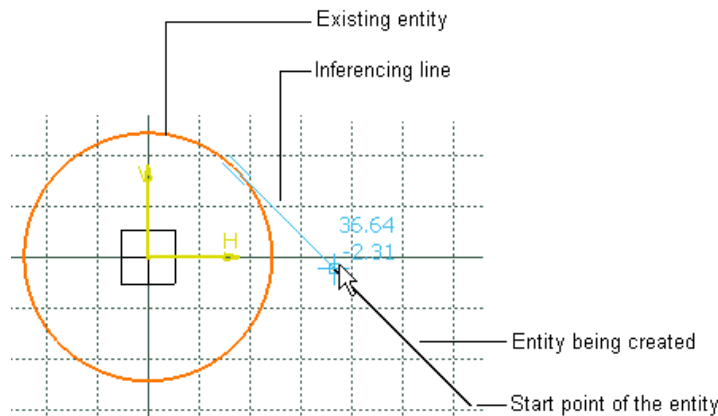


Figure 2-15 Use of inferencing line to draw a tangent line

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 sub-toolbar. As mentioned earlier, the name of the sub-toolbar will be the name of the toolbar that will be obtained by detaching it from the parent toolbar. The procedure to draw sketches using the sketch tools is discussed next.

Drawing Lines

Menubar: Insert > Profile > Line > Line
Toolbar: Profile > Line sub-toolbar > 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 in nature, it allows user to first draw a line of any length 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 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 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.

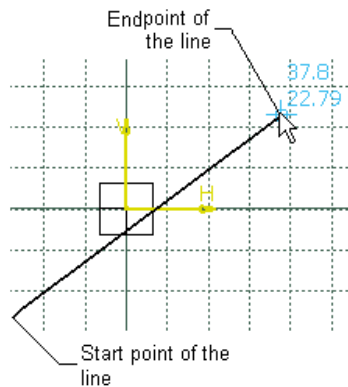


Figure 2-16 The line drawn by selecting its start and end points from the geometry area



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 **Sketch tools** toolbar after invoking the **Line** tool. The two methods to draw a line using the **Sketch tools** toolbar are discussed next.

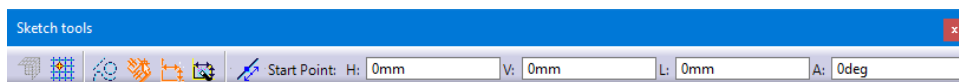


Figure 2-17 The expanded **Sketch tools** toolbar displayed after invoking the **Line** tool

Drawing Lines by Entering the Values of the Start and End Points

To draw a line using the start point and endpoint values, invoke the **Line** tool. On doing so, the **Sketch tools** toolbar will expand. In the **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 select the endpoint. Specify values in the **H** and **V** edit boxes and press Enter again; a line will be drawn in the geometry area corresponding to the values entered in the start point and endpoint edit boxes. Also, the horizontal and vertical dimensions of the start point and endpoint with respect to the origin are displayed. 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 by 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 Constraints** button is chosen in the **Sketch tools** toolbar, the specified dimension values for the start point and the 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** tool from the expanded **Sketch tools** toolbar. When you draw the line using this option, its total length is double the distance you have 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, line normal to curve, 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** sub-toolbar will appear, as shown in Figure 2-18. The types of lines that can be drawn using the tools available in the **Line** sub-toolbar are discussed next.

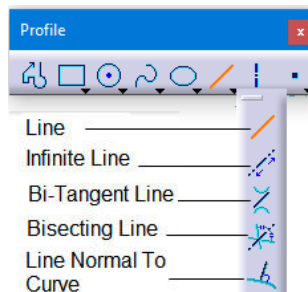


Figure 2-18 The **Line** sub-toolbar

**Tip**

The **Grid** button in the **Visualization** 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 Infinite Lines

Menubar: Insert > Profile > Line > Infinite Line
Toolbar: Profile > Line sub-toolbar > Infinite Line



To draw an infinite line, invoke the **Infinite Line** tool from the **Line** sub-toolbar in 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 by using the options in this toolbar.

Drawing Bi-Tangent Lines

Menubar: Insert > Profile > Line > Bi-Tangent Line
Toolbar: Profile > Line sub-toolbar > 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 **Line** sub-toolbar in 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 coincidence symbol will be displayed at the endpoints of the bi-tangent line. These are the coincidence constraints. You will learn more about these constraints in the later chapters.

Drawing Bisecting Lines

Menubar: Insert > Profile > Line > Bisecting Line
Toolbar: Profile > Line sub-toolbar > Bisecting Line



A bisecting line passes 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** tool from the **Line** sub-toolbar in the **Profile** toolbar. Select the first line and then select the second line; a bisecting line of infinite length will be drawn.

Drawing Lines Normal to a Curve

Menubar: Insert > Profile > Line > Line Normal To Curve
Toolbar: Profile > Line sub-toolbar > Line Normal To Curve



To draw a line normal to a curve, invoke the **Line Normal To Curve** tool from the **Line** sub-toolbar in the **Profile** toolbar; you will be prompted to select the curve. Specify the start point of the line anywhere on the periphery of the curve; you will be prompted to specify the other end point of the line. Click to specify the endpoint; a line normal to it will be drawn.

Drawing Center Lines

Menubar: 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 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.

Drawing Rectangles, Oriented Rectangles, and Parallelograms

CATIA V5 provides some set of tools that help you draw predefined profiles faster. These tools are grouped together in the **Predefined Profile** sub-toolbar. To view this sub-toolbar, choose the arrow on the right of the **Rectangle** tool in the **Profile** toolbar; the **Predefined Profile** sub-toolbar 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.

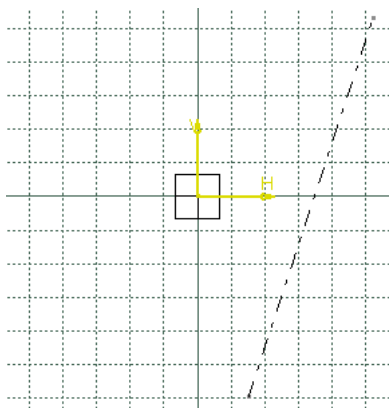


Figure 2-19 An axis drawn in the geometry area

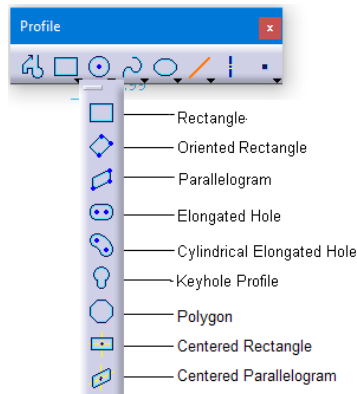


Figure 2-20 The **Predefined Profile** sub-toolbar

Drawing Rectangles

Menubar: Insert > Profile > Predefined Profile > Rectangle

Toolbar: Profile > Predefined Profile sub-toolbar > Rectangle



To draw a rectangle, invoke the **Rectangle** tool from the **Predefined Profile** sub-toolbar; refer to Figure 2-20; the **Sketch tools** toolbar will expand and you will be prompted to specify the first point for the 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 in the geometry area; preview of the rectangle is displayed. 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 using this method, you will notice that dimensions and constraints are applied to the resulting rectangle. You will learn more about dimensioning and constraining in the later chapters.

Drawing Oriented Rectangles

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



To draw an oriented rectangle, invoke the **Oriented Rectangle** tool from the **Predefined Profile** sub-toolbar in the **Profile** toolbar; you will be prompted to specify the start point.

Click in the geometry area to specify it; you will be prompted to specify the end point of the first side. 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.

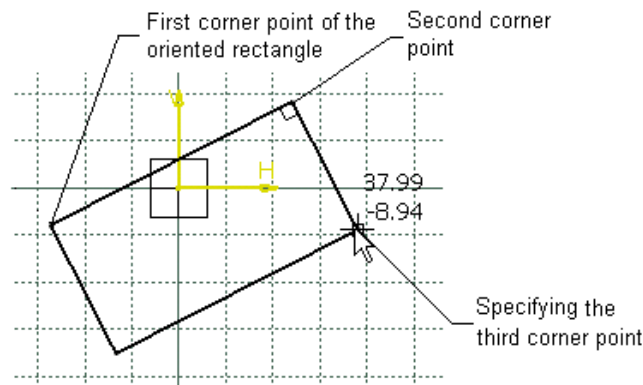


Figure 2-21 Oriented rectangle being drawn by specifying the corner points



Note

You can also use the **Sketch tools** toolbar to enter the coordinate values for the first, second, and third corners 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. Once you have specified the values, you need to press the Enter key to accept them.

Drawing Parallelograms

Menubar: Insert > Profile > Predefined Profile > Parallelogram
Toolbar: Profile > Predefined Profile sub-toolbar > 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** sub-toolbar in 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 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; 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, you can use the **Sketch tools** toolbar to enter the coordinate values of the corner points of the parallelogram. You can also enter the values for width, angle, and height of the parallelograms in the respective edit boxes in the expanded **Sketch tools** toolbar.

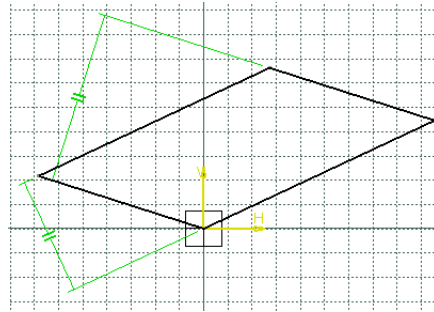


Figure 2-22 A parallelogram created by specifying the corner points

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** sub-toolbar will be displayed, as shown in Figure 2-23. The methods for creating points are discussed next.

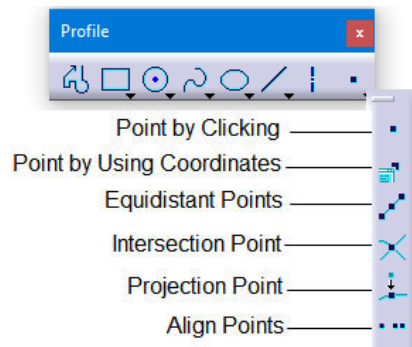


Figure 2-23 The **Point** sub-toolbar

Creating Points by Clicking

Menubar: Insert > Profile > Point > Point
Toolbar: Profile > Point sub-toolbar > Point by Clicking



To create points by clicking, invoke the **Point by Clicking** tool from the **Point** sub-toolbar in 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 to create the point. 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** sub-toolbar.

Creating Points by using Coordinates

Menubar: Insert > Profile > Point > Point by Using Coordinates
Toolbar: Profile > Point sub-toolbar > Point by Using Coordinates



To create points by using coordinates, invoke the **Point by Using Coordinates** tool from the **Point** sub-toolbar in the **Profile** toolbar; the **Point Definition** dialog box will be displayed, as shown in Figure 2-24. In this dialog box, you can use either cartesian (H and V) or polar coordinates to define a point. You can also select a previously created point as a reference for the point you want to create. On choosing the **Cartesian** tab, you need to specify **H** and **V** parameters or use the corresponding spinners. On choosing the **Polar** tab, you need to specify the values for **Radius** and **Angle** in the corresponding edit boxes.

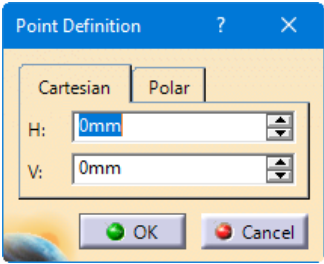


Figure 2-24 The *Point Definition* dialog box

Creating Equidistant Points

Menubar: Insert > Profile > Point > Equidistant Points
Toolbar: Profile > Point sub-toolbar > Equidistant Points



To create equidistant points on a line, curve, or between two points, invoke the **Equidistant Points** tool from the **Point** sub-toolbar in the **Profile** toolbar; you will be prompted to select the origin point or a curve on which the points are to be created. Select the object; the **Equidistant Point Definition** dialog box will appear, as shown in Figure 2-25. Specify the number of equidistant points to be created in the **New Points** spinner. If you select a line or a curve and then click on the extremity of that line or curve, the options in the **Parameters** drop-down will become available. You can select the **Points & Length** option from this drop-down. By using this option, you can specify the number of points and length in the **New Points** and **Length** spinners, respectively. If you select the **Points & Spacing** option, you can specify the number of points and spacing between the points in the **New Points** and **Spacing** spinners, respectively. If you choose the **Spacing & Length** option, you can specify the spacing and length for the

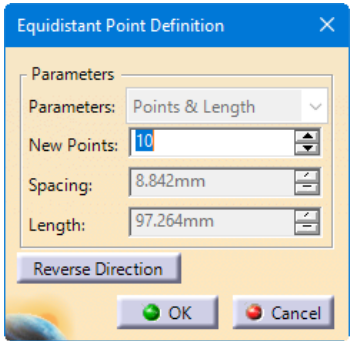


Figure 2-25 The *Equidistant Point Definition* dialog box

points. Choose the **Reverse Direction** button to reverse the direction of the points if required. Choose **OK** to create equidistant points, refer to Figure 2-26.

