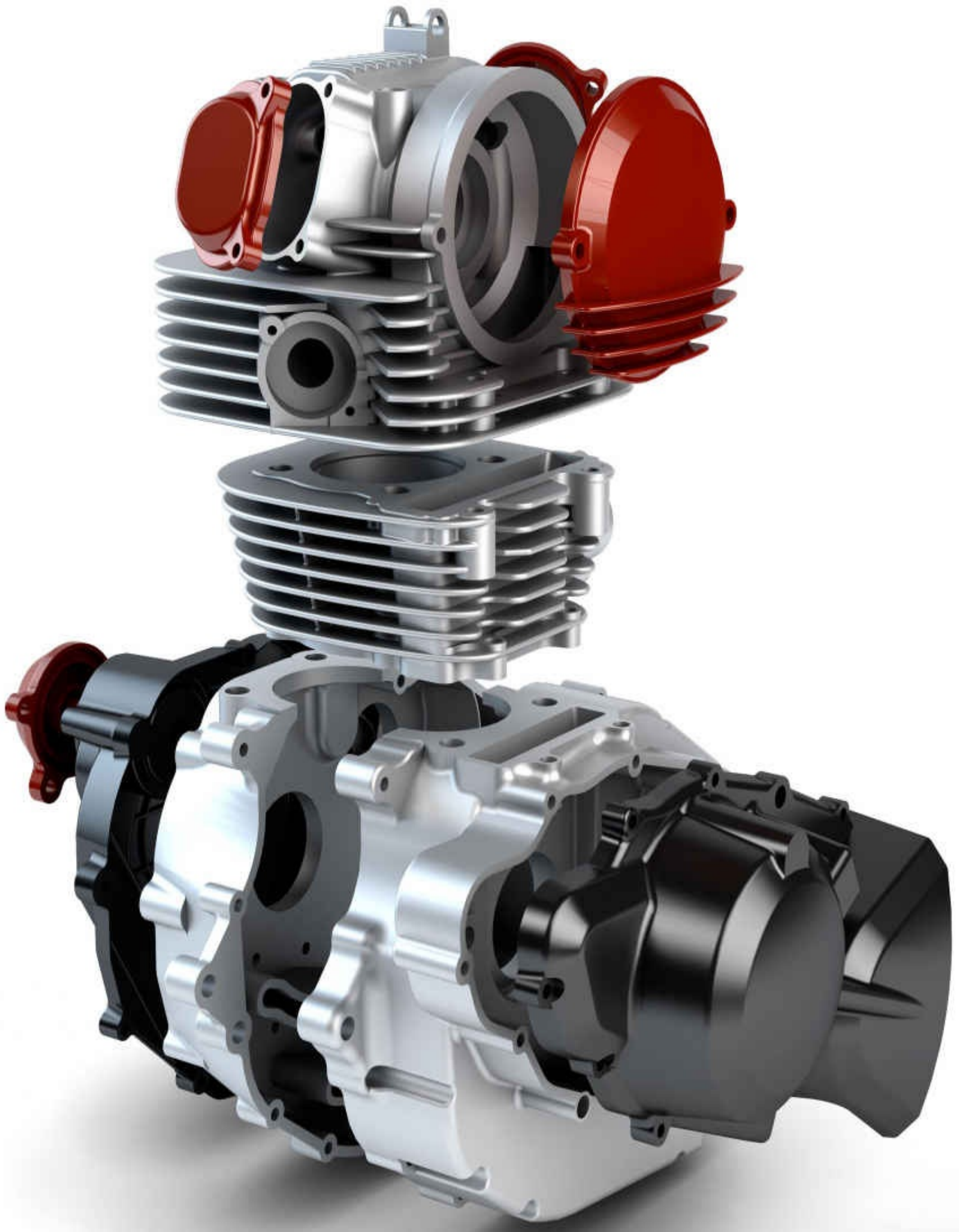
# CATIA V5-6R2015 Basics - Part I

## Getting Started and Sketcher Workbench



Tutorial Books

# CATIA V5-6R2015

# Basics – Part I

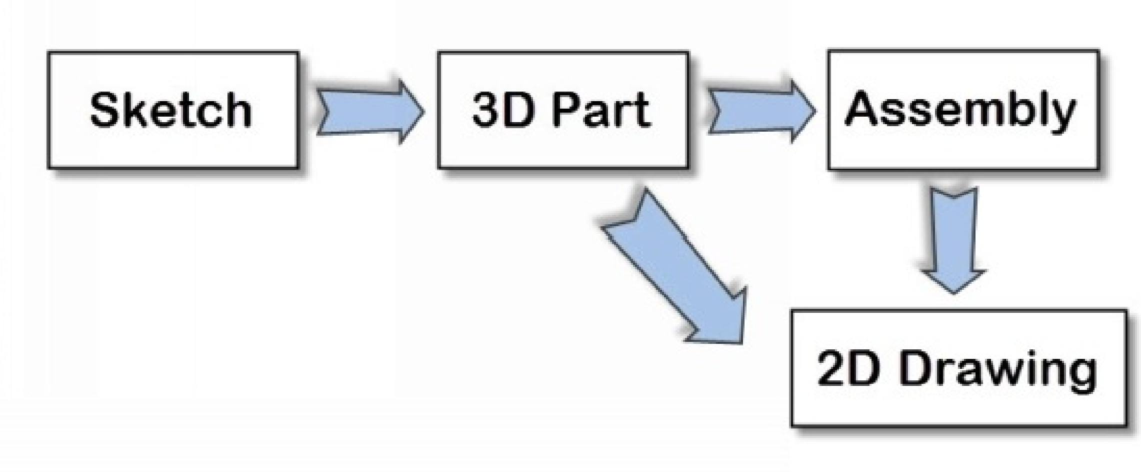
# Tutorial Books

# Contents

# Getting Started with CATIA V5-6R2015

## Introduction to CATIA V5-6R2015

CATIA V5-6R2015 is a parametric and feature-based system that allows you to create 3D parts, assemblies, and 2D drawings. The design process in CATIA V5 is shown below.

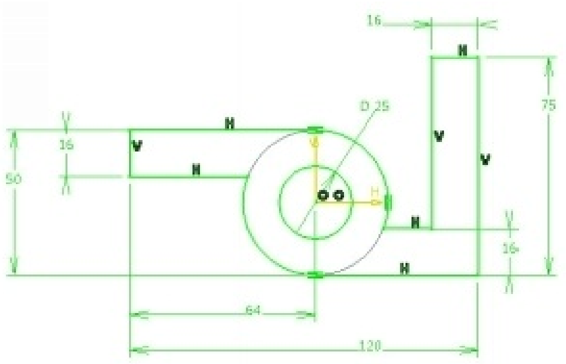


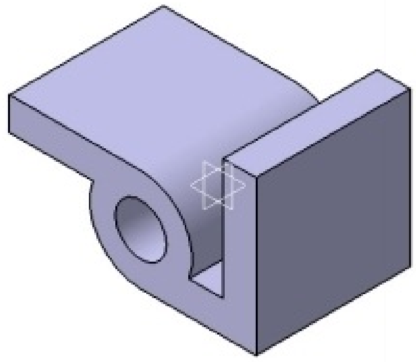

## Workbenches in CATIA V5-6R2015

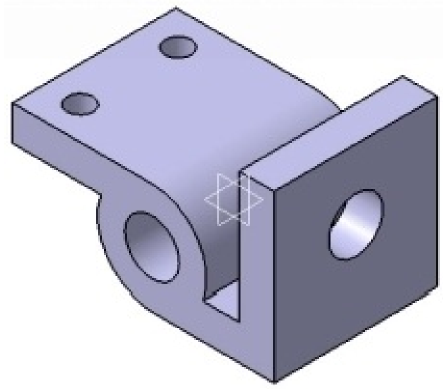
CATIA V5 offers many workbenches to perform various design processes. For example, CATIA V5 provides you with the **Generative Sheetmetal Design** workbench to design a sheet metal part. Likewise, there are many workbenches to perform advanced operations such are static analysis, mold design, automotive design, and so on. However, in this book we cover the basic workbenches such as **Sketcher**, **Part Design**, **Assembly Design**, **Drafting**, **Generative Sheetmetal Design**, and **Generative Shape Design**. A brief introduction to these workbenches is given next.

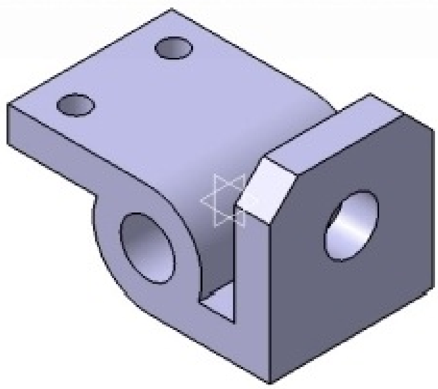
### Part Design

The **Part Design** workbench provides you with commands to create parametric solid models. You can activate this workbench by clicking **Start > Mechanical Design > Part Design** on the Menu bar. To create solid models, you must draw parametric sketches in the **Sketcher** workbench, and then convert them into solids. However, you can add some additional features to the solid models, which do not require sketches.







## Assembly Design

The **Assembly Design** workbench (click **Start > Mechanical Design > Assembly Design**) has commands to combine individual parts in an assembly. There are two ways to create an assembly. The first way is to create individual parts and assemble them in the **Assembly Design** Workbench (Bottom-up assembly design). The second way is to start an assembly file and create individual parts in it (Top-down assembly design).

## Drafting

The **Drafting** workbench **(**click **Start** > **Mechanical Design** > **Drafting**) has commands to create 2D drawings, which can be used for the manufacturing process. There are two ways to create drawings. The first way is to generate the standard views of a 3D component or assembly. The second way is to sketch the drawings manually.

## Generative Sheetmetal Design

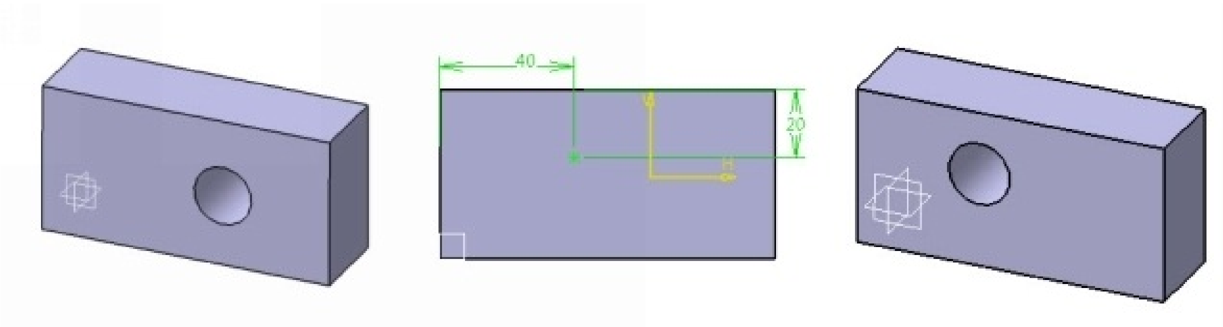
The **Generative Sheetmetal Design** workbench (click **Start** > **Mechanical Design** > **Generative Sheetmetal Design**) has commands to create sheet metal geometry. You can create sheet metal model either by building features in a systematic manner or by converting a part geometry in to a sheet metal.

## Generative Shape Design

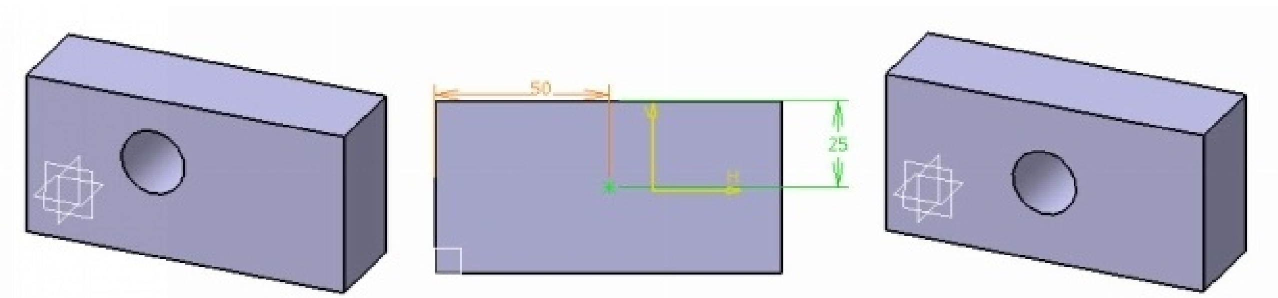
The **Generative Shape Design** workbench (click **Start** > **Shape** > **Generative Shape Design**) has commands to create complex geometries, which cannot be created by using the commands in the **Part Design** workbench. You can create a surface geometry, and then convert it into a solid geometry. A surface is an infinitely thin feature, which acts as a reference. Whereas, a solid geometry has properties such as weight, center of gravity, and so on.

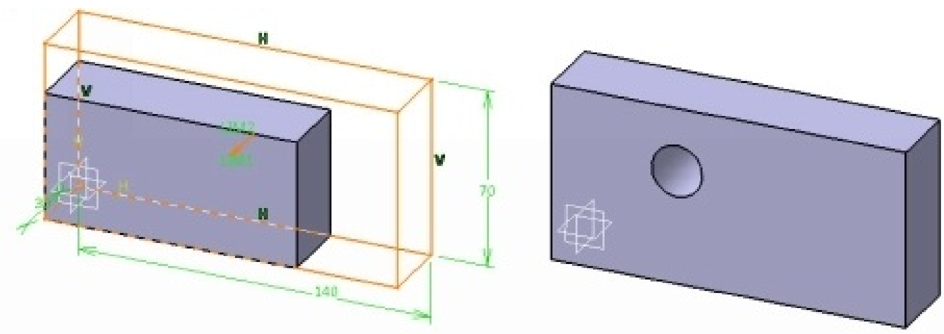
## Parametric Modeling in CATIA V5

In CATIA V5, parameters, dimensions, or constraints control everything. For example, if you want to change the position of the hole shown in figure, you need to change the dimension or constraint that controls its position.