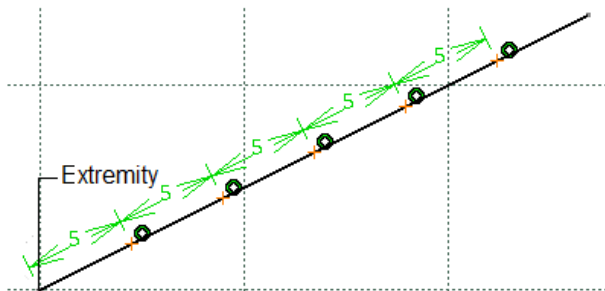


Figure 2-26 The equidistant points created on the line

Creating Intersection Points

Menubar: Insert > Profile > Point > Intersection Point
Toolbar: Profile > Point sub-toolbar > Intersection Point



To create points at the intersection of the selected elements, invoke the **Intersection Point** tool from the **Point** sub-toolbar of the **Profile** toolbar. On doing so, you will be prompted to select the set of the elements to be intersected with another element. Select the set of the elements; the intersection points will be created on the intersection of the selected elements, as shown in Figure 2-27.

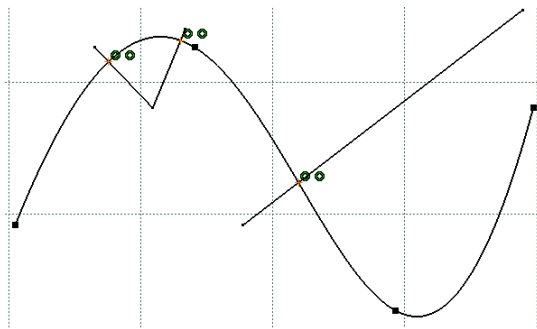


Figure 2-27 The intersection points created on the curve

Creating Projection Points

Menubar: Insert > Profile > Point > Projection Point
Toolbar: Profile > Point sub-toolbar > Projection Point



You can create one or more points by projecting points onto a curve element. To create projection points on a curve, select the points that are to be projected and choose the **Projection Point** tool from the **Point** sub-toolbar in the **Profile** toolbar; you will be prompted to select the element on which the selected points will be projected. Select the curve; the points will be projected automatically on it and construction lines representing the direction of the projection will also appear in the graphics area. The projection options available in the **Sketch tools** toolbar are discussed next.

Orthogonal Projection

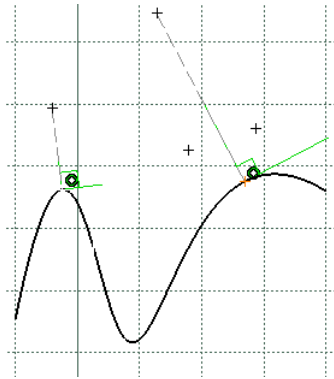


This option is selected by default. Select the points and then select the curve. All the selected points will be projected on the curve according to a normal direction on this curve, as shown in Figure 2-28.

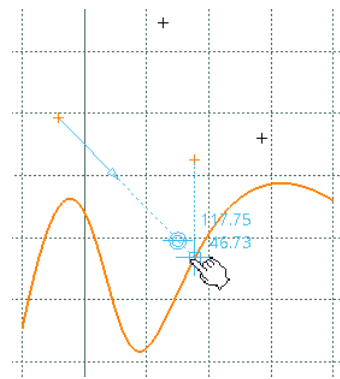
Along a Direction



Choose the **Along a Direction** option from the **Sketch tools** toolbar. Select a point and then define the position of the point on the curve; the selected points will get projected along the given direction, as shown in Figure 2-29.



*Figure 2-28 The projection points created using the **Orthogonal Projection** option*



*Figure 2-29 The projection points created using the **Along a Direction** option*

Aligning Points

Menubar: Insert > Profile > Point > Align Points
Toolbar: Profile > Point sub-toolbar > Align Points



You can use this tool to align points in a particular direction. Choose the **Align Points** tool from the **Point** sub-toolbar in the **Profile** toolbar and select all the points that you want to align; a small arrow indicating the direction of alignment will be displayed at the origin point. The direction of this arrow will change with respect to the pointer position in the graphics area, refer to Figure 2-30.

By default, the **Along a Direction** button is chosen in the **Sketch tools** toolbar. If you want to change the origin point, choose the **Change Origin Point** button in the **Sketch tools** toolbar and select a point to make it the origin point. If you choose the **Horizontal Alignment** button then the points will be aligned horizontally. Similarly, choose the **Vertical Alignment** button to align the points vertically. In the **Sketch tools** toolbar, choose the **Align along selected linear element** button and select the line; the points will be aligned along the direction of the selected line, as shown in Figure 2-31.

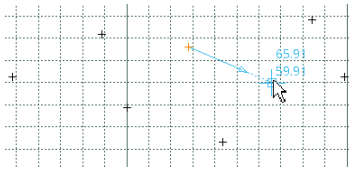


Figure 2-30 The point alignment direction

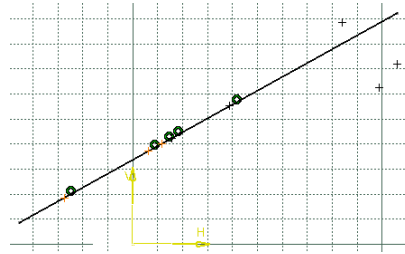


Figure 2-31 The points aligned along the line

Drawing Circles

You can draw a circle in CATIA by defining its center and radius, by specifying three points on its periphery, by using coordinates or so on. All the tools to draw a circle in CATIA are grouped in the **Circle** sub-toolbar. To view the **Circle** sub-toolbar, choose the down arrow on the right of the **Circle** tool in the **Profile** toolbar; the **Circle** sub-toolbar will be displayed, as shown in Figure 2-32. The tools available in this sub-toolbar will help you to draw various types of circles and arcs. Different methods to draw circles and arcs are discussed next.

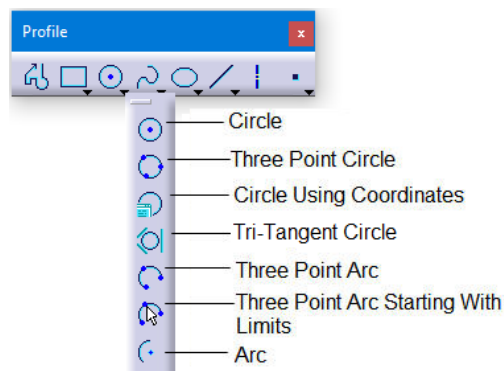


Figure 2-32 The **Circle** sub-toolbar

Drawing Circles using the Circle Tool

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



To draw a circle using this method, invoke the **Circle** tool from the **Circle** sub-toolbar; 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; 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

Menubar: Insert > Profile > Circle > Three Point Circle
Toolbar: Profile > Circle sub-toolbar > Three Point Circle



In CATIA V5, a circle can also be drawn by specifying any three points that will lie on its circumference. To draw a three point circle, invoke the **Three Point Circle** tool from the **Circle** sub-toolbar in 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, preview of the circle will be displayed. Click to specify the third point to create the circle.



Note
 You can also enter radius value in the **R** edit box of the expanded **Sketch tools** toolbar. Remember that when you enter 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

Menubar: Insert > Profile > Circle > Circle Using Coordinates
Toolbar: Profile > Circle sub-toolbar > Circle Using Coordinates



In CATIA V5, a circle can also be drawn by specifying the absolute coordinate values for the center and the radius. To do so, invoke the **Circle Using Coordinates** tool from the **Circle** sub-toolbar in the **Profile** toolbar; the **Circle Definition** dialog box will be displayed, as shown in Figure 2-33. You can specify the coordinate values of the center point and radius using the options in this dialog box.

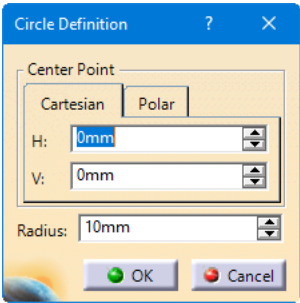


Figure 2-33 The Circle Definition dialog box

Drawing Tri-Tangent Circles

Menubar: Insert > Profile > Circle > Tri-Tangent Circle
Toolbar: Profile > Circle sub-toolbar > 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

with which a circle can form a tangent relation. Next, invoke the **Tri-Tangent Circle** tool from the **Circle** sub-toolbar in 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 the 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** sub-toolbar. In CATIA V5, there are three methods to draw arcs. These methods are discussed next.

Drawing Arcs by Defining the Center Point

Menubar: Insert > Profile > Circle > Arc
Toolbar: Profile > Circle sub-toolbar > Arc



To draw an arc by defining its center point, invoke the **Arc** tool from the **Circle** sub-toolbar in the **Profile** toolbar; you will be prompted to specify the center point. Click to specify the arc center; you will be 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 will be then prompted to specify the endpoint of the arc. As you move the cursor, preview of the arc is displayed, as shown in Figure 2-34. Click in the geometry area to specify the endpoint.

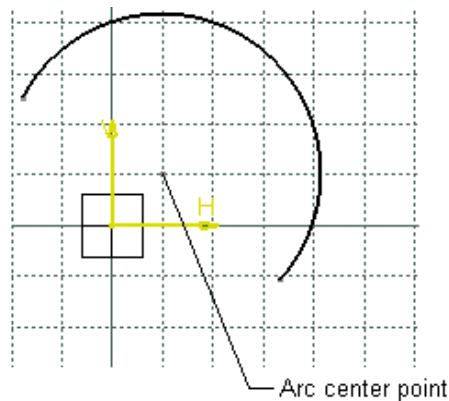


Figure 2-34 An arc drawn by defining its center point

Drawing Three Point Arcs

Menubar: Insert > Profile > Circle > Three Point Arc
Toolbar: Profile > Circle sub-toolbar > Three Point Arc



To draw a three point arc, choose the **Three Point Arc** tool from the **Circle** sub-toolbar in the **Profile** toolbar; you will be prompted to specify the start point of the arc. Click anywhere in the geometry area to specify the start point; you will be 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 will be displayed. Click to specify the second point; you will be prompted to specify the endpoint of the arc. As you move the cursor away to specify this point, the preview of the arc is displayed. Click in the geometry area to specify the endpoint. Figure 2-35 shows first, second, and third points being selected to draw a three point arc.

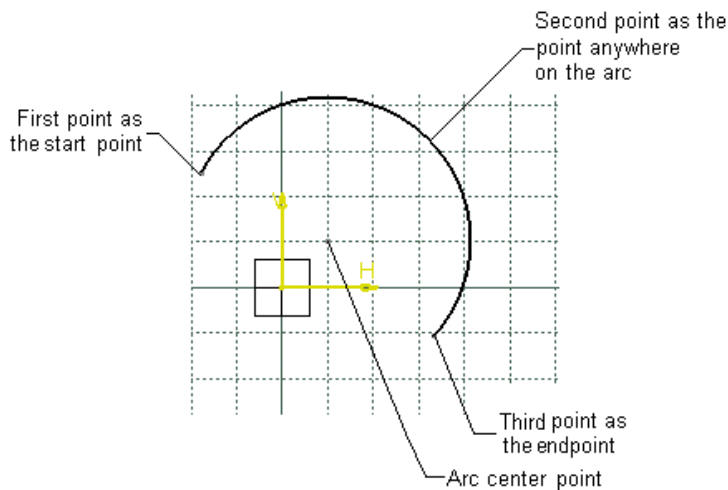


Figure 2-35 Selecting points to draw a three point arc

Drawing Three Point Arc Starting with Limits

Menubar: Insert > Profile > Circle > Three Point Arc Starting With Limits
Toolbar: Profile > Circle sub-toolbar > Three Point Arc Starting With Limits



While drawing a three point arc starting with limits, you can first specify the start point and the endpoint of the arc and then the third point anywhere on it. To draw this arc, invoke the **Three Point Arc Starting With Limits** tool from the **Circle** sub-toolbar in 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 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 2-36 shows the selection of first, second, and third points to draw an arc using this option.

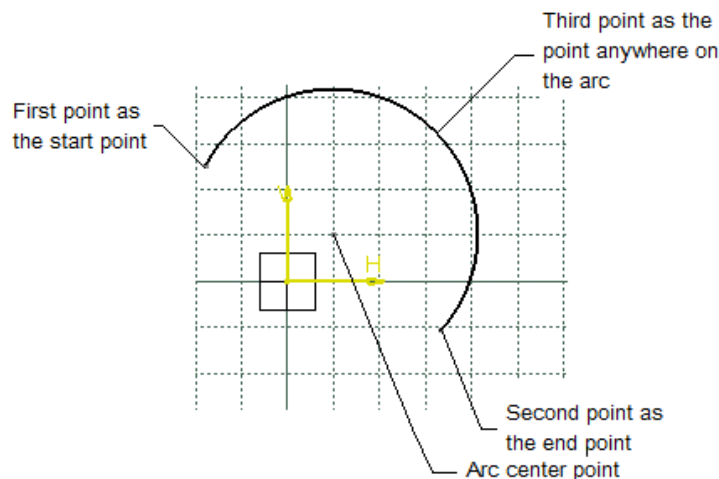


Figure 2-36 Selecting points to draw an arc using the *Three Point Arc Starting With Limits* tool

Drawing Profiles

Menubar: 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 get 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. The tool will be automatically terminated when you specify the point to close the profile. Figure 2-37 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 2-38. 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.

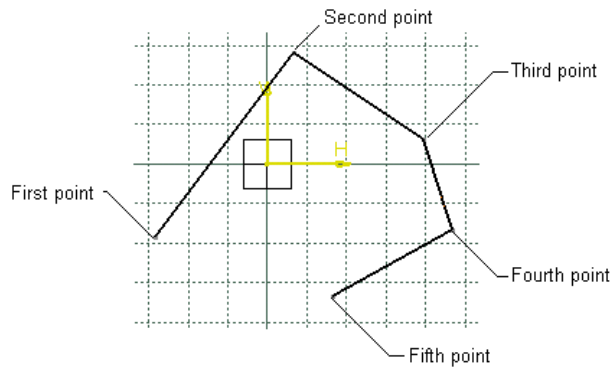


Figure 2-37 An open profile drawn using the **Profile** tool

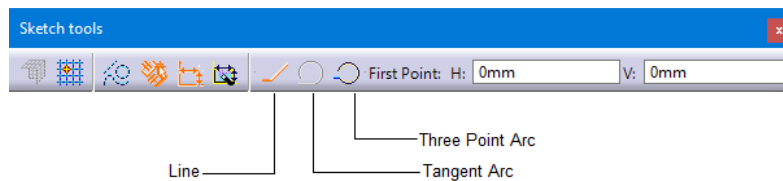


Figure 2-38 The **Sketch tools** toolbar displayed on choosing the **Profile** tool

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 get 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 and tangent to the line will be drawn and displayed in the geometry area. Figure 2-39 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.

Alternatively, you can invoke the **Profile** tool and specify the start and end points of the line segment. While specifying the end point, hold and drag the left mouse button; an orange color rectangle gets attached to the cursor. Release the left mouse button; a tangent arc to the line will get drawn.

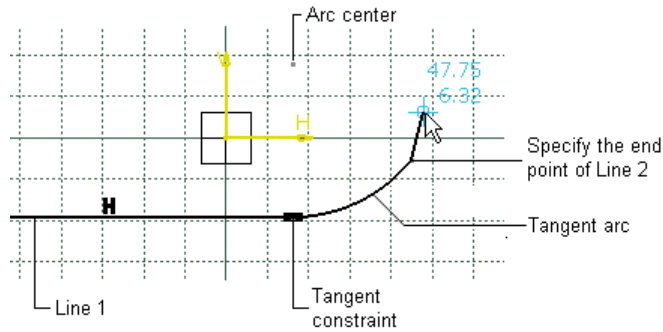


Figure 2-39 A tangent arc being drawn using the **Profile** tool

Drawing Three Point Arcs using the Profile Tool

To draw a three point arc using the **Profile** tool, invoke this tool from the **Profile** toolbar. You will notice the **Three Point Arc** button available in the **Sketch tools** toolbar. 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 as, shown in Figure 2-40. The basic tools such as **Fly Mode**, **Fit All In**, **Pan**, **Zoom In**, **Zoom Out**, **Zoom Area**, **Rotate**, **Normal View**, and **Hide/Show** are discussed next. You will learn about the remaining tools in the later chapters.

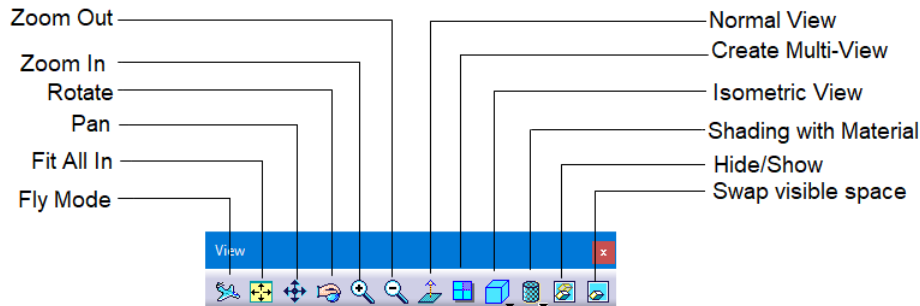


Figure 2-40 The **View** toolbar

Fly Mode

Toolbar: View > Fly Mode



The **Fly Mode** tool is used to view sketched elements in flying mode. On choosing this button, you will enter into the mode and notice that five new buttons are added to this toolbar namely, **Examine Mode**, **Turn Head**, **Fly**, **Accelerate**, and **Decelerate**.

On choosing the **Fly** button and by clicking the left mouse button in the graphics window, you will begin to fly forward in the chosen direction. Dragging the mouse in left, right, top, and down direction while holding the left button changes the direction accordingly. The **Accelerate** and **Decelerate** buttons are used to increase and decrease the flying speed and the **Examine Mode** button is used to exit the fly mode.



Note

When you switch to fly mode, the **Parallel** view changes into the **Perspective** view. To switch back to the **Parallel** view, choose **View > Render Style > Parallel**.

Fit All In

Menubar: View > Fit All In

Toolbar: View > Fit All In



The **Fit All In** tool is used to display all sketched elements or geometries 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 the later chapters.

Pan

Menubar: 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

Menubar: View > Modify > Zoom In

Toolbar: View > Zoom In



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

Zoom Out

Menubar: 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.



Tip

*You can also dynamically zoom in or zoom out an entity 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

Menubar: 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. Choose the **Zoom Area** option from the **View** menu and 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

Menubar: View > Modify > Normal View
Toolbar: View > Normal View



The **Normal View** tool is used to orient the view normal to the view direction. If, the current view is already normal to the view direction and you choose the **Normal View** button from the **View** toolbar; the viewing plane will 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

1. By default, whenever you invoke the **Sketcher** workbench without defining any particular orientation, the positive horizontal reference direction will point toward the right of the geometry area and also towards its upper area. 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.
2. While working in the **Sketcher** workbench, you can choose the **Normal View** button to orient the sketch normal to the sketching plane.

Create Multi View

Toolbar: View > Create Multi-View



The **Create Multi-View** tool is used to split the geometry area into four viewports. These viewports can be used to display different views of the model. To restore the single viewport configuration, choose this tool again.

Hide/Show Geometric Elements

Menubar: View > Hide/Show > Hide/Show

Toolbar: View > Hide/Show



The **Hide/Show** tool is used to hide sketcher elements or geometric elements from the current display. To do so, invoke this 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.

Swap 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.



Tip

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.

APPLYING DIMENSIONAL CONSTRAINTS

Menubar: Insert > Constraint > Constraint Creation > Constraint

Toolbar: Constraint > Constraint Creation sub-toolbar > Constraint



After applying the geometric constraints, you need to apply the dimensional constraints to fully define the sketches. It is recommended that you apply the dimensional constraints to the sketches using the **Constraint** tool. To invoke this tool, choose the down arrow on the right of the **Constraint** tool; the **Constraint Creation** sub-toolbar will be displayed, as shown in Figure 2-41. Now, choose the **Constraint** tool from this sub-toolbar.

When you choose this tool, you will be prompted to select an element to be constrained. Select the element that you need to dimension; a dimension will be attached to the cursor. Click anywhere in the geometry

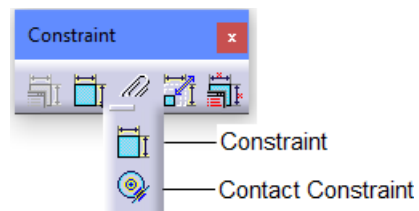


Figure 2-41 The **Constraint** toolbar with the **Constraint Creation** sub-toolbar

area to place the dimension. Using this tool, you can apply various types of dimensions such as linear dimensions, radius dimensions, diameter dimensions, and so on. These dimensions are discussed next.

Applying Linear Dimension to a Line or between Two Points

You can apply a linear dimension to a line or between two points using the **Constraint** tool. To do so, choose the **Constraint** tool and select a line; a linear dimension will be attached to the cursor. Move the cursor and click in the geometry area at a location where you need to place the dimension; a linear dimension will be placed in the geometry area.

To apply a linear dimension between two points, choose the **Constraint** tool, and then select two points from the geometry area. Next, right-click to display the contextual menu, as shown in Figure 2-42.

Choose the **Horizontal Measure Direction** option to place the linear dimension along the horizontal axis, or the **Vertical Measure Direction** option to place it along the vertical axis. On doing so, a linear dimension will be attached to the cursor. Click in the geometry area to place the dimension. Figure 2-43 shows the line to be selected for dimensioning and Figure 2-44 shows the resultant linear dimension. Figures 2-45 shows the points to be selected and Figure 2-46 shows the resultant linear dimensions.

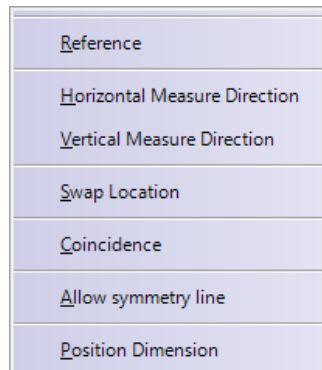


Figure 2-42 The contextual menu for the **Constraint** tool



Tip

If you choose the **Reference** option from the contextual menu, the selected dimension will be displayed as a reference dimension and will be displayed inside parenthesis. A reference dimension does not drive the geometry but it is the geometry which drives the reference dimension.