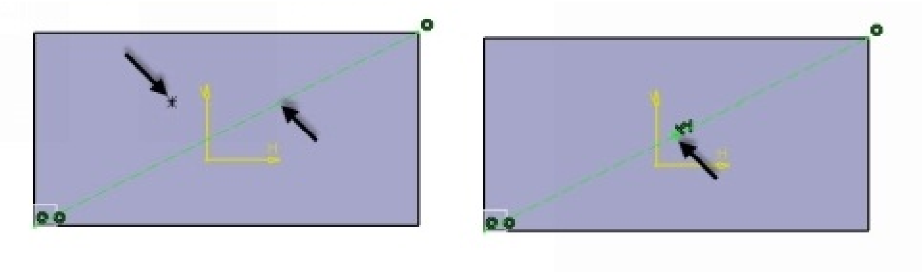


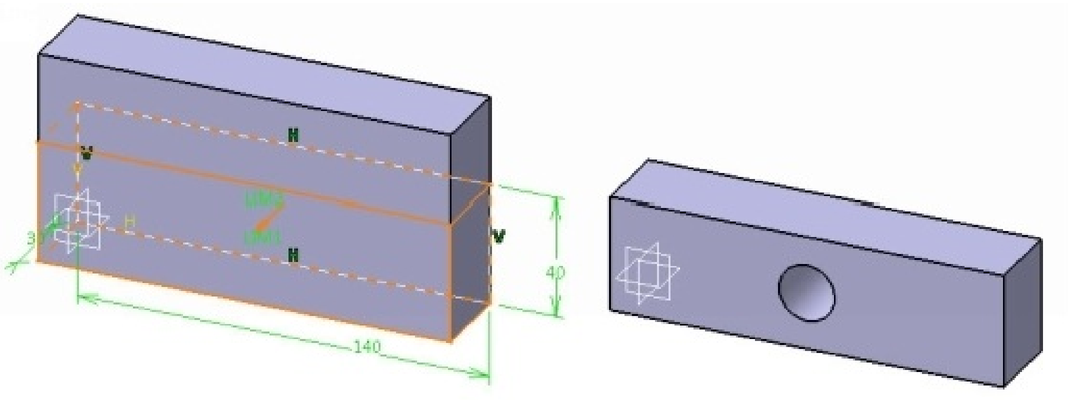

The parameters and constraints that you set up allow you to have control over the design intent. The design intent describes the way your 3D model will behave when you apply dimensions and constraints to it. For example, if you want to position the hole at the center of the block, one way is to add dimensions between the hole and the adjacent edges. However, when you change the size of the block, the hole will not be at the center.



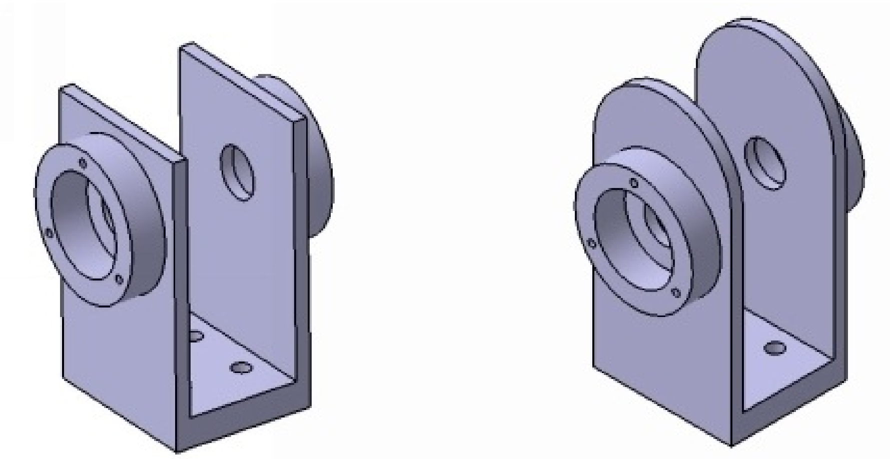

You can make the hole to be at the center, even if the size of the block changes. To do this, you need to delete the dimensions and create a diagonal line. Next, apply the **Midpoint** constraint between the hole point and the diagonal line. Now, even if you change the size of the block, the hole will always remain at the center.

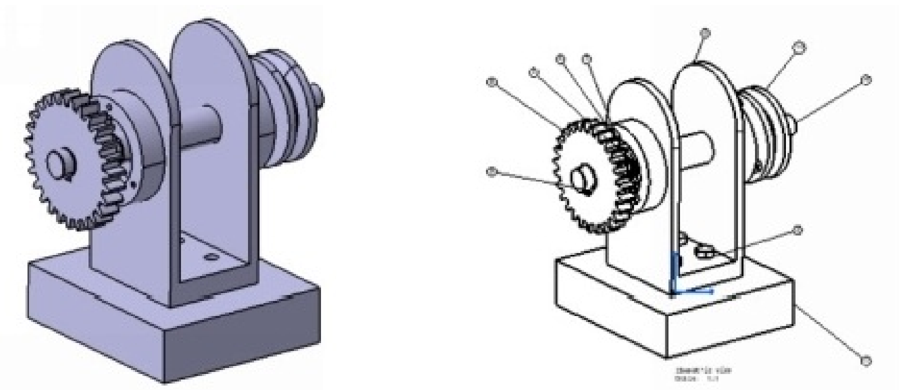





# Associativity

The other big advantage of CATIA V5 is the associativity between parts, assemblies and drawings. When you make changes to the design of a part, the changes will take place in any assembly that it is a part of. In addition, the 2D drawing will update automatically.





# File Types in CATIA V5

CATIA V5 offers three main file types:

**CATPart:** This type of file has geometry of individual part. The files created in **Sketcher**, **Part Design**, **Generative Sheetmetal Design**, and **Wireframe and Surface Design**, and so on will have this extension.

**CATProduct:** This type of file is an assembly of one or more parts. In fact, it is a link of one or more parts.

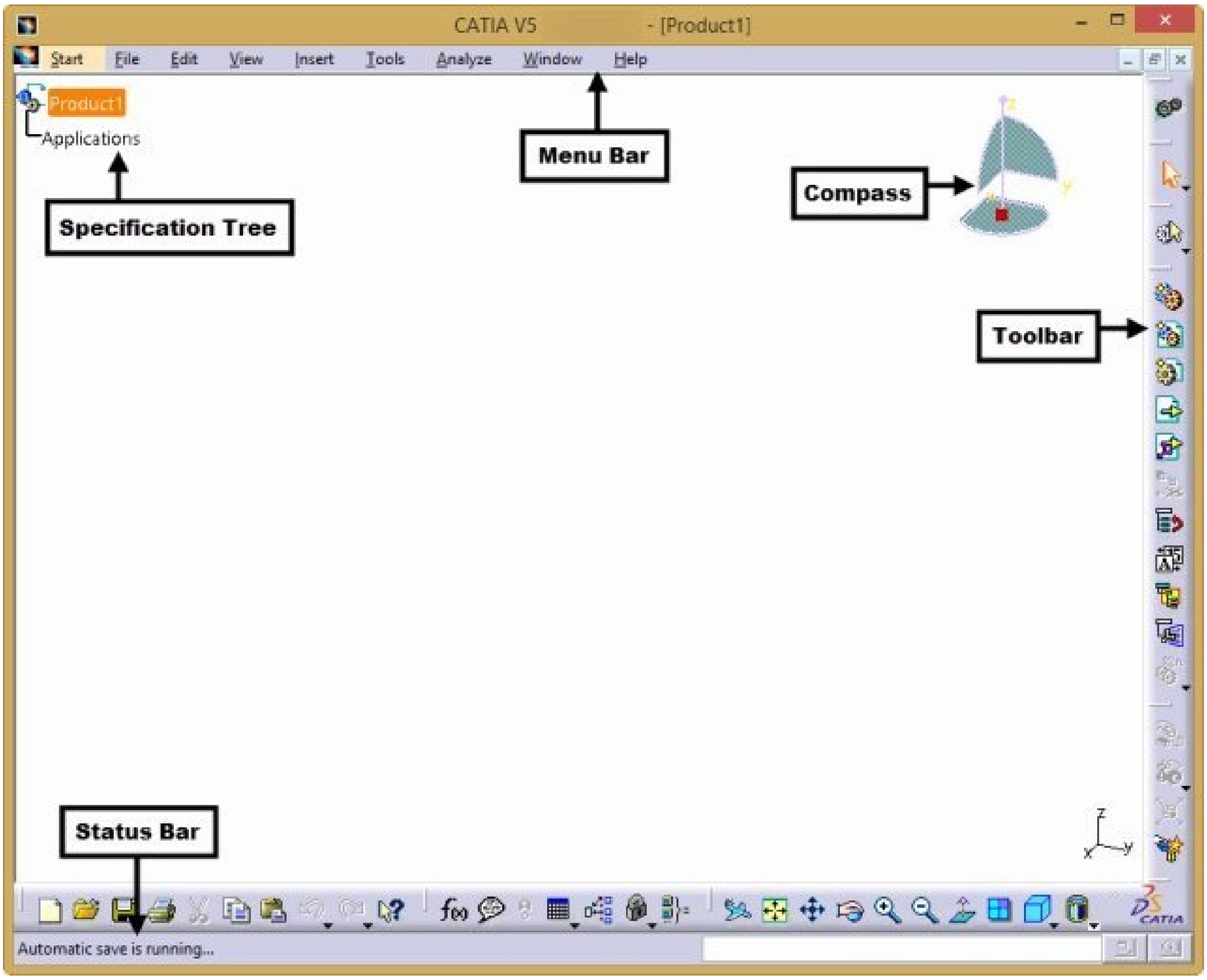
**CATDrawing:** The files created in the Drafting workbench have this extension.

# Starting CATIA V5-6R2015

To start **CATIA V5-6R2015**, click the **CATIA V5-6R2015** icon on your computer screen (or) click **Start > All Programs > CATIA > CATIA V5-6R2015**. If you are working in Windows 8, then click the Windows icon on the bottom left corner. Click the down arrow on the Start screen. On the Apps screen, go to the CATIA section and click the **CATIA V5-6R2015** icon.

# User Interface

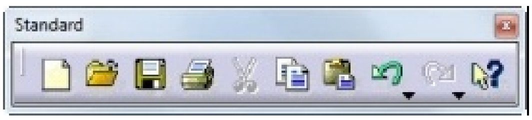
The following image shows the **CATIA V5-6R2015** application window.




Various components of the user interface are:

## Standard Toolbar

This toolbar has some commonly used commands such as **New**, **Open**, **Save**, **Undo, Redo, Copy**, and so on.

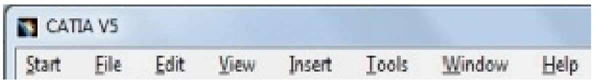



## Start Menu

The **Start Menu** appears when you click on the **Start** button located at the top left corner of the window. The **Start Menu** has a list of workbenches. You can switch between different workbenches using this menu.

## Menu bar

Menu bar is located at the top of the window. It has various options (menu titles). When you click on a menu title, a drop-down appears. Select any option from this drop-down.

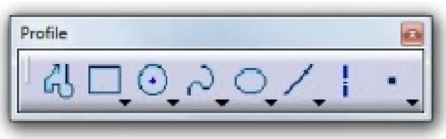



## Toolbar

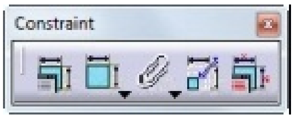
A toolbar is a set of commands, which help you to perform various operations. Various toolbars available in different workbenches are given next.

## Part Design Toolbars

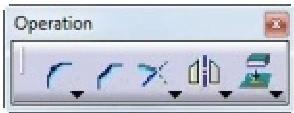

Starts the **Sketcher** workbench




This toolbar has commands to create sketch elements




This toolbar has commands to apply constraints between sketch elements.

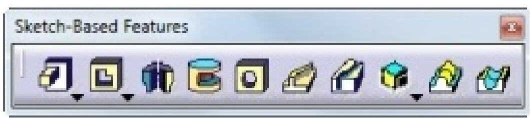



This toolbar has commands to modify sketch elements.

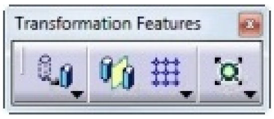

This toolbar has options to filter the type of element that can be selected.


This toolbar has options that help you to create sketch elements.


Exits the workbench.

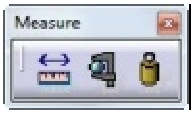



This toolbar has commands to create solid features based on the sketch geometry.




This toolbar has commands to replicate solid features.


This toolbar has commands to additional features, which do not require any sketch.




This toolbar has commands to measure physical properties of the geometry.

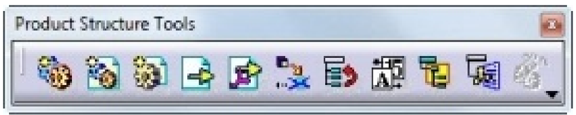

Sections the geometry to view its inside portion.


Applies material to a solid geometry.

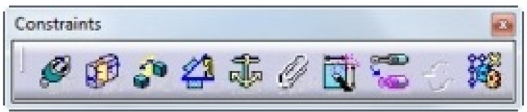

This toolbar has commands to convert a surface model in to solid.


This toolbar has commands to zoom, pan, rotate, or change the view of a 3D model.

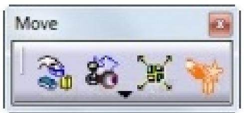
## Assembly Design Toolbars




This toolbar has commands to create components or insert existing components into an assembly.



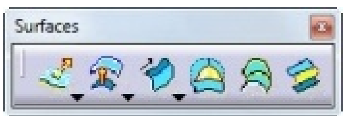

This toolbar has commands to apply constraints between components.



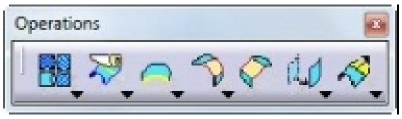

This toolbar has commands to manipulate the position of a component.