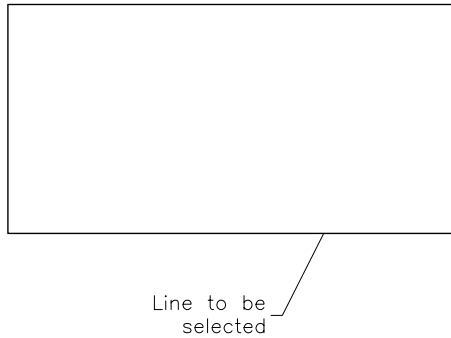


Figure 2-43 Line to be selected for dimensioning

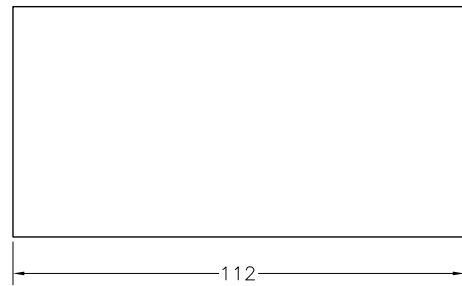


Figure 2-44 Resultant linear dimension

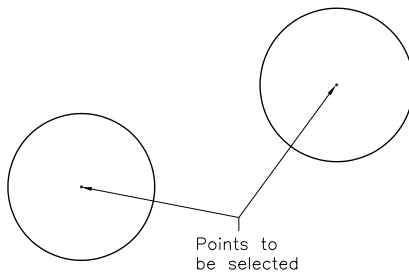


Figure 2-45 Points to be selected for dimensioning

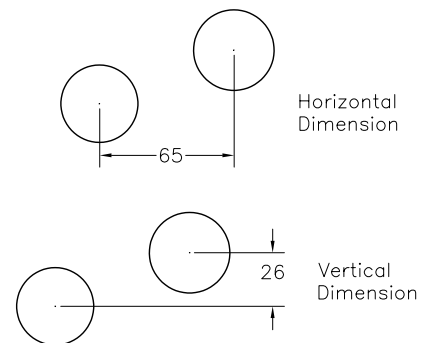


Figure 2-46 Resultant linear dimension

Dimensioning an Inclined Line

By default, whenever you select an inclined line, the aligned dimension is applied to it. You can also apply a horizontal or vertical dimension to it. To apply a horizontal dimension to an inclined line, choose the **Constraint** tool and select the line; an aligned dimension is attached to the cursor. Right-click to invoke the contextual menu. Choose the **Horizontal Measure Direction** or **Vertical Measure Direction** option depending on whether you want to place the horizontal or vertical dimension. To switch back to the aligned dimensioning, right-click in the geometry area again before placing the dimension, and choose the **No Measure Direction** option from the contextual menu.

Dimensioning an Arc or a Circle

To dimension an arc or a circle, choose the **Constraint** tool and select the arc or the circle that you need to dimension. By default, the diameter dimension is applied to circles and the radius dimension is applied to arcs. Move the cursor and click in the geometry area to place the dimension.



Note

1. While applying dimensions, you can convert a radius dimension into a diameter dimension. To do so, before placing the dimension, invoke the contextual menu by right-clicking. Choose the **Diameter** option from the contextual menu. Similarly, you can convert a diameter dimension into a radius dimension.

2. To convert a radius dimension already applied to an arc into a diameter dimension, double-click on the radius dimension; the **Constraint Definition** dialog box is displayed. Select the **Diameter** option from the **Dimension** drop-down list. You will notice that the radius dimension will be replaced by the diameter dimension. Choose the **OK** button from the **Constraint Definition** dialog box to apply the change. Similarly, you can change a diameter dimension into a radius dimension. You will learn more about applying dimensions using the **Constraint Definition** dialog box later in this chapter.

Applying Angular Dimensions

To apply an angular dimension, choose the **Constraint** tool and select the first line; a linear dimension is attached to the cursor. Select the second line; an angular dimension is attached to the cursor. Next, move the cursor and place the angular dimension. Remember that the type of angular dimension depends on its placement point. Figures 2-47 through 2-50 show the angular dimensions placed at different locations.

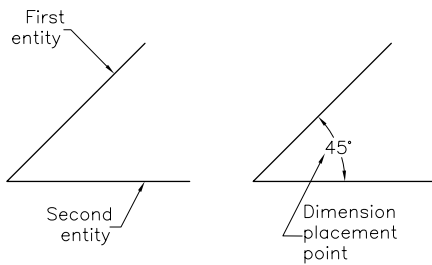


Figure 2-47 The angular dimension placed according to the placement point

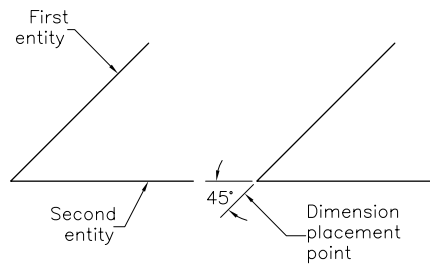


Figure 2-48 The angular dimension placed according to the placement point

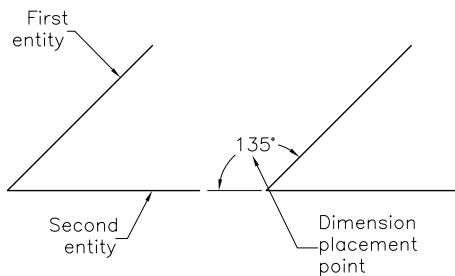


Figure 2-49 The angular dimension placed according to the placement point

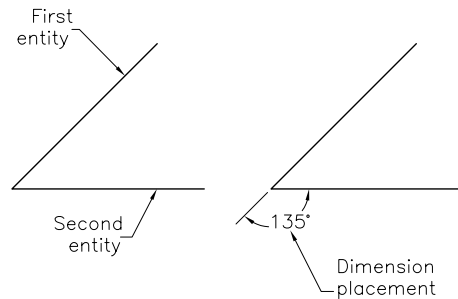


Figure 2-50 The angular dimension placed according to the placement point

Applying Linear Diameter Dimensions

Linear diameter dimensions are applied to the sketches of the features that need to be created by revolving the sketch using the **Shaft** or **Groove** tool. Figure 2-51 shows a component created using the revolved feature. Note that the sketch must have a center line drawn using the **Axis** tool, around which the sketch will be revolved. To apply a linear diameter dimension, choose the

Constraint tool and then select the entity that you need to dimension; a linear dimension will be attached to the cursor. Next, select the centerline that was drawn using the **Axis** tool; a linear dimension will be attached to the cursor. Right-click in the geometry area to invoke the contextual menu. Choose the **Radius / Diameter** option from the contextual menu; a linear diameter will be attached to the cursor. Now, move the cursor and place the dimension. Figure 2-52 shows the element and the centerline to be selected and the resultant linear diameter dimension.

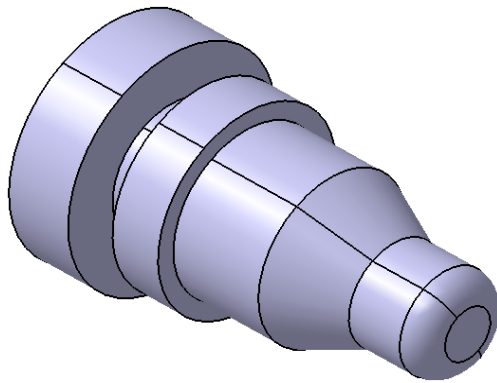


Figure 2-51 Model created by revolving a sketch around the horizontal centerline

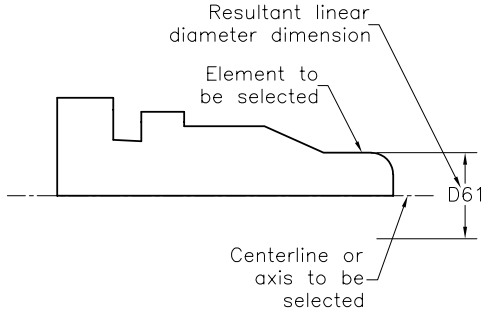


Figure 2-52 Element and centerline selected and the resultant linear diameter dimension

Modifying Dimensions after Placing Them

As discussed earlier, CATIA V5 is a parametric software. Therefore, you can change the dimensions and modify the design at any stage of the design cycle. In the previous section, you learned to place various types of dimensions. Note that the default value placed while dimensioning the entity may not be the required value. To modify the dimension value, double-click on it; the **Constraint Definition** dialog box will be displayed, refer to Figure 2-53. Set the value of the dimension in the **Value** spinner and choose the **OK** button from the **Constraint Definition** dialog box.

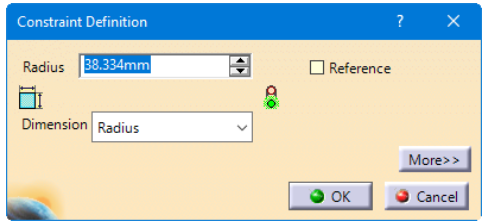


Figure 2-53 The Constraint Definition dialog box

**Tip**

If you choose the **More** button from the **Constraint Definition** dialog box, the dialog box will expand and the **Supporting Elements** area will be displayed. The **Type** column of this area displays the type of element on which the dimensional constraint is applied. The **Component** column displays the name of the element or the elements on which it is applied. The **Status** column displays the status of the constraint. You will learn more about the status of the constraints later in this chapter.

The name of the dimension is displayed in the **Name** edit box provided above the **Supporting Elements** area. You can modify the name of the dimension using this edit box.

CREATING BASE FEATURES BY EXTRUSION

Menubar: Insert > Sketch-Based Features > Pad
Toolbar: Sketch-Based Features > Pad sub-toolbar > Pad



The **Pad** tool is one of the most widely used tool to create the base features. To create a base feature using this tool, draw the sketch and exit the **Sketcher** workbench. Then, choose the down arrow on the right of the **Pad** tool in the **Sketch-Based Features** toolbar; the **Pad** sub-toolbar will be displayed, as shown in Figure 2-54. Select the sketch and then choose the **Pad** tool from the **Pad** sub-toolbar; the **Pad Definition** dialog box will be displayed, as shown in Figure 2-55 and you will be prompted to enter the required data to modify the pad. Also, the name of the selected sketch will be displayed in the **Selection** display box and the preview of the pad feature will be displayed in the geometry area.

Set the value of the depth of extrusion in the **Length** spinner. You can also define the extrusion depth dynamically in the geometry area. To do so, move the cursor close to **LIM1** in the default preview of the extrude feature. When the Limit drag handle (hand cursor) is displayed, refer to Figures 2-56 and 2-57, press and hold the left mouse button and drag the cursor; the depth of the extrusion will get dynamically defined. Then, choose the **OK** button from the **Pad Definition** dialog box. Figure 2-58 shows the model after creating the base feature by extruding the sketch.

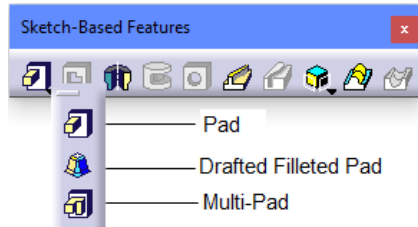


Figure 2-54 The *Sketch-Based Features* toolbar with the *Pad* sub-toolbar

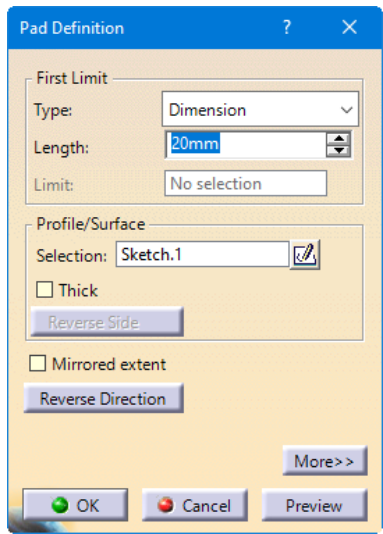


Figure 2-55 The *Pad Definition* dialog box

You can extrude the sketch feature about the sketch plane. To do so, choose the **More** button; the **Pad Definition** dialog box will expand. Enter the depth of extrusion for the second direction in the **Length** spinner of the **Second Limit** area; the changes will be displayed in the geometric area.

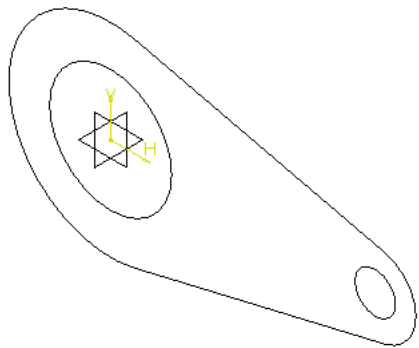


Figure 2-56 The sketch after exiting the *Sketcher* workbench

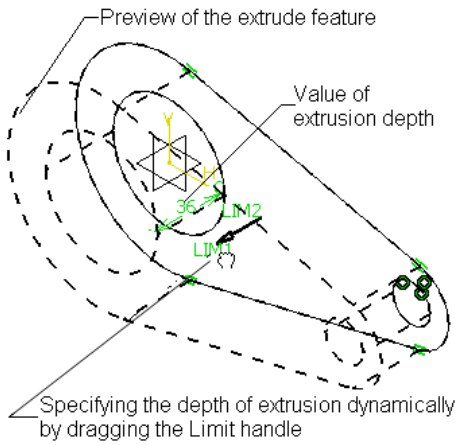


Figure 2-57 Dynamically dragging the *Limit* drag handle to specify the depth of extrusion

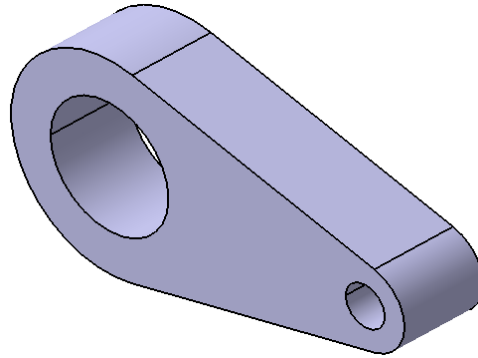


Figure 2-58 The model after extruding the sketch

GENERATING DRAWING VIEWS

After creating a solid model or an assembly, you need to generate its drawing views. Drawing views are the two-dimensional (2D) representations of a solid model or an assembly. CATIA V5 provides you with the **Drafting** workbench which is a specialized environment for generating 2D drawing views. This workbench provides all tools required to generate drawing views, modify, and apply dimensions and add annotations. In other words, you can get the final shop floor drawing using this workbench of CATIA V5. There are two types of drafting techniques in CATIA V5: Generative drafting and Interactive drafting. Generative drafting is a technique of generating the drawing views using a solid model or an assembly model. Interactive drafting is a technique, in which the sketcher tools are used to draw the 2D drawing views.

Tutorial 1

Draw the sketch shown in Figure 2-59. Extrude the sketch by 60 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-60. You do not need to dimension the drawing.

(Expected time: 30 min)

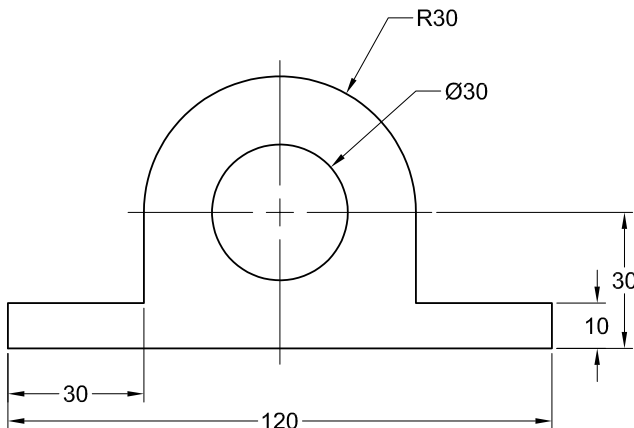


Figure 2-59 The sketch for the solid model

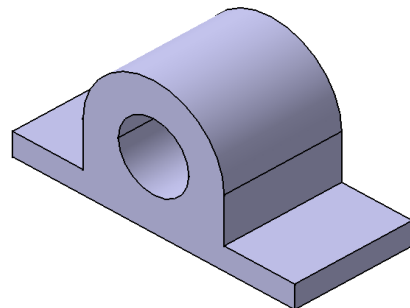


Figure 2-60 The solid model for Tutorial 1

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.
- Extrude the sketch up to a distance of 60 mm using the **Extrude** tool.
- Save the model and then generate its drawing views.

Starting a New Part File

- Start CATIA V5 by double-clicking on the shortcut icon of **CATIA V5-6R2025** on the desktop of your computer; a new file, **Product1** is started.
- Next, choose **Start > Mechanical Design > Part Design** from the menu bar; the **New Part** dialog box is displayed, as shown in Figure 2-61.
- Enter **C02_Tut1** as the name of the file in the **Enter part name** edit box.
- Select the **Enable hybrid design** check box from the **New Part** dialog box if not selected.
- Choose the **OK** button; a new file in the **Part Design** workbench is started.
- 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 2-62.

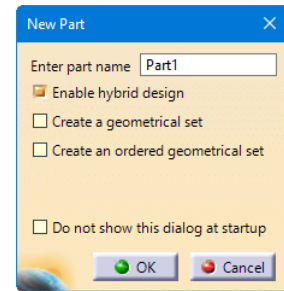


Figure 2-61 The *New Part* dialog box

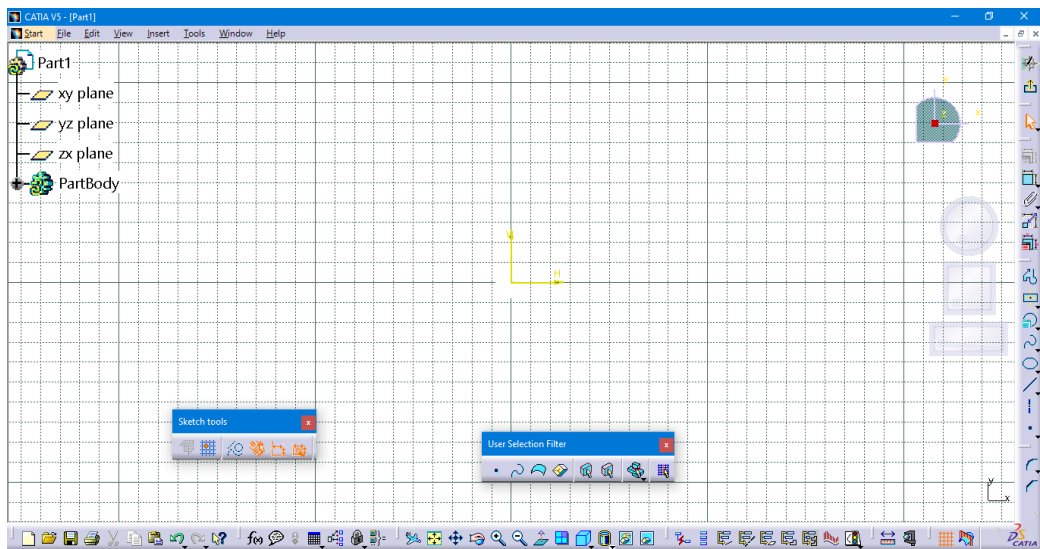


Figure 2-62 The *Sketcher* workbench screen

In this tutorial, you need to draw the sketch in two parts: first draw the outer loop and second draw the inner circle in the Sketcher workbench.

Drawing the Outer Loop of the Sketch

In this section, 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. 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 not chosen.
3. Move the cursor to the location whose coordinates are (0,0) (at the origin) and click to 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 right, the line turns blue. Click to specify the endpoint of the line where the coordinates are 120,0.



Note

The constraints that get automatically applied to the sketch drawn will be explained in the later chapters.

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

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

8. Choose the **Tangent Arc** tool from the **Sketch tools** toolbar to switch to the **Tangent Arc** mode.



Note

*If the **Sketch tools** toolbar is not displayed in the graphics area then right-click on any of the available toolbars and select the **Sketch tools** from the shortcut menu displayed, the **Sketch tools** toolbar will become available in the graphics area.*

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 a 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 click to specify the endpoint of the line when the value of the coordinates is 30,10.
11. Move the cursor horizontally toward the left and click to specify the endpoint of the line when the value of the coordinates is 0,10.
12. Move the cursor vertically downward and click to 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 2-63. In this figure, the constraints are hidden for better visualization.

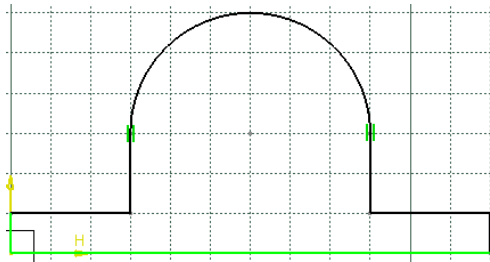


Figure 2-63 The sketch after drawing the outer loop

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 click to 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 2-64. Note that in this figure, the display of constraints is turned on.

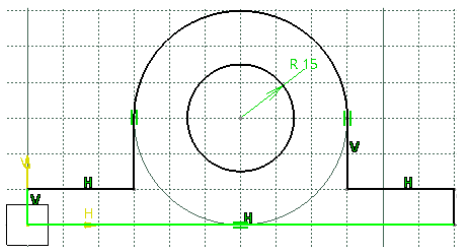


Figure 2-64 The final sketch for Tutorial 1

Extruding the Sketch

Next, you need to extrude the sketch.

1. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed and a preview of the extruded feature is displayed in the geometry area.
2. Set **60** as the value in the **Length** spinner of the **Pad Definition** dialog box, and then choose **OK** from this dialog box.
3. Click once in the geometry area to remove the newly created base feature from the current selection set.
4. Orient the view of the model using the **Isometric View** tool from the **View** toolbar. The result is shown in Figure 2-65.

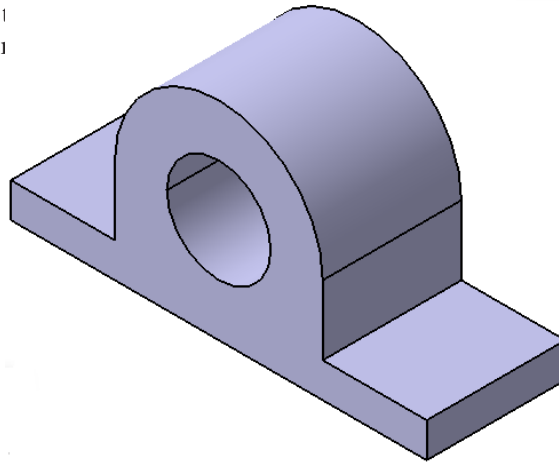



Figure 2-65 The final model for Tutorial 1

Saving the Model


After completing the model, 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 a folder named **CATIA** inside the **C:** drive. Then create the folder **c02** inside the **CATIA** folder.
2. Next, choose the **Save** button; the file is saved at **C:\CATIA\c02**.

Generating Drawing Views of Model

1. Choose **Start > Mechanical Design > Drafting** from the menubar to display the **New Drawing Creation** dialog box.
2. Choose the **OK** button from the **New Drawing Creation** dialog box: a new file is started in the **Drafting** workbench.
3. Choose the **Front View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
4. Choose **Window > c02tut1.CATPart** from the menu bar; the part file is displayed.
5. Select the **yz** plane (of the front planar face) from the Specification tree; the drawing file is invoked.

A preview of the front view is displayed and a knob appears on the view to set the orientation of the front view.

6. Move the cursor on the frame that shows the preview of the front view; the cursor is replaced by a hand cursor. Press and hold the left mouse button and drag the cursor close to the lower left corner of the drawing sheet.
7. Click anywhere on the drawing sheet to generate the front view.
8. Next double click on the **Projection View** tool from the **Projections** sub-toolbar in the **Views** toolbar then place the cursor on the front view; a preview of the projected view is attached to the cursor.
9. Similarly, taking the front view as the parent view, generate the drawing of right-side view.
10. Choose the **Isometric View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
11. Choose **Window > c02tut1.CATPart** from the menu bar; the part file is displayed.
12. Select the front face of the model from the geometry area; the drawing sheet is displayed. Also, a preview of the isometric view of the model is displayed on the drawing sheet along with a knob which can be used to orient the isometric view.
13. Click anywhere on the drawing sheet to generate the isometric view. The drawing sheet after generating the isometric view is shown in Figure 2-66.

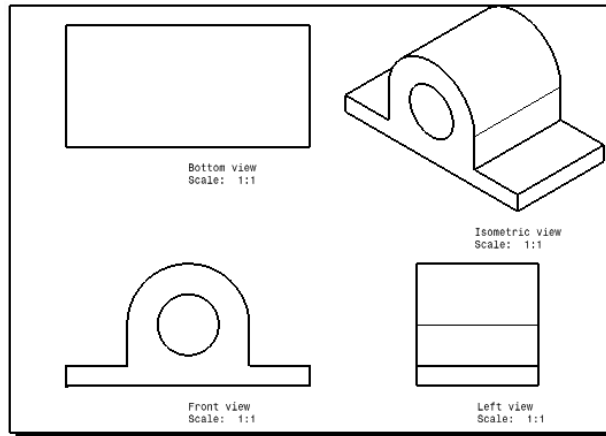


Figure 2-66 Drawing sheet after generating the drawing views

Tutorial 2

Draw the sketch shown in Figure 2-67. Extrude the sketch by 20 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-68. You do not need to dimension the drawing.