## Generative Shape design Toolbars


This toolbar has commands to create three dimensional curves and wireframe geometry.

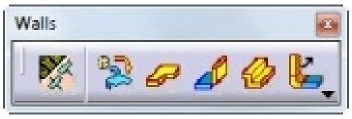



This toolbar has commands to create surfaces.



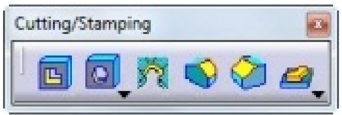

The commands on this toolbar help you modify or transform surfaces.

## Generative Sheetmetal Design Toolbars



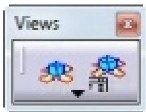

The commands on this toolbar help you to create walls of a sheet metal part.


The commands on this toolbar help you to apply bends to a sheet metal wall.



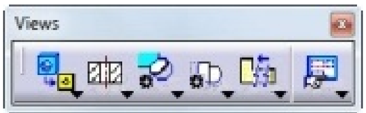

This toolbar has commands to add cuts and stamps to a sheet metal part.


This toolbar has commands to create rolled sheets and funnels.

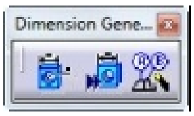



This toolbar has commands to switch between folded and unfolded views of a sheet metal part.

## Drafting Toolbars

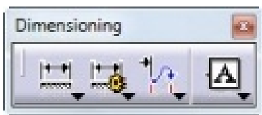



This toolbar has commands to generate standard views of a 3D geometry.




This toolbar has commands to generate dimensions and balloons.

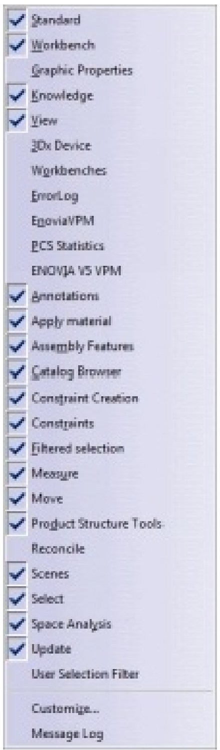

The commands on this toolbar will help you to add a new sheet, drawing view and so on.




The commands on this toolbar help you to add driven dimensions to the drawing views.

This toolbar has commands to add centerlines, hatches, and arrows the drawing view.


Some toolbars are not visible by default. To display a particular toolbar, right-click on any toolbar, and then select the toolbar name from the list displayed.

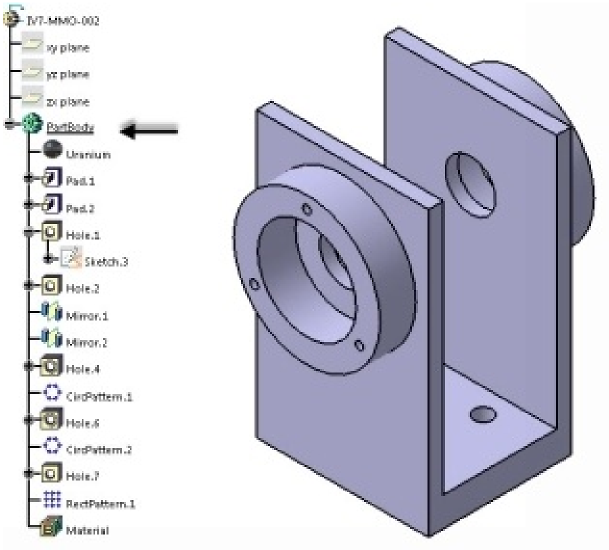



## Status bar

This is available below the graphics window. It shows the prompts and the action taken while using the commands.

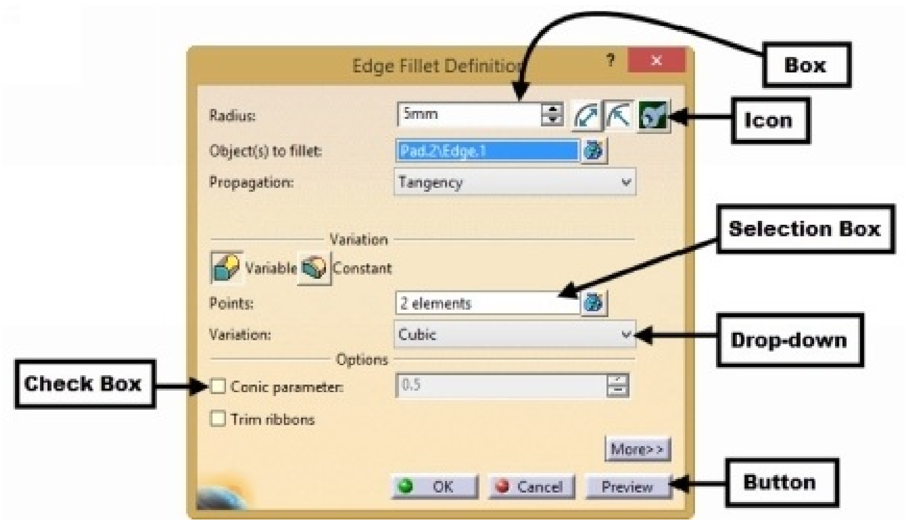

## Specification Tree

Contains the list of operations carried while constructing a part.




## Dialogs

When you execute any command in CATIA V5, the dialog related to it appears. A dialog has various options. The following figure shows various components of a dialog.




This textbook uses the default options on the dialog.

## Mouse Functions

Various functions of the mouse buttons are:

## Left Mouse button (MB1)

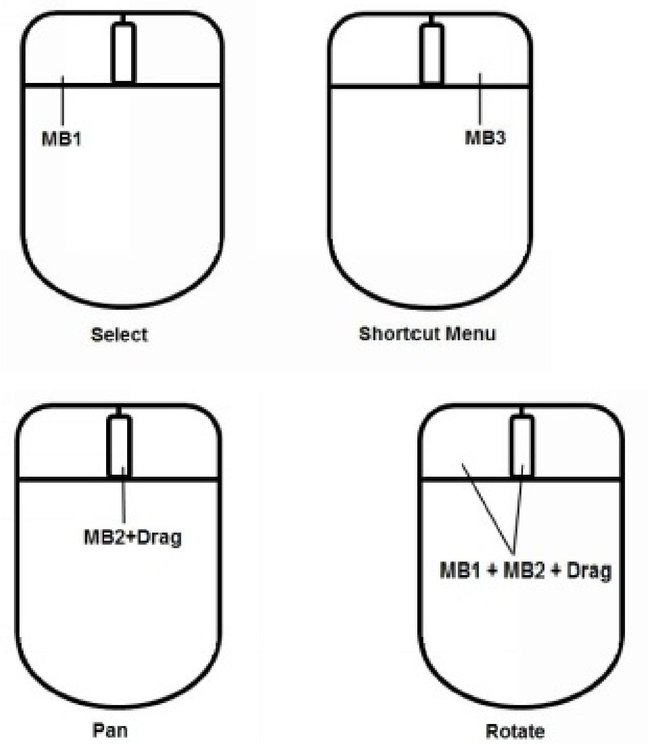
When you double-click the left mouse button (MB1) on an object, the dialog related to the object appears. Using this dialog, you can edit the parameters of the objects.

## Middle Mouse button (MB2)

Press the middle mouse and drag the mouse to pan the view.

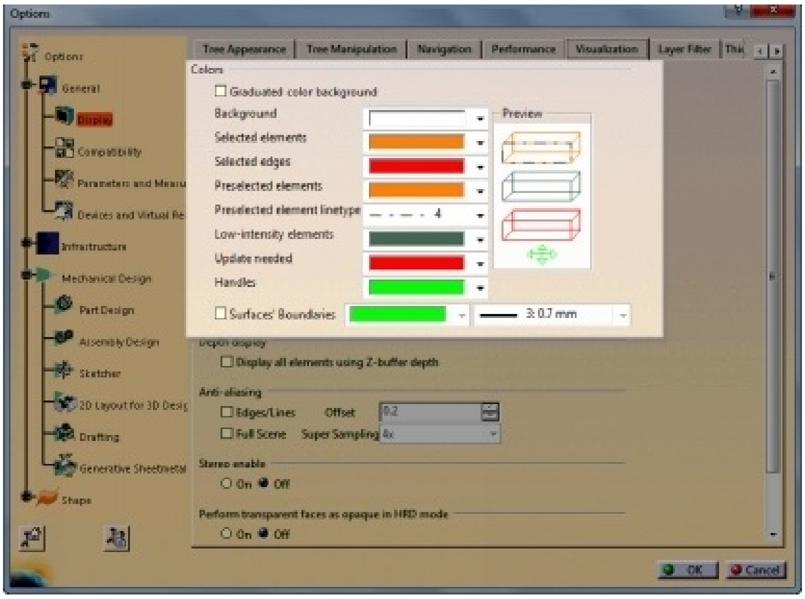
## Right Mouse button (MB3)

Click this button on an object to open the shortcut menu related to it.

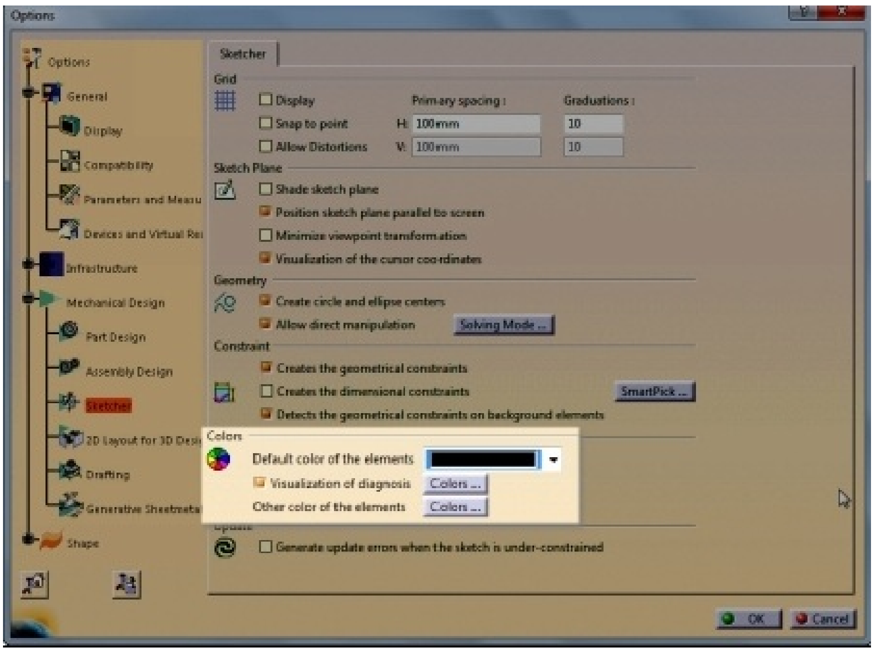


## Background

To change the background color of the window, click **Tools > Options** on the Menu bar. On the **Options** dialog, click **General > Display** on the left side. Click the **Visualization** tab and set the colors of various element types.




To change the color of sketch elements, click **Mechanical Design > Sketcher** on the left side, and then change the **Default color of the elements**.




## Shortcut Keys

| | |
|---|---|
| CTRL+Z | Undo |

| | |
|---|---|
| CTRL+Y | Redo |
| CTRL+S | Save |
| F1 | CATIA V5 Help |
| CTRL+N | New File |
| CTRL+O | Open File |
| CTRL+P | Plot |
| Shift+Left | Rotate To The Left |
| Shift+Right | Rotate To the Right |
| Shift+Up | Rotate Upward |
| Shift-Down | Rotate Downward |
| Alt+F8 | Start Macros |
| Alt+F11 | Visual Basic |
| Ctrl+Page Up | Zoom In |
| Ctrl+Page Down | Zoom Out |
| Ctrl+Left | Pan Left |
| Ctrl+Right | Pan Right |
| Ctrl+Up | Pan Up |
| Ctrl+Down | Pan Down |
| Ctrl+Shift+Left | Rotate Around Z Axis clockwise |
| Ctrl+Shift+Right | Rotate Around Z Axis CounterClockwise |
| Ctrl+Tab | Swap Windows |
| Ctrl+F | Search |

| | |
|---|---|
| Ctrl+U | Update |
| Ctrl+X | Cut |
| Ctrl+C | Copy |
| Ctrl+V | Paste |

# Questions

1. Explain how to display hidden toolbars.
2. What is the design intent?
3. Give one example of where you would establish a constraint between a part's features.
4. Explain the term 'associativity' in CATIA V5.
5. List the procedure to access CATIA V5 Help.
6. How can you change the background color of the graphics window
7. How can you activate the shortcut Menu?
8. How is CATIA V5 a parametric modeling application?

# Sketcher Workbench

This chapter covers the methods and commands to create sketches used in the Sketcher Workbench. In CATIA V5, you can create sketches in the Sketcher Workbench. You will learn to create sketches in this Workbench.

In CATIA, you create a rough sketch, and then apply dimensional and geometric constraints that define its shape and size. The dimensional constraints define the length, size, and angle of a sketch element, whereas geometric constraints define the relations between sketch elements.

The topics covered in this chapter are:

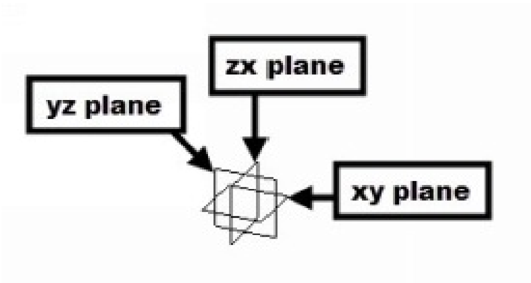
- Sketching in Sketcher Workbench
- Use geometric and dimensional constraint to control the shape and size of a sketch
- Learn sketching commands
- Learn commands and options that help you to create sketches easily

## Sketching in the Sketcher Workbench

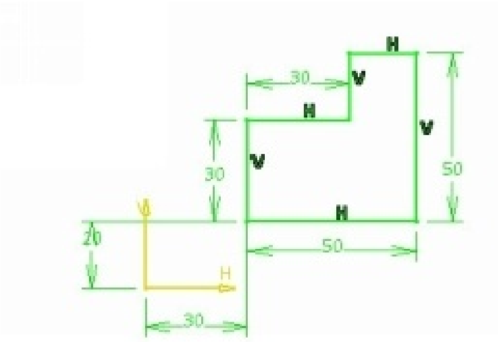
Creating sketches in the Sketcher Workbench is very easy. You have to activate the **Sketch** command, and then define a plane on which you want to create the sketch.