(Expected time: 30 min)

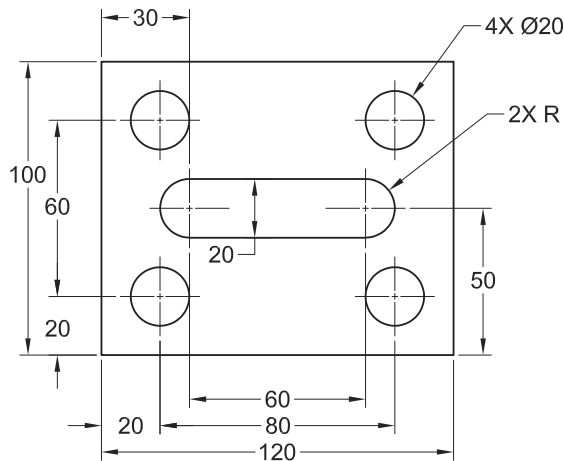


Figure 2-67 The sketch for the solid model

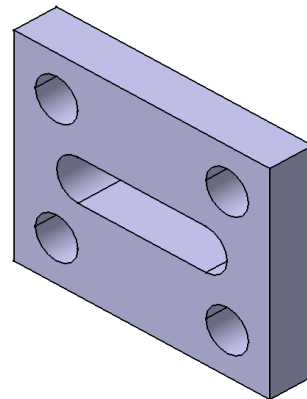


Figure 2-68 The solid model for Tutorial 2

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.
- c. Extrude the sketch upto a distance of 20 mm using the **Extrude** tool.
- d. Save the model and then generate its drawing views.

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, as shown in Figure 2-69. Alternatively, choose the **New** tool from the **Standard** toolbar to invoke the **New** dialog box.
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.
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; the **Sketcher** workbench is invoked.

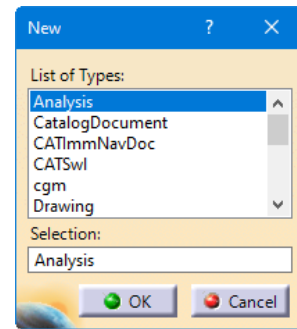


Figure 2-69 The New dialog box

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 circular 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 click to specify the lower left corner of the rectangle.
3. Move the cursor to the location whose coordinates are 60,50 and click to specify the upper right corner of the rectangle. Figure 2-70 shows the outer loop of the sketch drawn using the **Rectangle** tool.



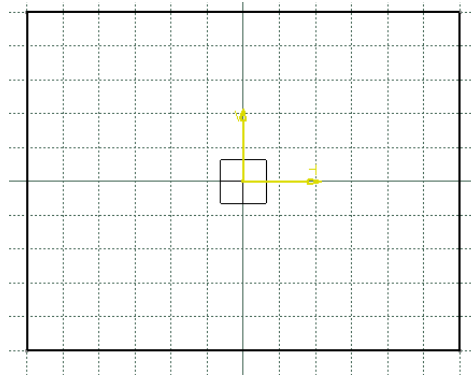


Figure 2-70 The outer loop of the sketch

Drawing the Inner Loop of the Sketch

After drawing 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 click to specify this point as the start point of the line.
3. Move the cursor horizontally toward the right and click to specify the endpoint of the line where the coordinates are 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** tool 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 click to 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 click to specify the endpoint of the line.
7. Choose the **Tangent Arc** tool 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 and then click to specify the endpoint of the arc when it snaps to the start point.

The sketch of the elongated hole is shown in Figure 2-71. In this figure, the constraints are hidden for better visualization.

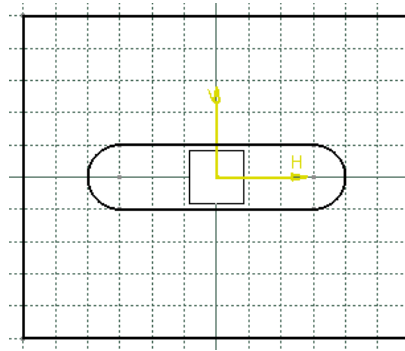


Figure 2-71 The sketch of the elongated hole

9. Choose the **Circle** tool from the **Profile** toolbar.
10. Move the cursor to the location whose coordinates are 40,30 and click to specify the center point of the circle.
11. Enter **10** as the radius value for 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.
12. Choose the **Circle** tool from the **Profile** toolbar.
13. Move the cursor to the location whose coordinates are 40,-30 and click to specify the center point of the circle.
14. Enter **10** as the radius value for the circle in the **R** edit box of the **Sketch tools** toolbar and press Enter.
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 2-72.

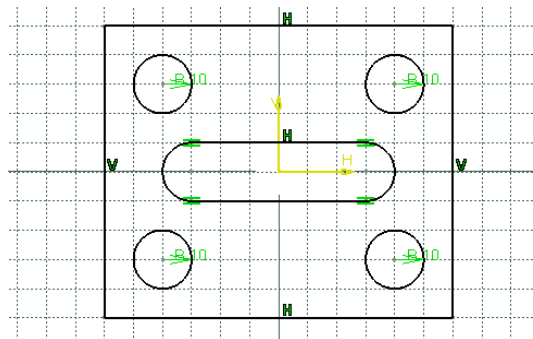


Figure 2-72 The final sketch

Extruding the Sketch

Next, you need to extrude the sketch.

1. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed and a preview of the extruded feature is displayed in the geometry area.
2. Set **20** as the value in the **Length** spinner of the **Pad Definition** dialog box, and then choose **OK** button from this dialog box.
3. Click once in the geometry area to remove the newly created base feature from the current selection set.
4. Orient the view of the model to the isometric view using the **Isometric View** tool from the **View** toolbar. The model after creating the extruded feature is shown in Figure 2-73.

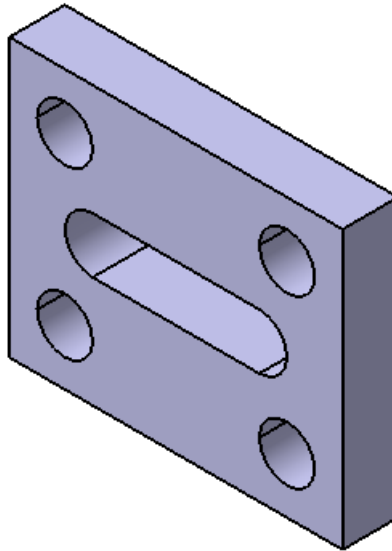


Figure 2-73 The final model for Tutorial 2


Saving the Model

After completing the model, 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 a folder named **CATIA** inside the **C:** drive. Then create the folder **c02** inside the CATIA folder.

2. Next, choose the **Save** button; the file is saved at *C:\CATIA\c02*.

Generating Drawing Views of Model

1. Choose **Start > Mechanical Design > Drafting** from the menubar to display the **New Drawing Creation** dialog box.
2. Choose the **OK** button from the **New Drawing Creation** dialog box; a new file is started in the **Drafting** workbench.
3. Choose the **Front View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
4. Choose **Window > c02tut2.CATPart** from the menu bar; the part file is displayed.
5. Select the **yz** plane (of the front planar face) from the Specification tree; the drawing file is invoked.

A preview of the front view is displayed and a knob appears on the view to set the orientation of the front view.

6. Move the cursor on the frame that shows the preview of the front view; the cursor is replaced by a hand cursor. Press and hold the left mouse button and drag the cursor close to the lower left corner of the drawing sheet.
7. Click anywhere in the drawing sheet to generate the front view.
8. Next, double-click on the **Projection View** tool from the **Projections** sub-toolbar in the **Views** toolbar then place the cursor on the front view; a preview of the projected view is attached to the cursor.
9. Similarly, taking the front view as the parent view, generate the drawing of right-side view.
10. Choose the **Isometric View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
11. Choose **Window > c02tut2.CATPart** from the menu bar; the part file is displayed.
12. Select the front face of the model from the geometry area; the drawing sheet is displayed. Also, a preview of the isometric view of the model is displayed on the drawing sheet along with a knob which can be used to orient the isometric view.
13. Click anywhere on the drawing sheet to generate the isometric view. The drawing sheet after generating the isometric view is shown in Figure 2-74.

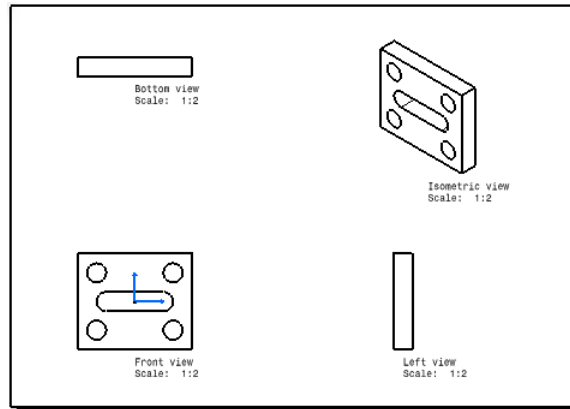


Figure 2-74 Drawing sheet after generating the drawing views

Tutorial 3

Draw the sketch shown in Figure 2-75. Extrude the sketch by 40 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-76. You do not need to dimension the drawing.

(Expected time: 20 min)

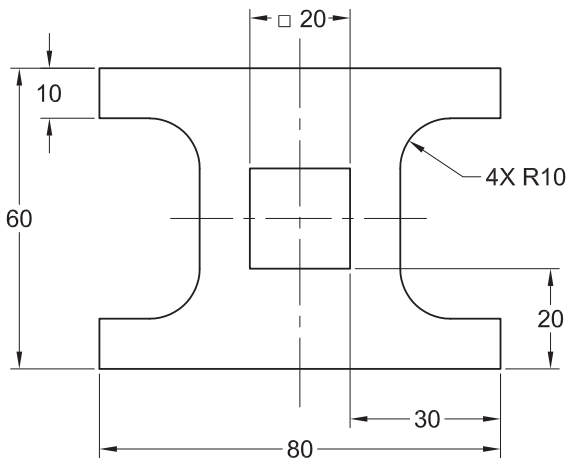


Figure 2-75 The sketch of the model

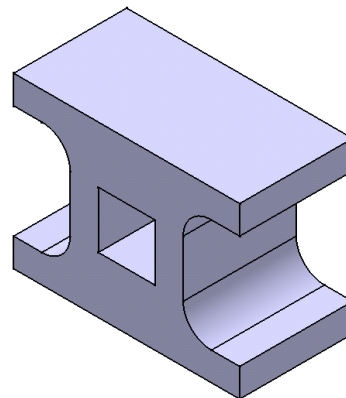


Figure 2-76 The solid model for Tutorial 3

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.
- Extrude the sketch up to a distance of 40 mm using the **Extrude** tool.
- Save the model and then generate its drawing views.

Starting a New Part File

1. Choose **File > New** from the menu bar; the **New** dialog box is displayed.
2. In the **New** dialog box, select **Part** from the **List of Types** list box and choose the **OK** button; the **New Part** dialog box is displayed.
3. Enter **c02tut3** as the name of the file in the **Enter part name** edit box. Retaining rest of the default settings in the **New Part** dialog box, 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 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.

1. Choose the **Profile** tool from the **Profile** toolbar.
2. Click to specify the start point of the line at the location whose coordinates are -40,-30 and then move the cursor horizontally toward the right; you will notice that the line turns blue.
3. Move the cursor to the location whose coordinates are 40,-30. Click to specify the endpoint of the line; a rubber band line is attached to the cursor.
4. Move the cursor vertically upward and click to specify the endpoint of the second line whose coordinates are 40,-20.
5. Move the cursor horizontally toward the left where the coordinates are 30,-20 and click to specify the endpoint of the third line.

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

6. Choose the **Tangent Arc** tool from the **Sketch tools** toolbar.

7. Move the cursor to the location whose coordinates are 20, -10 and click to specify the endpoint of the tangent arc; the **Line** mode is activated and the line gets attached to the cursor again. Figure 2-77 shows the sketch after drawing three lines and a tangent arc. In this figure, the constraints are hidden for better visualization.

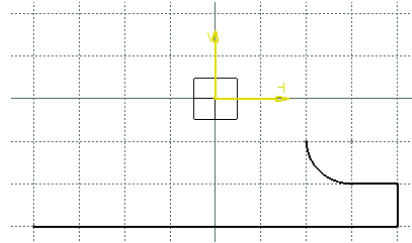


Figure 2-77 The sketch after drawing three lines and a tangent arc

8. Move the cursor vertically upward to the location whose coordinates are 20,10 and then click to 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.