1. On the **Sketcher** toolbar, click the **Sketch** icon (or) click **Insert > Sketcher > Sketch** on the menu.


2. Click on any of the reference planes located at the center of the graphics window.



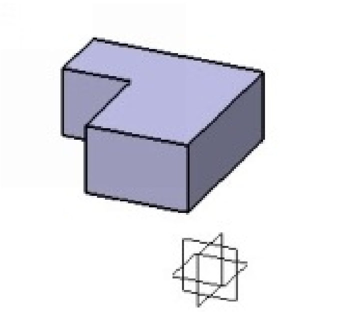

3. You can now start drawing sketches on the selected plane.



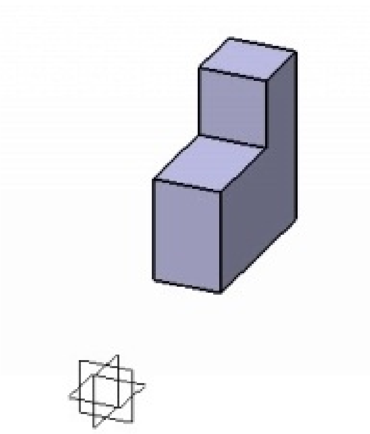

4. After creating the sketch, click **Workbenches** Toolbar > **Exit Workbench** to exit the sketch.


The following figures show the orientation of the part when the sketch is created on three different planes.

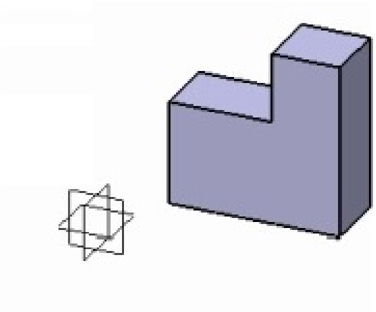
**XY plane**



**YZ Plane**

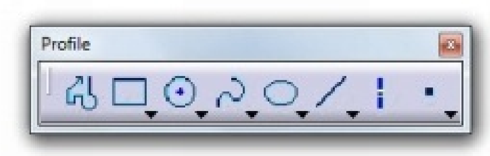


**ZX Plane**



# Draw Commands

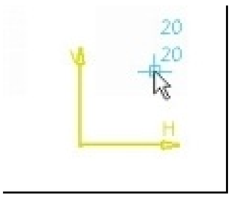
CATIA V5 provides you with a set of commands to create sketches. These commands are located on the **Profile** toolbar.



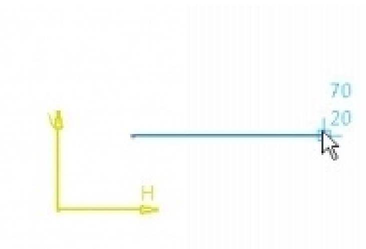

##  The Profile command

This is the most commonly used command while creating a sketch.

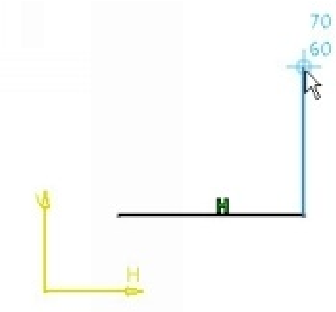
1. To activate this command, click the **Profile** button on the **Profile** toolbar (or) click **Insert > Profile > Profile** on the menu. As you move the pointer in the graphics window, you will notice the X and Y coordinates of the pointer.




2. To create a line, click in the graphics window, move the pointer and click again. After clicking for the second time, you can see an end point is added and another line segment is started. This is a convenient way to create a chain of lines.



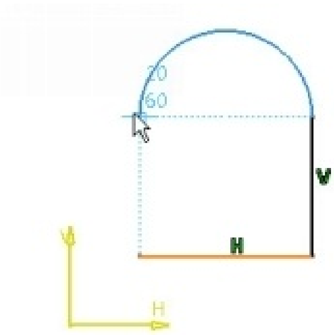

3. Continue to click to add more line segments.




The **Profile** command can also be used to draw arcs continuous with lines.

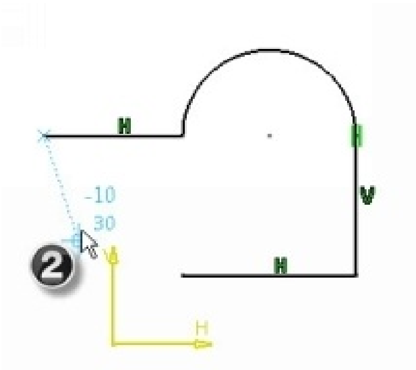
4. On the **Sketch Tools** toolbar, click the **Tangent Arc** button.

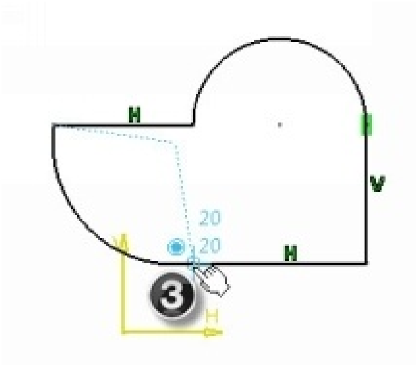

5. Move the pointer and click to draw an arc tangent to the previous line.



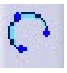

6. On the **Sketch Tools** toolbar, click the **Three Point Arc** button to create an arc normal to the previous line.


7. Define the second and third points of the arc.



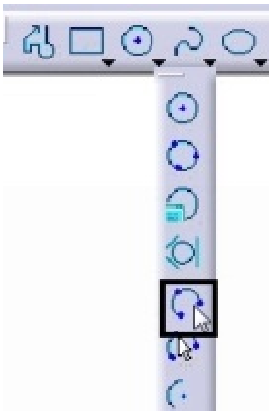



To delete a line, select it and press the **Delete** key. To select more than one line, press the Ctrl key and click on multiple line segments; the lines will be highlighted. You can also select multiple lines by dragging a box from left to right. Press and hold the left mouse button and drag a box from left to right; the lines inside the box boundary will be selected.

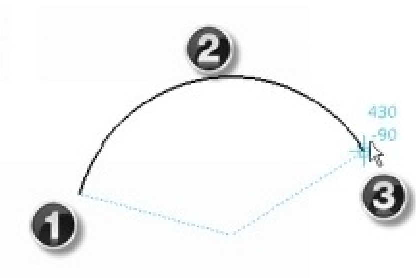
##  Three Point Arc

This command creates an arc by clicking three points in the graphic window.

1. On the **Profile** toolbar, click **Circle** drop-down **> Three Point Arc**.



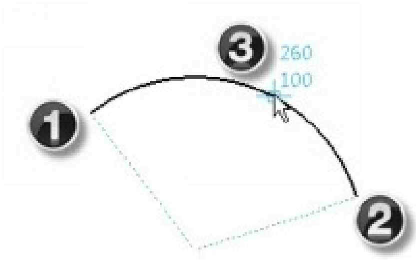
2. Click to define the start point of the arc.
3. Move the pointer and click to define a point on the periphery of the arc.
4. Again, click to define the end point.

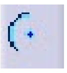



##  Three Point Arc Starting with Limits

This command creates an arc by defining its start, end, and radius.

1. On the **Profile** toolbar, click **Circle** drop-down > **Three Point Arc Starting with Limits**.
2. Click to define the start point of the arc.
3. Move the pointer and click again to define the end point.
4. After defining the start and end of the arc, you need to the define size of the arc. Move the pointer and click to define the radius of the arc.

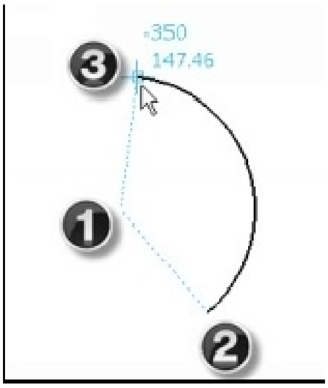


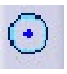



## Arc

This command creates an arc by defining its center, start and end.

1. On the **Profile** toolbar, click **Circle** drop-down > **Arc**.
2. Click to define the center point.
3. Next, move the pointer and you will notice that a circle appears attached to the pointer. This defines the radius of the arc.
4. Now, click to define the start point of the arc and move the pointer; you will notice that an arc is drawn from the start point.
5. Once the arc appears the way you want, click to define its end point.

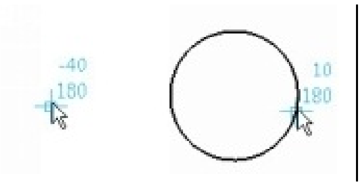


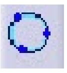



## Circle

This is the most common way to draw a circle.

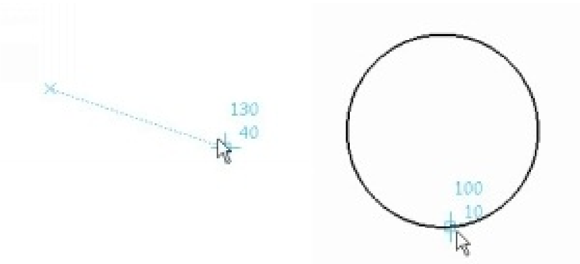
1. Click the **Circle** icon on the **Profile** toolbar, (or) click **Insert** > **Profile** > **Circle** > **Circle** on the menu.
2. Click to define the center point of the circle.
3. Drag the pointer, and then click again to define the diameter of the circle.

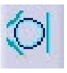



##  Three Point Circle

This command creates a circle by using three points.

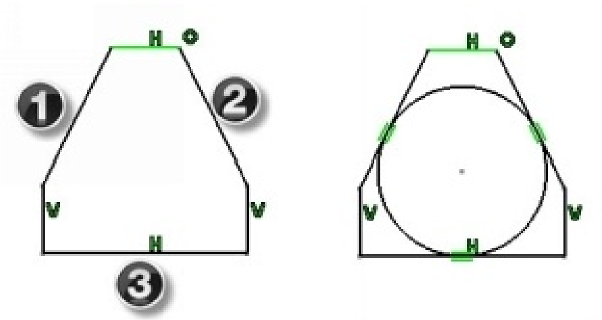
1. On the **Profile** toolbar, click **Circle** drop-down > **Three Point Circle**.
2. Select three points from the graphics window. You can also select existing points from the sketch geometry. The first two points define the location of the circle and the third point defines its diameter.




##  Tri-Tangent Circle

This command creates a circle tangent to three lines, arcs, or circles.

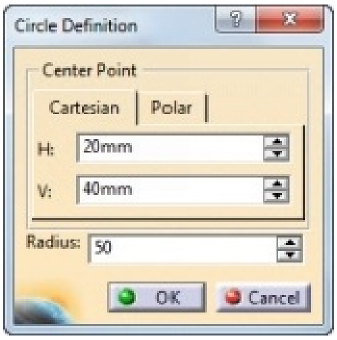
1. On the **Profile** toolbar, click **Circle** drop-down > **Tri-Tangent Circle**.
2. Select three lines, arcs or circles. This creates a circle tangent to selected lines.



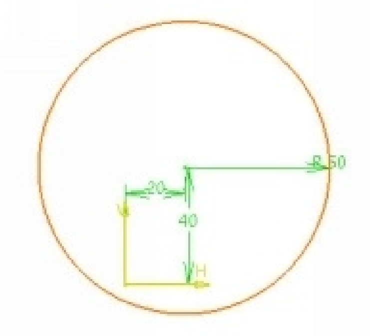

##  Circle Using Coordinates

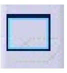
This command creates a circle by using the coordinate values of its center point and the radius value that you specify.

1. On the **Profile** toolbar, click **Circle** drop-down > **Circle Using Coordinates**.
2. On the **Circle Definition** dialog, type-in values in the **X**, **Y** and **Radius** boxes.



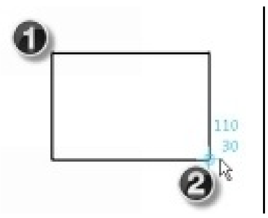

3. Click **OK** to create the circle.

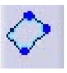



##  Rectangle

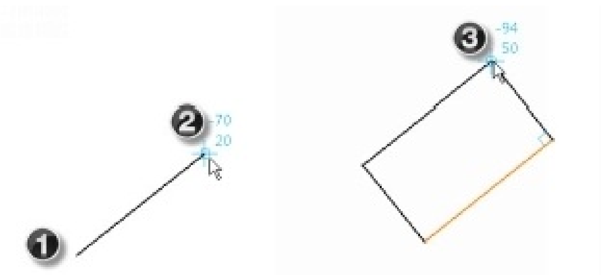
This command creates a rectangle by defining its diagonal corners.

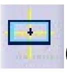
1. On the **Profile** toolbar, click the **Rectangle** icon.
2. Click to define the first corner.
3. Drag the pointer and click to define the second corner.




##  Oriented Rectangle

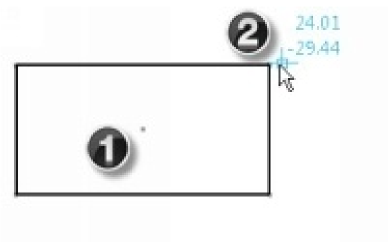
This command creates an inclined rectangle. The first two points define the width and inclination angle of the rectangle. The third point defines its height.

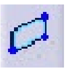


## Centered Rectangle

This command creates a rectangle by defining two points: center of the rectangle and its corner.