9. Choose the **Tangent Arc** tool from the **Sketch tools** toolbar; the **Tangent Arc** mode is activated.
10. Move the cursor to the location whose coordinates are 30, 20 and click to 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.
11. Move the cursor horizontally toward the right and click to specify the endpoint of the line when the coordinates are 40,20.
12. Move the cursor vertically upward and click to specify the endpoint of the line when the coordinates are 40,30.
13. Move the cursor horizontally toward the left and click to specify the endpoint of the line when the coordinates are -40,30.
14. Move the cursor vertically downward and click to specify the endpoint of the line when the coordinates are -40,20.
15. Move the cursor horizontally toward the right and click to 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** tool from the **Sketch tools** toolbar.

16. Choose the **Tangent Arc** button from the **Sketch tools** toolbar; the **Tangent Arc** mode is activated.
17. Move the cursor to the location whose coordinates are -20,10 and click to specify the endpoints of the arc at this location; the **Line** mode is activated and line is attached to the cursor.
18. Move the cursor vertically downward and specify the endpoint of the line when the coordinates are -20,-10.

19. Switch to the **Tangent Arc** mode by choosing the **Tangent Arc** tool from the **Sketch tools** toolbar and then move the cursor to the location whose coordinates are -30,-20 and click to specify the endpoint of the tangent arc at this location.
20. Move the cursor horizontally toward the left and click to specify the endpoint of the line where the coordinates are -40,-20.
21. Move the cursor vertically downward and click to 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 2-78.

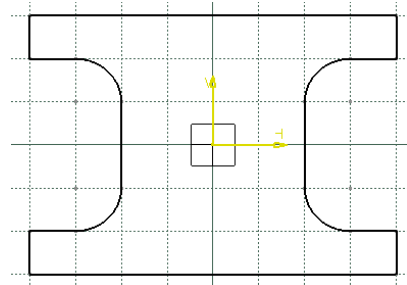


Figure 2-78 The sketch after drawing the outer loop and hiding the constraints

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 click to specify the upper-left corner of the rectangle at this location.
3. Move the cursor to the location whose coordinates are 10,-10 and click to 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 2-79. Note that in this figure, the display of constraints has been turned on.

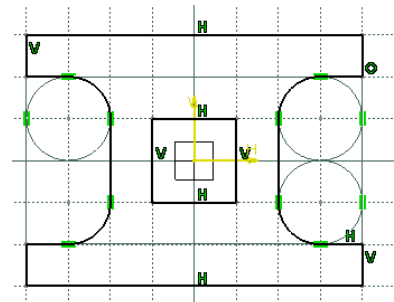


Figure 2-79 The final sketch after drawing the inner loop

Extruding the Sketch

Next, you need to extrude the sketch.

1. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed and a preview of the extruded feature is displayed in the geometry area.
2. Set **40** as the value in the **Length** spinner of the **Pad Definition** dialog box, and then choose **OK** button from this dialog box.
3. Click once in the geometry area to remove the newly created base feature from the current selection set.
4. Orient the view of the model to the isometric view using the **Isometric View** tool from the **View** toolbar. The model after creating the extruded feature is shown in Figure 2-80.

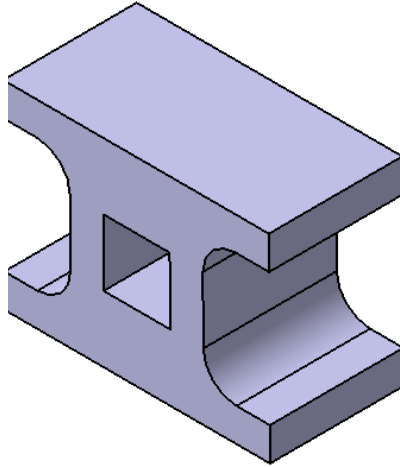



Figure 2-80 The final model for Tutorial 3

Saving the Model

After completing the model, 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 a folder named **CATIA** inside the **C:** drive. Then create the folder **c02** inside the **CATIA** folder.
2. Next, choose the **Save** button; the file is saved at **C:\CATIA\c02**.

Generating Drawing Views of Model

1. Choose **Start > Mechanical Design > Drafting** from the menubar to display the **New Drawing Creation** dialog box.
2. Choose the **OK** button from the **New Drawing Creation** dialog box: a new file is started in the **Drafting** workbench.
3. Choose the **Front View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
4. Choose **Window > c02tut3.CATPart** from the menu bar; the part file is displayed.
5. Select the **yz** plane (of the front planar face) from the Specification tree; the drawing file is invoked.

A preview of the front view is displayed and a knob appears on the view to set the orientation of the front view.

6. Move the cursor on the frame that shows the preview of the front view; the cursor is replaced by a hand cursor. Press and hold the left mouse button and drag the cursor close to the lower left corner of the drawing sheet.

7. Click anywhere on the drawing sheet to generate the front view.
8. Next double-click on the **Projection View** tool from the **Projections** sub-toolbar in the **Views** toolbar then place the cursor on the front view; a preview of the projected view is attached to the cursor.
9. Similarly, taking the front view as the parent view, generate the drawing of right-side view.
10. Choose the **Isometric View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
11. Choose **Window > c02tut3.CATPart** from the menu bar; the part file is displayed.
12. Select the front face of the model from the geometry area; the drawing sheet is displayed. Also, a preview of the isometric view of the model is displayed on the drawing sheet along with a knob which can be used to orient the isometric view.
13. Click anywhere on the drawing sheet to generate the isometric view. The drawing sheet after generating the isometric view is shown in Figure 2-81.

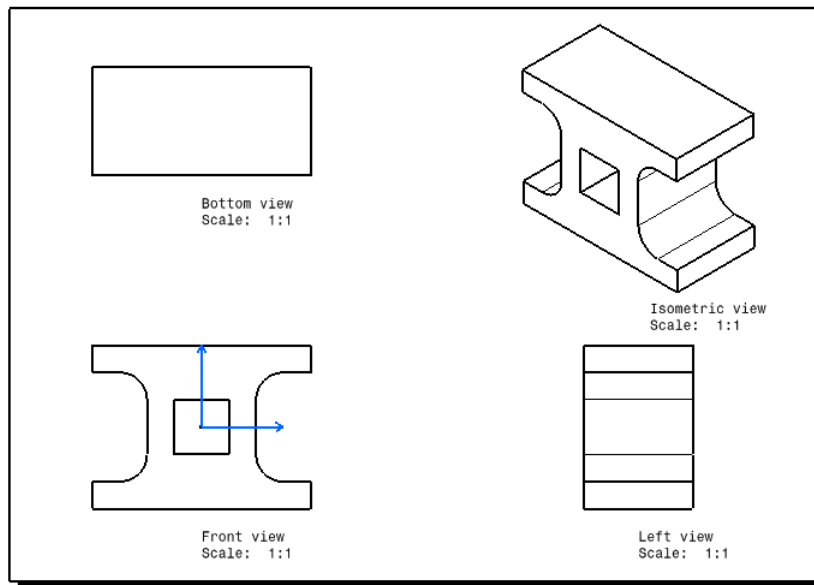


Figure 2-81 Drawing sheet after generating the drawing views

Tutorial 4

Draw the sketch shown in Figure 2-82. Extrude the sketch by 60 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-83. You do not need to dimension the drawing.

(Expected time: 30 min)

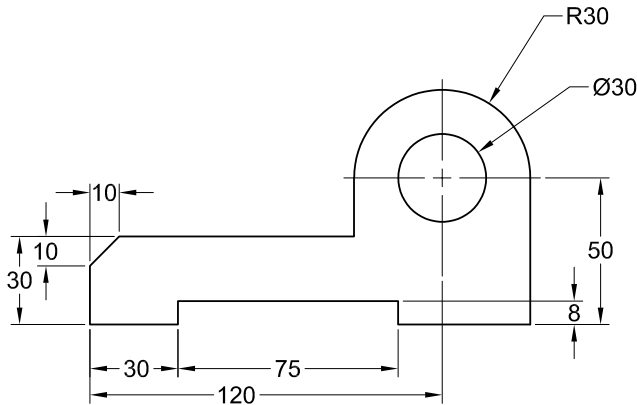


Figure 2-82 The sketch of the model

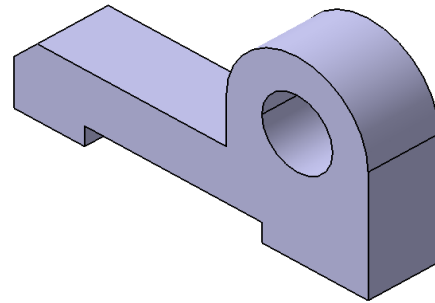


Figure 2-83 The solid model for Tutorial 4

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 **Line**, **Arc**, and **Circle** tools.
- Extrude the sketch upto a distance of 60 mm using the **Extrude** tool.
- Save the model and then generate its drawing views.

Starting CATIA V5 and then a New Part File

- Choose **File > New** from the menu bar; the **New** dialog box is displayed.
- 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.
- 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 setting 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 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.

- Invoke the **Line** tool by choosing the **Line** tool from the **Profile** toolbar.
- Choose the **Snap to Point** button from the **Sketch tools** toolbar if not chosen.



3. Move the cursor in the third quadrant; the coordinates of the point are displayed above the cursor.
4. Click to specify the point whose coordinates are -50,-30. Next, move the cursor horizontally toward the right.

It is evident from Figure 2-82 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.

5. Click to draw the line when the **L** edit box 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 twice 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 keeps 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. Choose the **Three Point Arc** tool from the **Circle** sub-toolbar.



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 70,50. These coordinates are displayed in the **Sketch tools** toolbar and also on the top of the cursor. Now, click in the geometry area to specify the second point.
16. Move the cursor to a location 40,20 in the geometry area to specify the third point of the arc and then click; the coordinate values are displayed on the 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.

17. Now, to continue drawing the outer loop, you need to invoke the **Line** tool again. 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 and click to draw the line when the length in the **L** edit box shows a value of 20mm in the **Sketch tools** toolbar.

The fourth vertical line of length 20mm is drawn. You will notice that the line is no longer in the selection mode and you are prompted to specify 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 to draw the line when the length in the **L** edit box shows a value of 80mm in the **Sketch tools** toolbar.

The fourth horizontal line of length 80mm is drawn. 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° . The current angle is 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; a horizontal inclined line of 10 unit is drawn.

21. Select the endpoint of the inclined line as the start point of the next line. Move the cursor vertically downward and click when the **L** edit box displays a value of 20mm. Next, press the Esc key to exit the active tool.

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.

22. Choose the **Fit All In** button from the **View** toolbar to fit the current sketch into the screen. The completed outer loop of the sketch is shown in Figure 2-84. Note that in this figure, the display of constraints and dimensions is turned off using the **Hide/Show** tool for clarity.

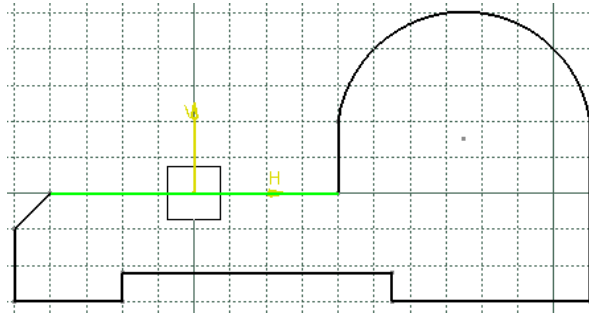


Figure 2-84 The completed outer loop of the sketch

Drawing the Inner Circle

Now, you need to draw a circle using the **Circle** tool.

1. Choose the **Circle** tool from the **Circle** sub-toolbar to invoke it; you are prompted to define the center point of the circle.
2. Move the cursor to the point whose coordinates are 70,20. Click when the cursor snaps to this point.
3. Move the cursor horizontally toward the right and click when the **R** edit box of the **Sketch tools** toolbar displays a value of 15mm. Click anywhere in the geometry area to remove the circle from the selection.



The final sketch after drawing the circle is shown in Figure 2-85. Note that in this figure, the display of the constraints has been turned on.