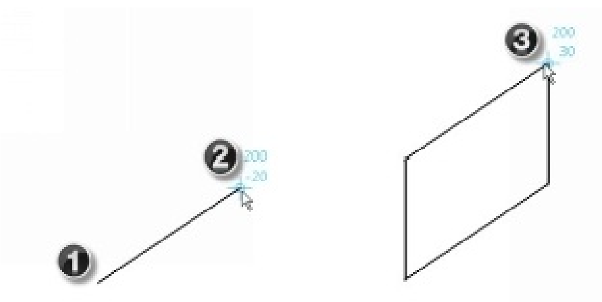
1. On the **Profile** toolbar, click **Rectangle** drop-down **> Centered Rectangle**.
2. Click to define the center of the rectangle.
3. Move the pointer and click again to define the corner point.

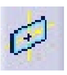



##  Parallelogram

This command creates a parallelogram by using three points that you specify.

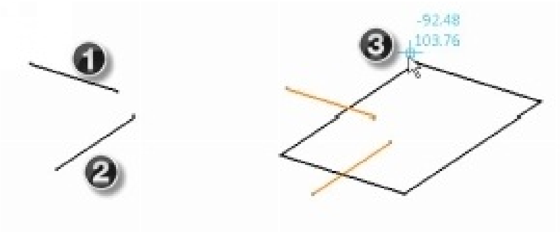
1. On the **Profile** toolbar, click **Rectangle** drop-down > **Parallelogram**.
2. Select two points to define the width of the parallelogram.
3. Drag the pointer and click to define the height of parallelogram.




##  Centered Parallelogram

This command creates a parallelogram by selecting two intersecting lines. The point of intersection will become the center of the parallelogram.

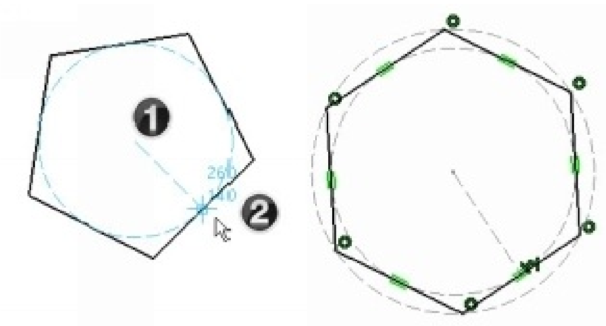
1. On the **Profile** toolbar, click **Rectangle** drop-down **> Centered Parallelogram**.
2. Select two intersecting lines.
3. Drag the pointer and click to define the corner of the parallelogram.



## Polygon

This command provides a simple way to create a closed profile with equal length sides.

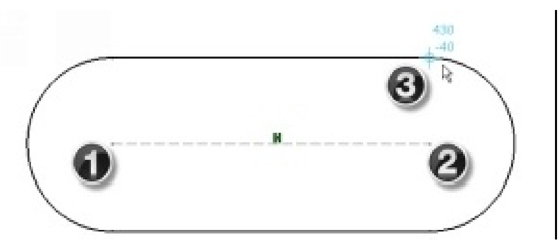
1. On the **Profile** toolbar, click **Rectangle** drop-down **>** **Polygon**.
2. Click to define the center point of the polygon. The pointer will be on one of the vertices of the polygon. This is because the **Circum Circle** option is activated on the **Sketch Tools** toolbar.
3. Click the **In Circle** icon on the **Sketch Tools** toolbar. The pointer will be on one of the flat sides of the hexagon.
4. Drag the pointer and click to define the size and angle of the polygon.
5. Type a value in the **Number of Sides** box on the **Sketch Tools** toolbar, and then press Enter.



## Elongated Hole

This command creates a straight slot by defining its centerline and radius.

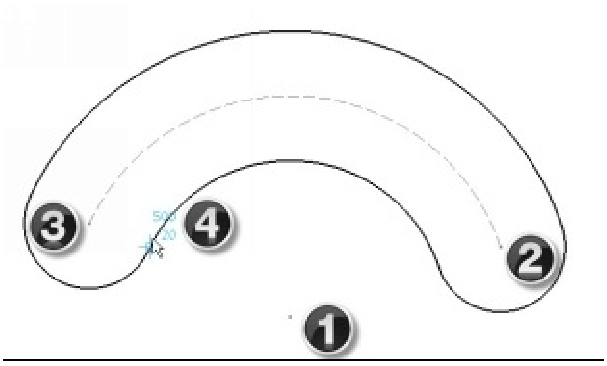
1. On the **Profile** toolbar, click **Rectangle** drop-down > **Elongated Hole**.
2. Click to define the start point of the slot.
3. Drag the pointer and click to define the end-point. This creates the centerline of the slot.
4. Now, drag the pointer and click to define the radius of the slot.



## Cylindrical Elongated Hole

This command creates a curved slot by defining the curve radius and slot radius. First, you have to create an arc, and then create a slot along the arc.

1. On the **Profile** toolbar, click **Rectangle** drop-down > **Cylindrical Elongated Hole**.
2. Click to define the center point of the arc.
3. Drag the pointer and define the start and end points of the arc. This defines the radius and size of the center arc.
4. Now, drag the pointer and click to define the radius of the slot.

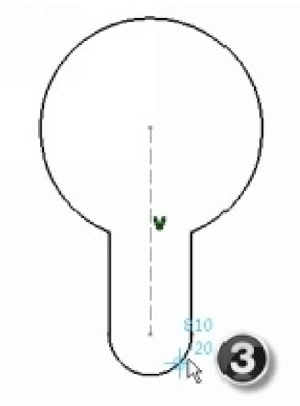



## Keyhole Profile

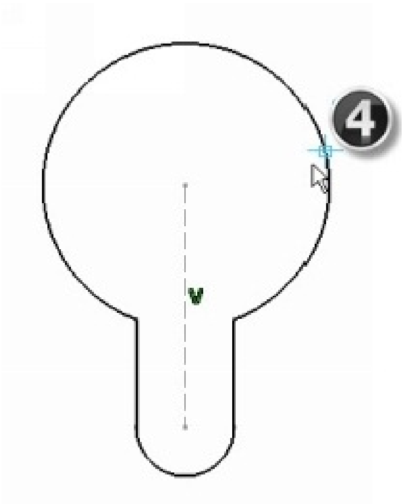
This command creates a keyhole profile. A keyhole profile has a large and small arcs connected through a slot.

1. On the **Profile** toolbar, click **Rectangle** drop-down > **Keyhole Profile**.
2. Click to define the center point of the large arc.
3. Drag the pointer and click to define the center point of the small arc.


4. Now, drag the pointer and click to define the small radius.



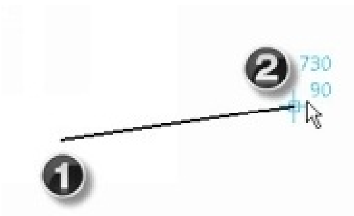

5. Again, drag the pointer and click to define the large radius.



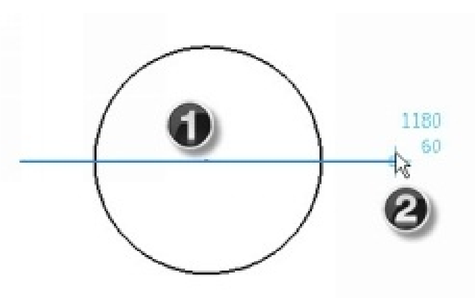

## Line

This command creates a line using the start and endpoints that you select.

1. On the **Profile** toolbar, click **Line** drop-down > **Line**.
2. Click the start and endpoints of the line.




If you want to define its midpoint and endpoint, then click **Symmetrical Extension** on the **Sketch tools** toolbar. Define the mid and endpoints of the straight line.

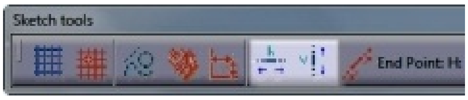


## Infinite Line

This command creates a line with infinite length.

1. On the **Profile** toolbar, click **Line** drop-down > **Infinite Line**.
2. On the **Sketch tools** toolbar, click the **Line Through Two Points** icon.
3. Click to define the origin of the line.
4. Drag the pointer to rotate the line.
5. Click to create an infinite line at an angle.

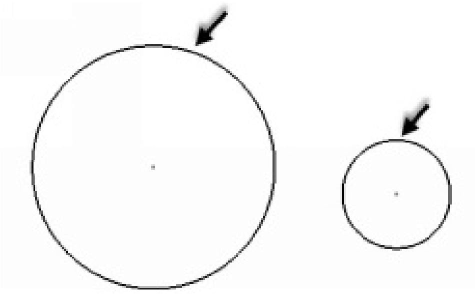
If you want to create a horizontal or vertical infinite line, click the **Horizontal** or **Vertical** button on the **Sketch tools** toolbar.

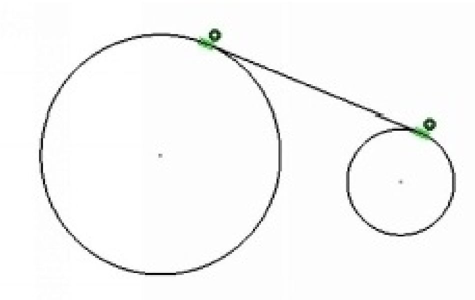



## Bi-Tangent Line

This command creates a line tangent to two circles or arcs.

1. On the **Profile** toolbar, click **Line** drop-down **> Bi-Tangent Line**.
2. Select two circles or arcs. This creates a line tangent to the selected elements.

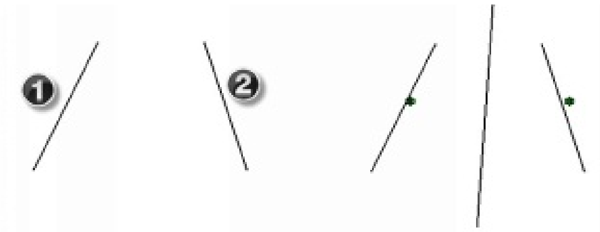


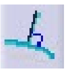


## Bisecting Line

This command creates an infinite line passing through the intersection of two lines. In case of parallel lines, an infinite line will be created at the center and parallel to both the lines.

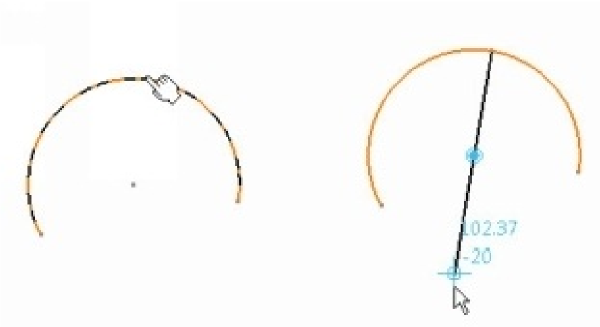
1. On the **Profile** toolbar, click **Line** drop-down > **Bisecting Line**.
2. Select two lines.

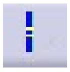


##  Line Normal to Curve

This command creates a line normal to arc, ellipse, circle, spline or any other curve.

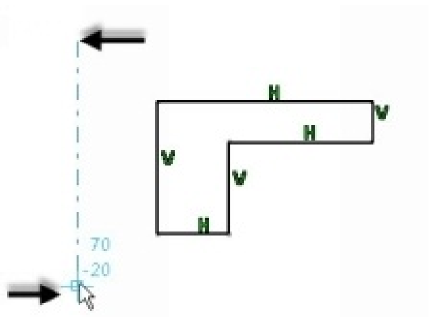
1. On the **Profile** toolbar, click **Line** drop-down > **Line Normal to Curve**.
2. Click on the curve to draw a normal line.
3. Drag the pointer and click to define the endpoint of the line.



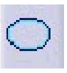

##  Axis

This command creates a sketch axis, which can be used while creating the revolved feature.

1. On the **Profile** toolbar, click the **Axis** icon.
2. Define the start and endpoints of the axis.

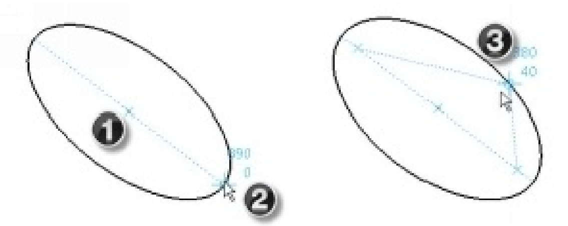



You can also convert an existing line into an axis by selecting it and clicking the **Axis** icon on the **Profile** toolbar.

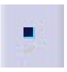
##  Ellipse

This command creates an ellipse using a center point, and major and minor axes.

1. On the **Profile** toolbar, click **Ellipse** drop-down > **Ellipse**.
2. Click to define the center of the ellipse.
3. Drag the pointer and click to define the major axis and orientation of the ellipse.
4. Drag the pointer and click again to define the minor axis.




On the **Sketch tools** toolbar, you can also type-in values in the **Major Radius**, **Minor Radius**, and **A** (angle) boxes.

##  Points by Clicking

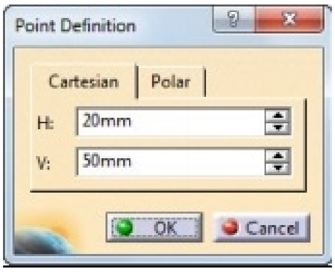
This command creates points as you click in the graphics window.

1. On the **Profile** toolbar, click **Points** drop-down > **Points by Clicking**.
2. Click in the graphics window to create points.

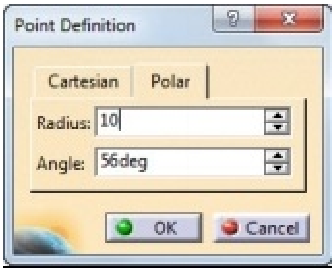
##  Point by Using Coordinates

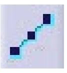
This command creates a point by entering its coordinate values in the Cartesian or Polar coordinate system.

1. On the **Profile** toolbar, click **Points** drop-down > **Points by Using Coordinates**.
2. On the **Point Definition** dialog, click the **Cartesian** tab and type-in values in the **H** and **V** boxes.




3. If you want to enter the coordinate values in the Polar coordinate system, then click the **Polar** tab and type-in values in the **Radius** and **Angle** boxes.

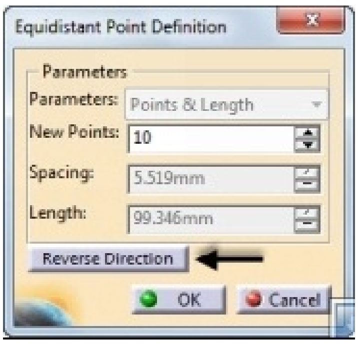



##  Equidistant Points

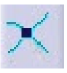
This command creates equidistant points on a selected sketch element.

1. On the **Profile** toolbar, click **Points** drop-down > **Equidistant Points**.
2. Select a sketch element.

3. On the **Equidistant Point Definition** dialog, type-in a value in the **New Points** box.
4. If you want to reverse the side of point creation, then click the **Reverse Direction** button.

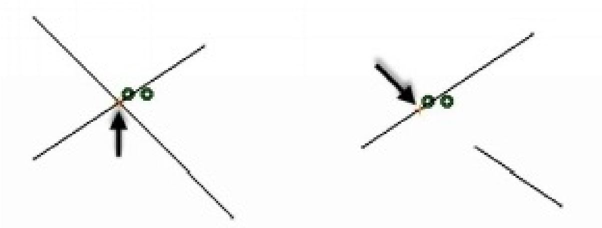


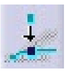
5. Click **OK** to complete the point creation.

##  Intersection Point

This command creates a point at the intersection of two elements.

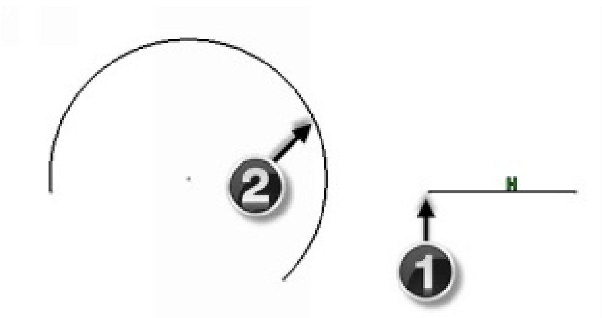
1. On the **Profile** toolbar, click **Point** drop-down > **Intersection Point**.
2. Click on two intersecting elements.

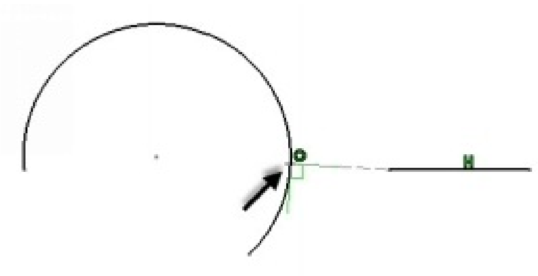


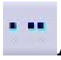
##  Projection Point

This command creates a new point by projecting a point onto a sketch element.

1. On the **Profile** toolbar, click **Point** drop-down > **Projection Point**.
2. Click on the point to be projected.
3. Click on the sketch element onto which the point will be projected.



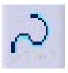





## Align Points

This command aligns a point along a straight line.

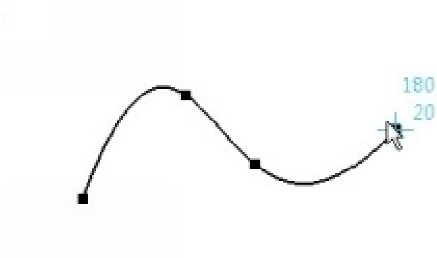
1. On the **Profile** toolbar, click **Point** drop-down > **Align Points**.
2. Click on the point to be aligned.
3. Select another point or click to define the alignment direction. A straight construction line is created and the selected point is aligned along the line.



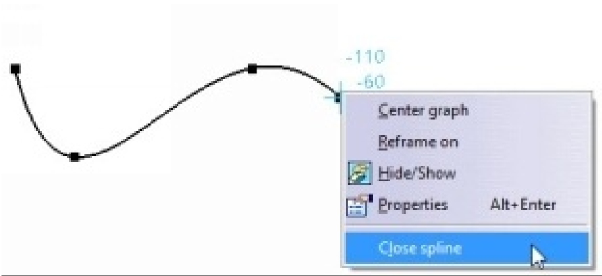
## Spline

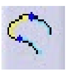
This command creates a smooth B-spline curve passing through the points you select.

1. On the **Profile** toolbar, click **Spline** drop-down > **Spline**.
2. Click to define points in the graphics window. This creates a spline passing through the selected points.
3. Press Esc to deactivate this command.




If you want to create a closed spline, then click the right mouse button and select **Close spline**.

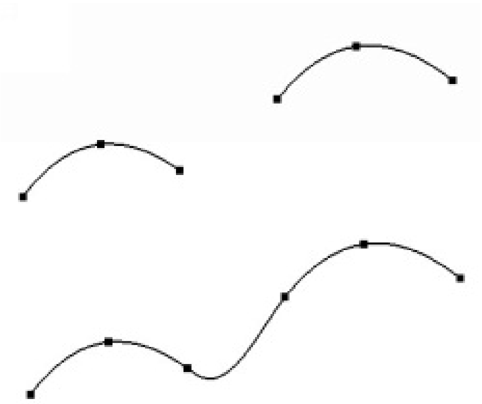


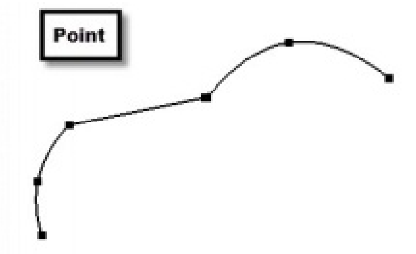
## Connect

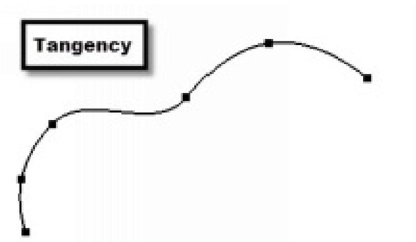
This command connects two splines or curves.

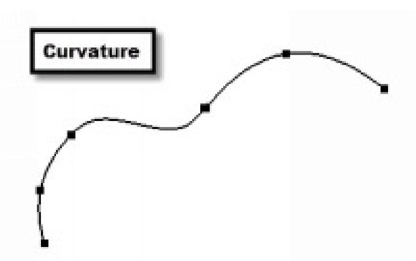
1. On the **Profile** toolbar, click **Spline** drop-down > **Connect**.
2. Click on two open curves to connect them.



On the **Sketch tools** toolbar, you can define the type of connecting curve by using the **Connect with an Arc** and **Connect with a Spline** button. If you click the **Connect with a Spline** button, then you can define the continuity of the bridge curve using the **Continuity in point**, **Continuity in tangency**, and **Continuity in curvature** buttons. The following examples show the continuity types.

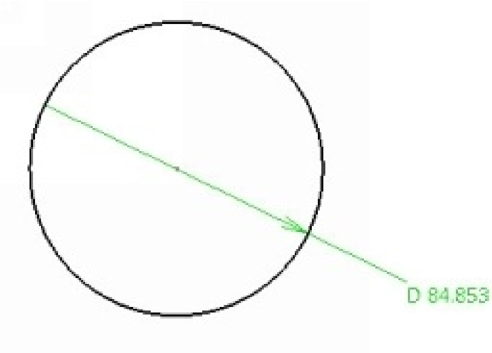


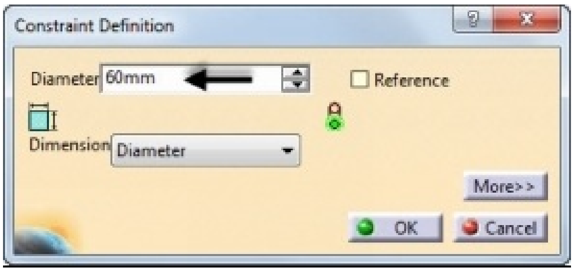



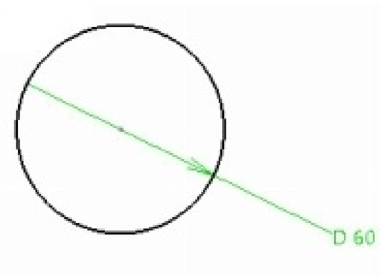

## The Constraint command

It is generally considered a good practice to ensure that every sketch you create is fully constrained before moving on to creating features. The term, ‘fully-constrained’ means that the sketch has a definite shape and size. You can fully-constrain a sketch by using dimensions and constraints. You can add dimensions to a sketch by using the **Constraint** command (on the **Constraint** toolbar, click **Constraint** drop-down > **Constraint**). You can use this command to add all types of dimensional constraints such as length, angle, and diameter and so on. This command creates a dimension based on the geometry you select. For instance, to dimension a circle, activate the **Constraint** command, and then click on the circle. Next, move the pointer and click again to position the dimension.

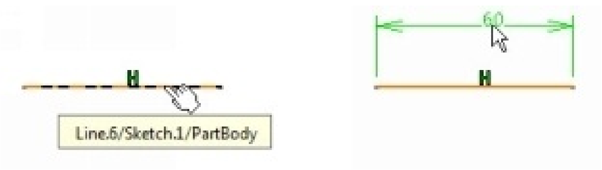



Now, you can change the size of the sketch element by modifying the dimension value. To do this, double-click on the dimension. You will notice that the **Constraint Definition** dialog pops up. Type-in a value in this box, and click **OK** to update the dimension.

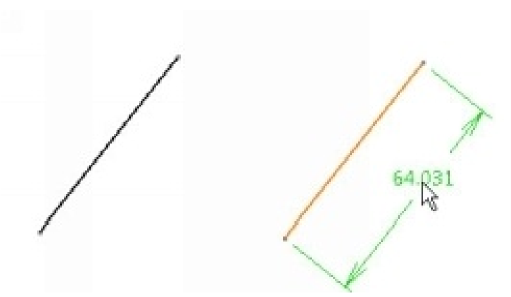


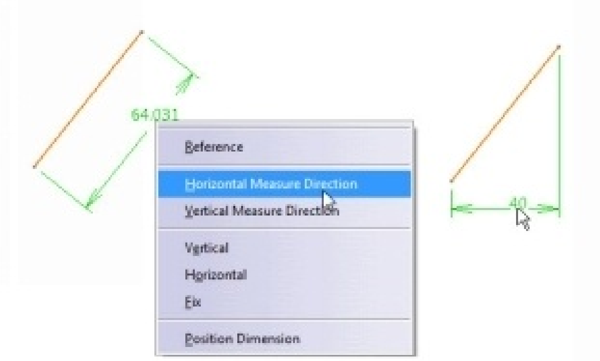

If you click a line, this command automatically creates a linear dimension. Click once more to position the dimension.



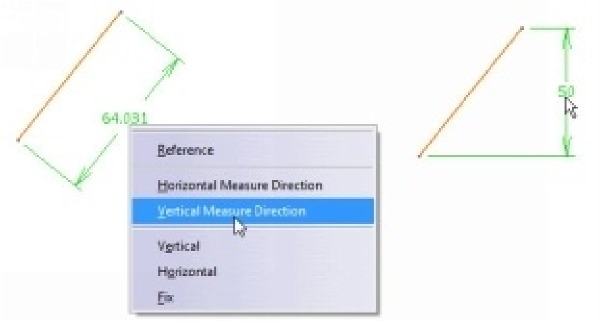

If you click on an inclined line, this command creates a dimension parallel to the line.



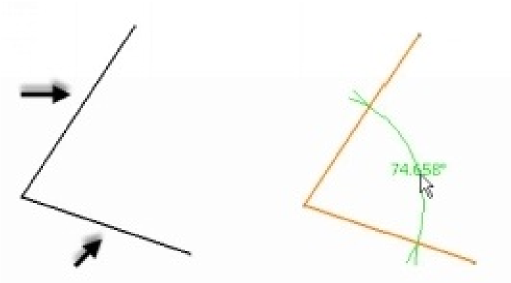
If you want to create a horizontal dimension, then click the right mouse button and select **Horizontal Measure Direction**.




Likewise, if you want to create a vertical dimension, then click the right mouse button and select **Vertical Measure Direction**.

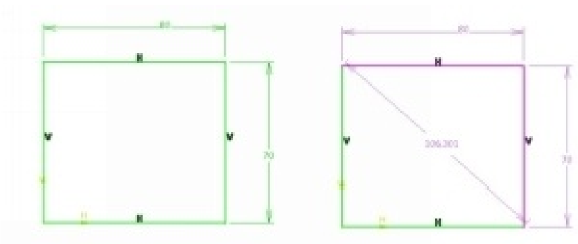



If you want to create an angle dimension between two elements, then activate the **Constraint** command and select the elements. Next, move the pointer and position the dimension.

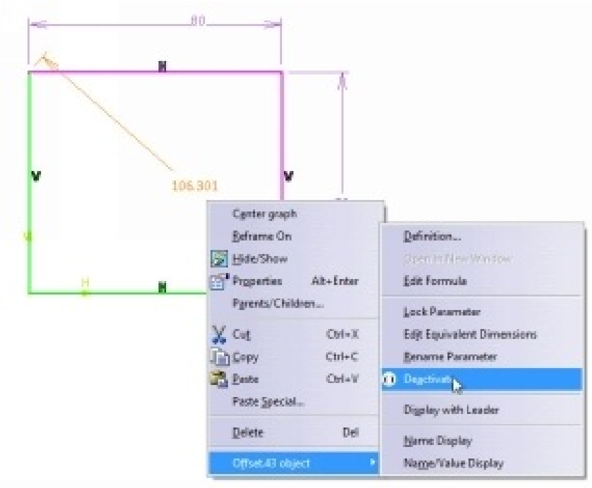



# Over-constrained Sketch

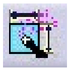
When creating sketches for a part, CATIA V5 will not allow you to over-constrain the geometry. The term 'over-constrain' means adding more dimensions than required. The following figure shows a fully constrained sketch. If you add another dimension to this sketch (e.g. diagonal dimension), the elements and dimensions affected by the additional dimension will turn into magenta color.