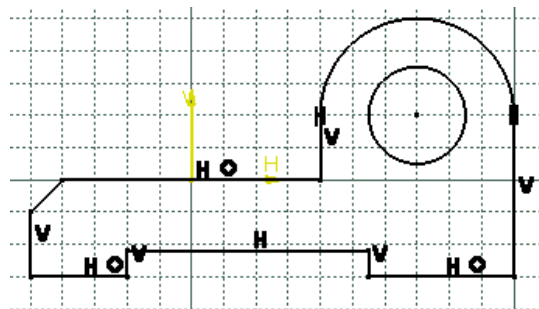


Figure 2-85 The final sketch for Tutorial 4

Extruding the Sketch

Next, you need to extrude the sketch.

1. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed and a preview of the extruded feature is displayed in the geometry area.
2. Set **60** as the value in the **Length** spinner of the **Pad Definition** dialog box, and then choose **OK** from this dialog box.

3. Click once in the geometry area to remove the newly created base feature from the current selection set.
4. Orient the view of the model to the isometric view using the **Isometric View** tool from the **View** toolbar. The model after creating the extruded feature is shown in Figure 2-86.

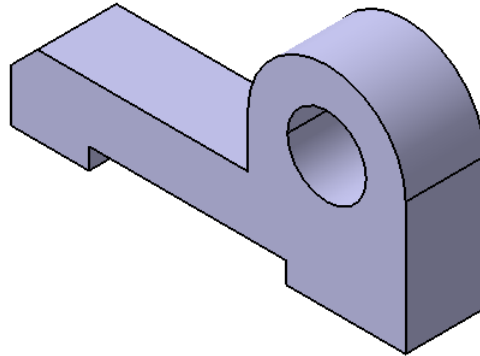



Figure 2-86 The final model for Tutorial 4

Saving the Model


After completing the model, 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 a folder named **CATIA** inside the **C:** drive. Then create the folder **c02** inside the **CATIA** folder.
2. Next, choose the **Save** button; the file is saved at **C:\CATIA\c02**.

Generating Drawing Views of Model

1. Choose **Start > Mechanical Design > Drafting** from the menubar to display the **New Drawing Creation** dialog box.
2. Choose the **OK** button from the **New Drawing Creation** dialog box: a new file is started in the **Drafting** workbench.
3. Choose the **Front View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
4. Choose **Window > c02tut4.CATPart** from the menu bar; the part file is displayed.
5. Select the **yz** plane (of the front planar face) from the Specification tree; the drawing file is invoked.

A preview of the front view is displayed and a knob appears on the view to set the orientation of the front view.

6. Move the cursor on the frame that shows the preview of the front view; the cursor is replaced by a hand cursor. Press and hold the left mouse button and drag the cursor close to the lower left corner of the drawing sheet.
7. Click anywhere in the drawing sheet to generate the front view.
8. Next, double-click on the **Projection View** tool from the **Projections** sub-toolbar in the **Views** toolbar then place the cursor on the front view; a preview of the projected view is attached to the cursor.
9. Similarly, taking the front view as the parent view, generate the drawing of right-side view.
10. Choose the **Isometric View** tool from the **Projections** sub-toolbar in the **Views** toolbar; you are prompted to select a reference plane on a 3D geometry. 
11. Choose **Window > c02tut4.CATPart** from the menu bar; the part file is displayed.
12. Select the front face of the model from the geometry area; the drawing sheet is displayed. Also, a preview of the isometric view of the model is displayed on the drawing sheet along with a knob which can be used to orient the isometric view.
13. Click anywhere on the drawing sheet to generate the isometric view. The drawing sheet after generating the isometric view is shown in Figure 2-87.

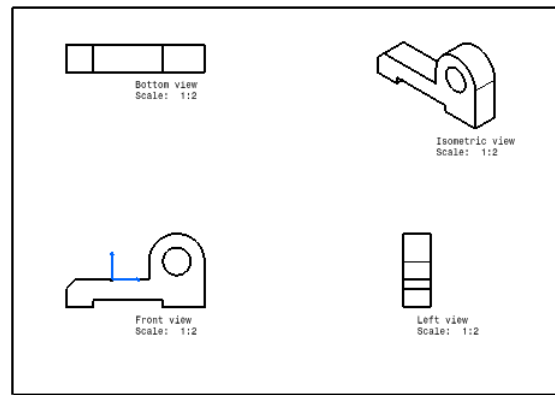


Figure 2-87 Drawing sheet after generating the drawing views

Self-Evaluation Test

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

1. You can convert a sketched element into a construction element by using the _____ tool.
2. To draw a rectangle at an angle, you need to use the _____ tool.

3. The rectangle is considered as a combination of individual _____.
4. The _____ tool is used to draw continuous lines.
5. Using the _____ tool, you can create a circle by specifying the coordinates of its center point.
6. _____ are temporary lines that are used to track a particular point on the screen.
7. The base feature of any design is a sketched feature which is created by drawing the sketch. (T/F)
8. You can draw an arc while working with the **Profile** tool. (T/F)
9. To enter the **Sketcher** workbench, you need to choose the **Sketch** tool. (T/F)
10. When you open a file that has been saved in the sketching environment, it opens in the part modeling environment. (T/F)

Review Questions

Answer the following questions:

1. 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
2. Which of the following tools is not available in the **Predefined Profile** sub-toolbar?
 - (a) **Rectangle**
 - (b) **Oriented Rectangle**
 - (c) **Parallelogram**
 - (d) **Circle**
3. 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
4. Which one of the following tools is available in the **Line** toolbar?
 - (a) **Line**
 - (b) **Infinite Line**
 - (c) **Bisecting Line**
 - (d) All of these

5. 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
6. The 3 point arc is the arc that is drawn by defining a start point, an endpoint, and a point on the arc. (T/F)
7. The **Parallelogram** tool is available in the **Predefined Profile** sub-toolbar. (T/F)
8. The **Symmetrical Extension** button when chosen from the **Sketch tools** toolbar draws a simple line. (T/F)
9. 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)
10. When you start CATIA V5, a file in the **Product** workbench is started by default. (T/F)

EXERCISES

Exercise 1

Draw the sketch shown in Figure 2-88. Extrude the sketch by 20 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-89. You do not need to dimension the drawing.

(Expected time: 20 min)

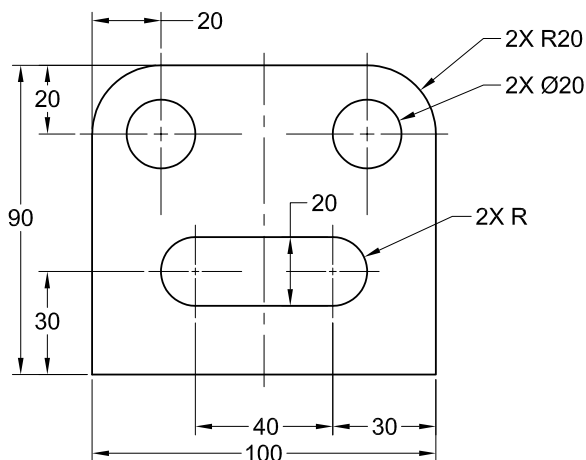


Figure 2-88 The sketch of the model

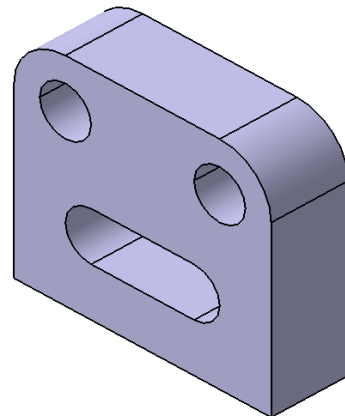


Figure 2-89 The solid model for Exercise 1

Exercise 2

Draw the sketch shown in Figure 2-90. Extrude the sketch by 30 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-91. You do not need to dimension the drawing.

(Expected time: 20 min)

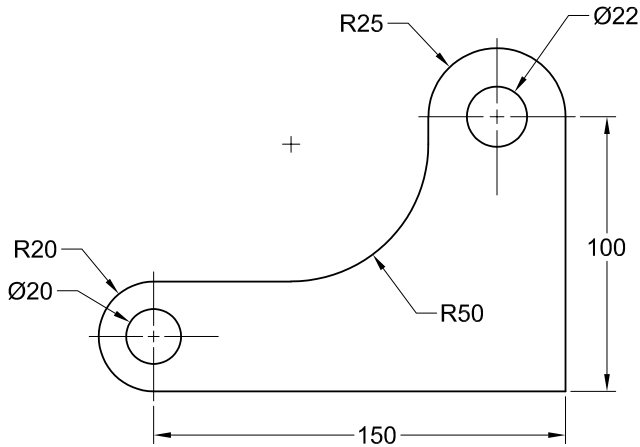


Figure 2-90 The sketch of the model

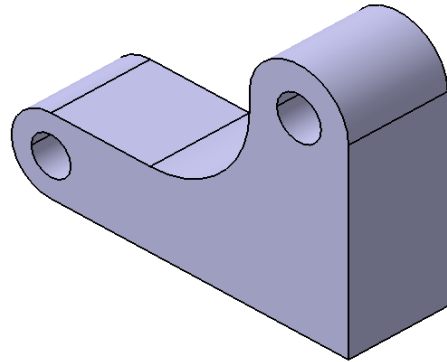


Figure 2-91 The solid model for Exercise 2

Exercise 3

Draw the sketch shown in Figure 2-92. Extrude the sketch by 25 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-93. You do not need to dimension the drawing.

(Expected time: 25 min)

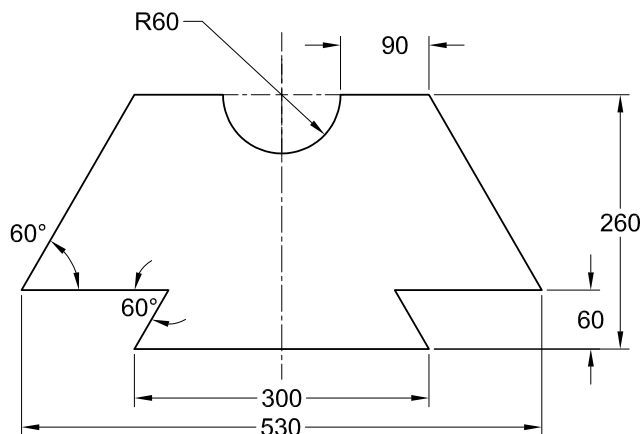


Figure 2-92 The sketch of the model

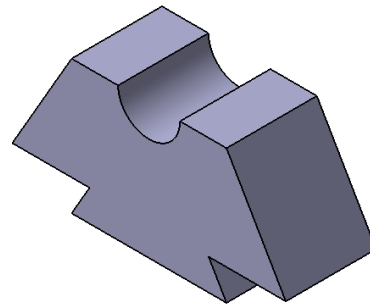


Figure 2-93 The solid model for Exercise 3

Exercise 4

Draw the sketch shown in Figure 2-94. Extrude the sketch by 15 units and then generate a drawing with the orthographic and isometric views of the model. The isometric view of the model is shown in Figure 2-95. You do not need to dimension the drawing.

(Expected time: 30 min)

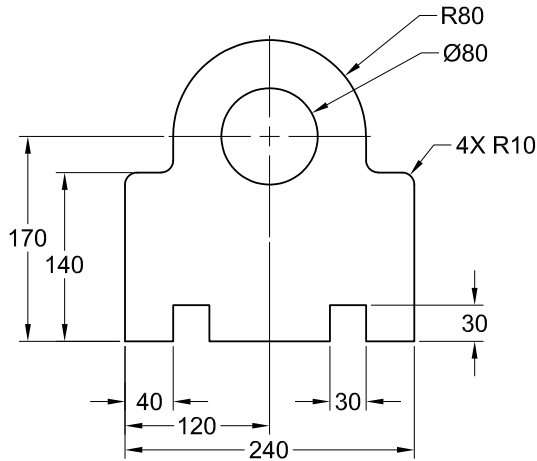


Figure 2-94 The sketch of the model

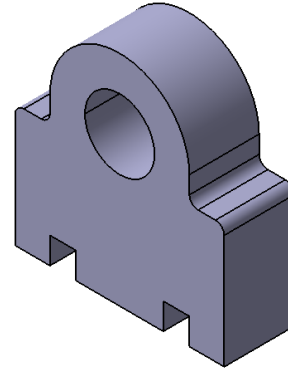


Figure 2-95 The solid model for Exercise 4

Answers to Self-Evaluation Test

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