Now, you have to deactivate one of the dimensions. Click the right mouse button on the diagonal dimension and select **object > Deactivate** to deactivate the dimension. The deactivated dimension will be in black color.

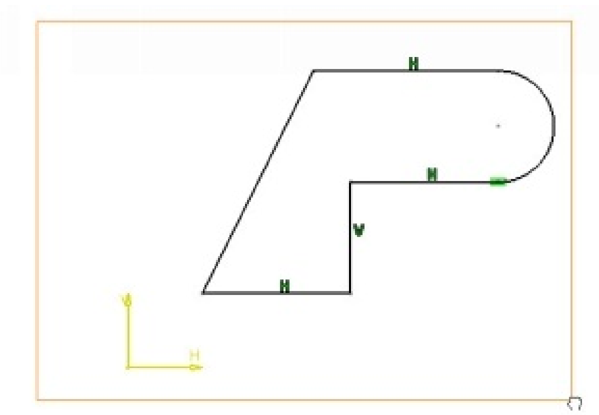



Now, if you change the value of the width, the inactive dimension along the diagonal updates, automatically. Also, note that the dimensions, which are initially created, will be driving dimensions, whereas the dimensions created after fully defining the sketch are over constraining dimensions.

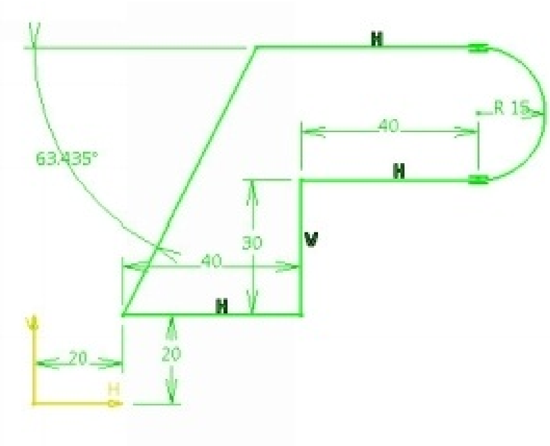
##  Auto Constraint

This command automatically creates dimensions and fully constrains the sketch.

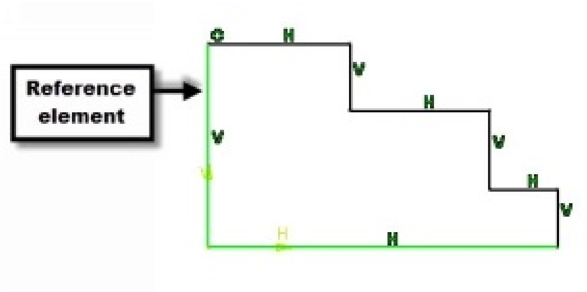
1. On the **Constraint** toolbar, click **Fix together > Auto Constraint**.
2. Press the left mouse button and drag a selection box around the sketch.

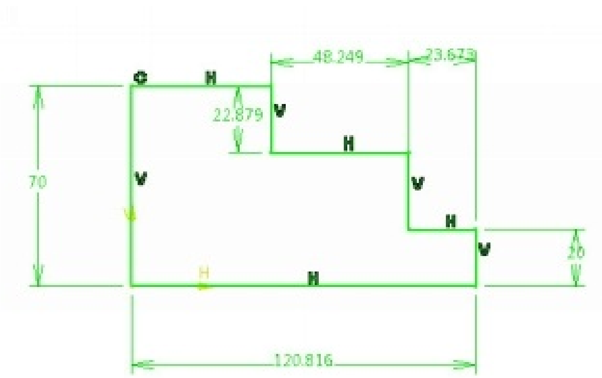


3. Click **OK** on the **Auto Constraint** dialog.

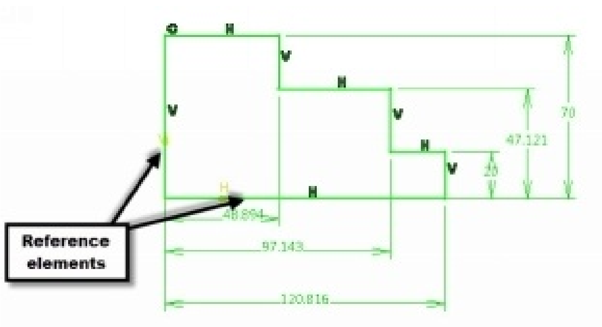


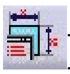

If you want to create chained dimensions, then select the complete sketch and click in the **Reference elements** selection box. Click on the longest element of the sketch to define the reference element. On the **Auto Constraint** dialog, select **Constraint mode > Chained** to create chained dimensions.




Likewise, use **Constraint Mode > Stacked** to create stacked dimensions.

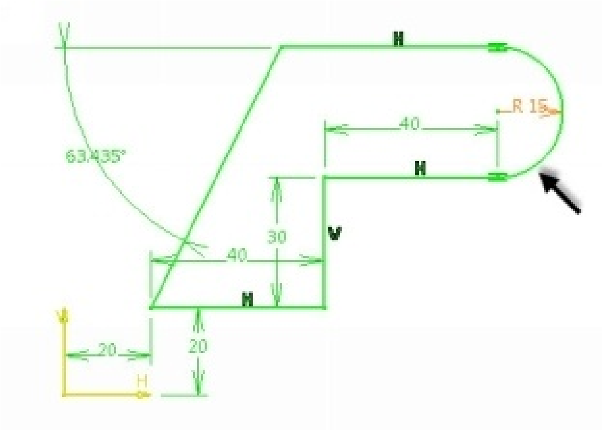


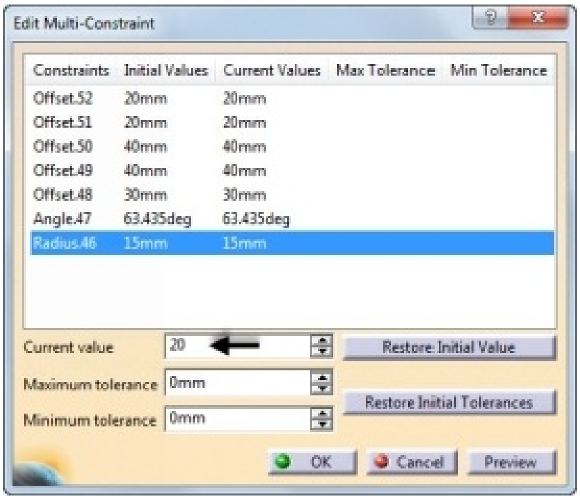



# Edit Multi-Constraint

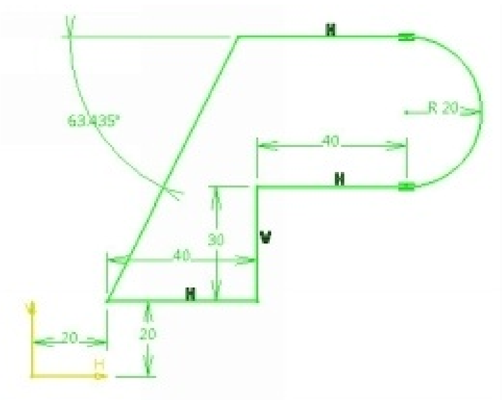
This command modifies all the constraints in a sketch using the **Edit Multi-Constraint** dialog.

1. On the **Constraint** toolbar, click the **Edit Multi-Constraint** icon.
2. On **Edit Multi-Constraint** dialog, select the dimensions and type-in a value in the **Current Value** box. For example, to change the radius value of the arc shown in figure, select the **Radius** dimension and type-in a new value in the **Current value** box.




3. If required, you can type-in the maximum and minimum tolerance values.
4. Click **OK** to update the dimension.

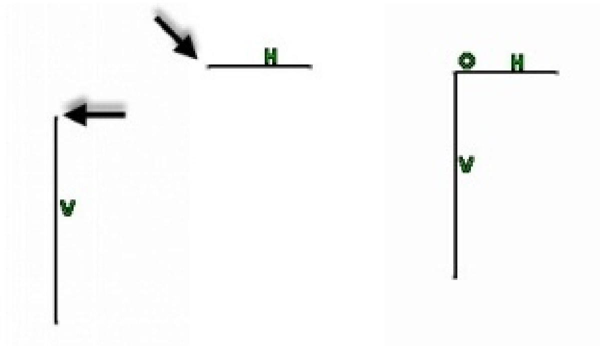



## Contact Constraint

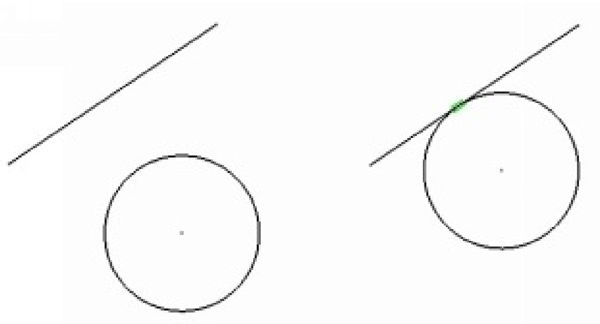
This command establishes contact between the sketch elements based on the selection.

1. On the **Constraint** toolbar, click Constraint > **Contact Constraint**.

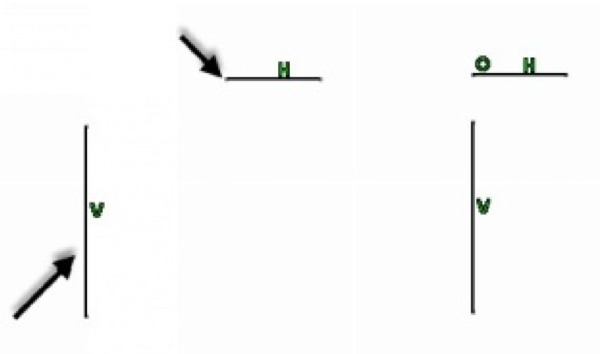
2. Select two points to make them coincident.



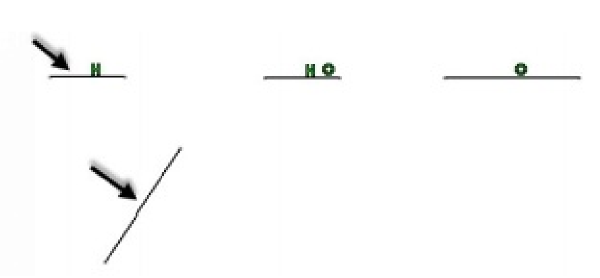

3. Select a curve and line to make them tangent to each other.



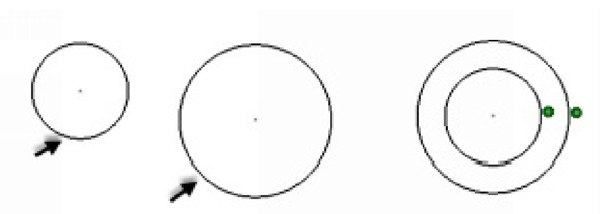
4. Select a line or curve and point to make them coincident.

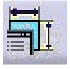



5. Select two lines to make them collinear.



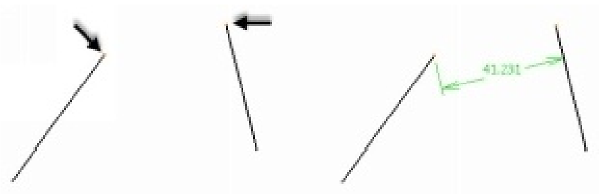

6. Select two circles or arcs to make them concentric.



#  Constraints Defined in Dialog

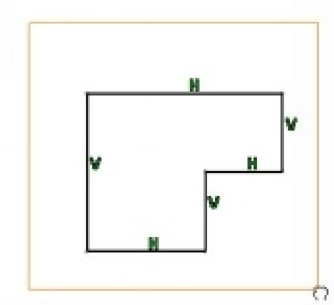
In addition to dimensional and contact constraints, there are other constraints, which you can establish between the sketch elements. You can do this using the **Constraints Defined in Dialog** command.

1. Press and hold the Ctrl key and click on two points.
2. On the **Constraint** toolbar, click the **Constraints Defined in Dialog** icon.
3. On the **Constraint Definition** dialog, check the **Distance** option to establish a distance constraint between two points.

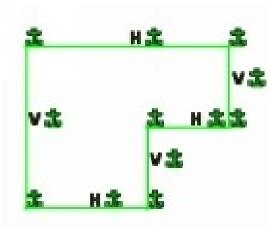


4. If you want to make the two points coincident with each other, then uncheck the **Distance** option.
5. Check the **Coincidence** option.

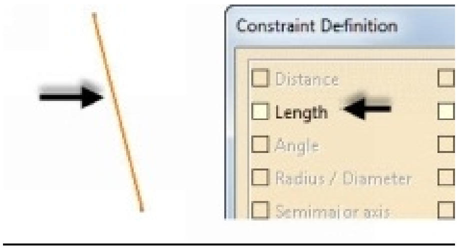
To fix a sketch element (or elements) at its current location, select it and activate the **Constraints Defined in Dialog** command.



On the **Constraint Definition** dialog, check the **Fix** option.

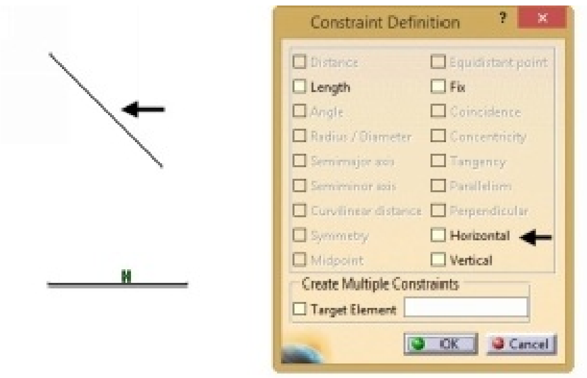


To apply the **Length** constraint, select a linear element and activate the **Constraint Defined in Dialog** command. Check the **Length** option on the **Constraint Definition** dialog.

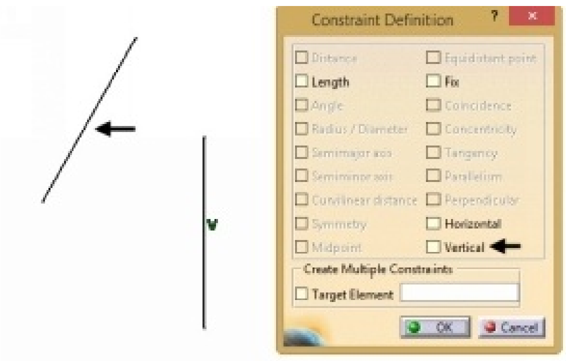




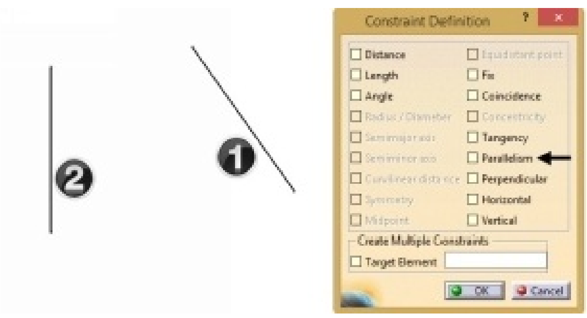
To apply the **Horizontal** constraint, select a linear element and activate **Constraint Defined in Dialog** command. Check the **Horizontal** option.

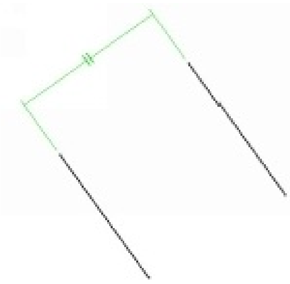



Likewise, use the **Vertical** option to make a line vertical.

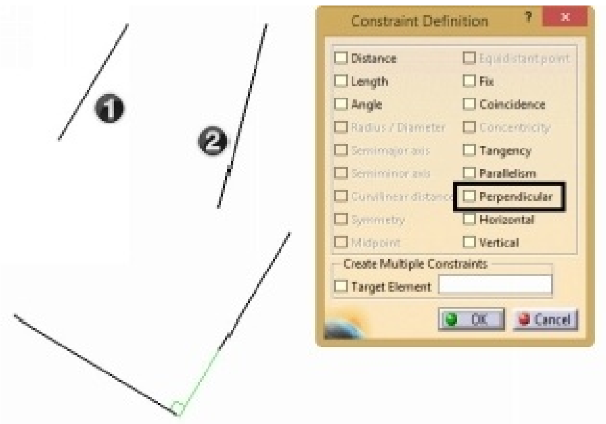



Use the **Parallelism** option to make two lines parallel to each other.

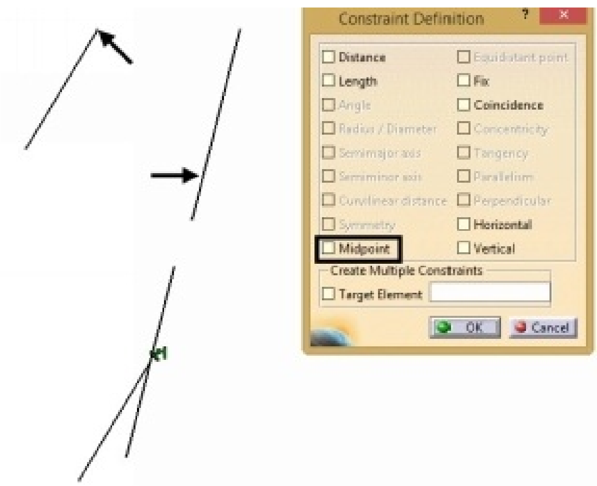




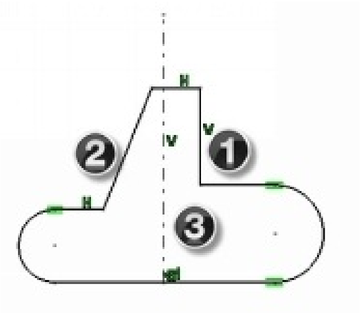
Use the **Perpendicular** option to make two lines perpendicular to each other.



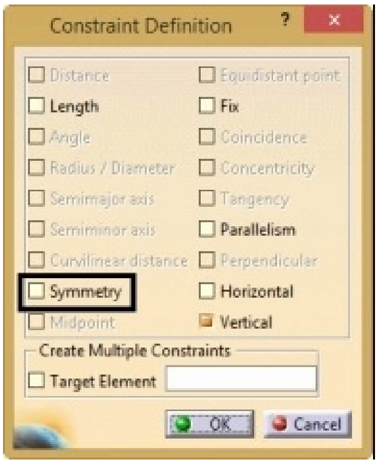

Use the **Midpoint** option to make a point coincide with the midpoint of a line.

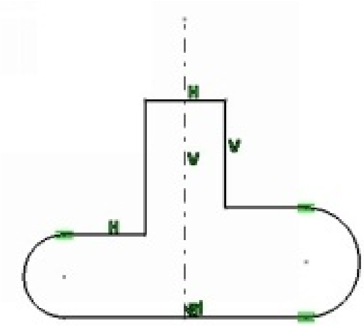



Use the **Symmetry** option to make two sketch elements symmetric about a centerline. Press the Ctrl key and click on the elements to make symmetric. Click on the symmetric line and activate the **Constraint Defined in Dialog** command.




On the **Constraint Definition** dialog, check the **Symmetry** option and click **OK**.



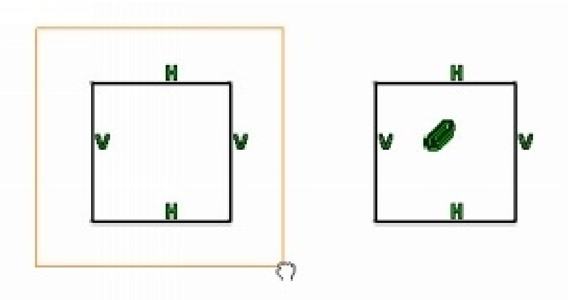



You can also create multiple constraints between two elements. To do so, check the **Target Element** option, click in **Target Element** selection box, and select the target element. Select the constraints from the **Constraint Definition** dialog.

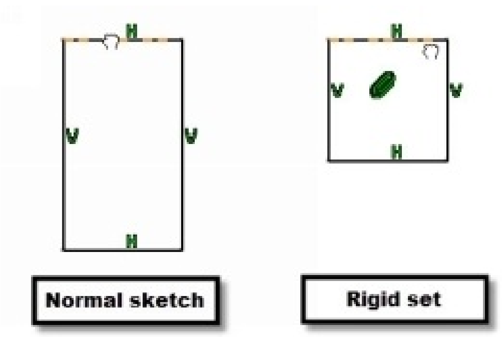
## The Fix Together command

This relation makes the selected elements act as a single unit.

1. On the **Constraint** toolbar, click the **Fix Together** icon.
2. Select two or more elements from the sketch.
3. Click **OK** on the **Fix Together Definition** dialog. The selected objects will form a rigid set.

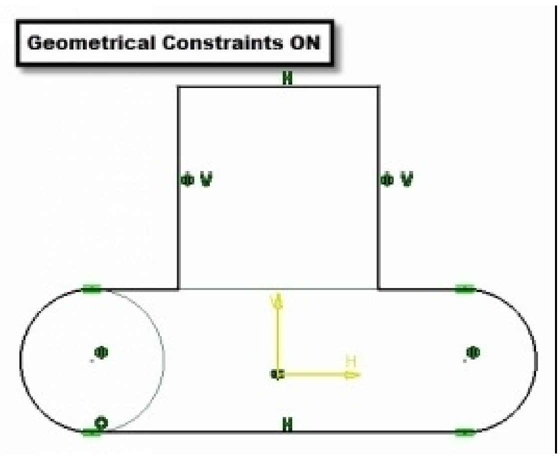


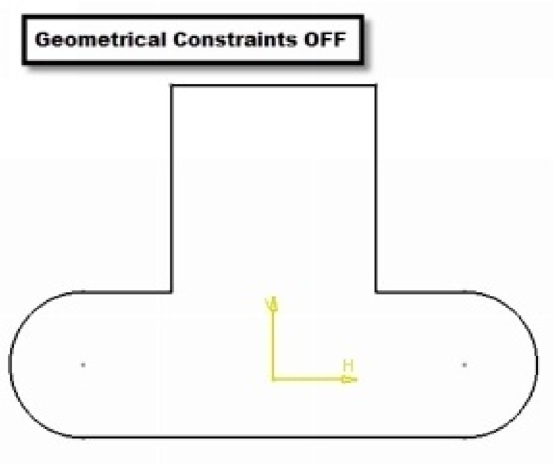
Now, click and drag anyone of the object from the rigid set. You will notice that entire set will be dragged.




##  Display Geometrical Constraints

As constraints are created, they can be shown or hidden using the **Geometrical Constraints** icon on the **Visualization** toolbar. When dealing with complicated sketches involving numerous constraints, you can deactivate this button to turn off the display of all geometrical constraints.



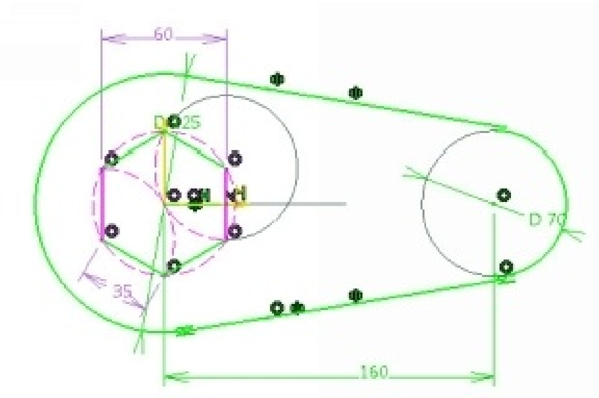



##  Sketch Solving Status

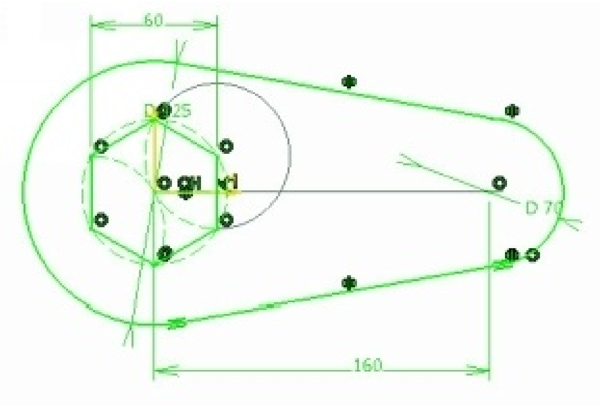
At any stage of the design process, you can check whether the sketch is fully constrained or not by viewing the sketch color. However, you can also use the **Sketch Solving Status** command to check the status of the sketch. Activate this command (On the **Tools** toolbar, click the **Sketch Solving Status** icon) to view the sketch status.

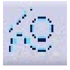
##  Sketch Analysis

This command can be used to analyse the sketch. For example, the following figure shows an over-constrained sketch.



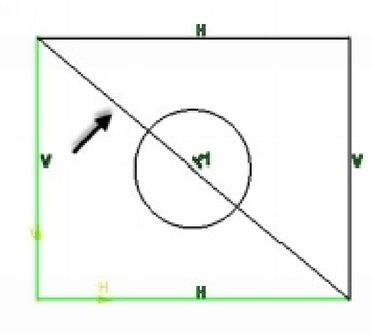

1. To analyse this sketch, activate the **Sketch Analysis** command (on the **Tools** toolbar, click **2D Analysis** drop-down > **Sketch Analysis**).
2. On the **Sketch Analysis** dialog, click the **Diagnostics** tab.
3. Under the **Solving Status** section, select the constraint marked as Over-constrained.
4. Under the **Action** section, click the **Delete geometry or constraint**  button.
5. Close the dialog.

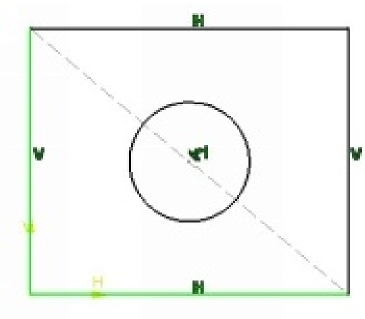


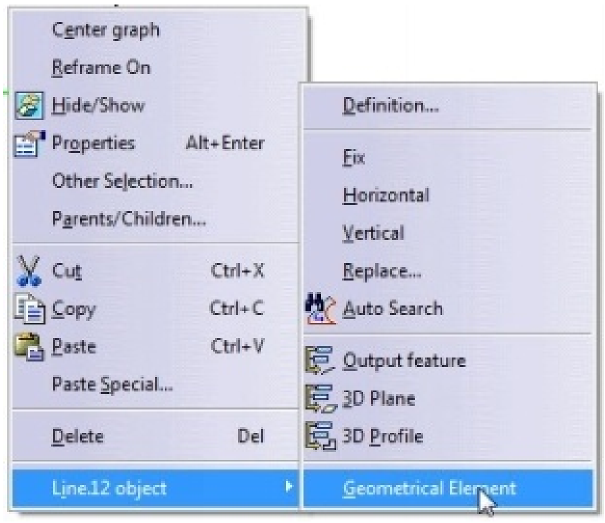
## Construction/Standard Element

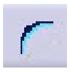
This command converts a standard sketch element into a construction element. Construction elements support you to create a sketch of a desired shape and size. To convert a standard sketch element to construction element, click on it and select **Construction/Standard Element** on the **Sketch tools** toolbar.





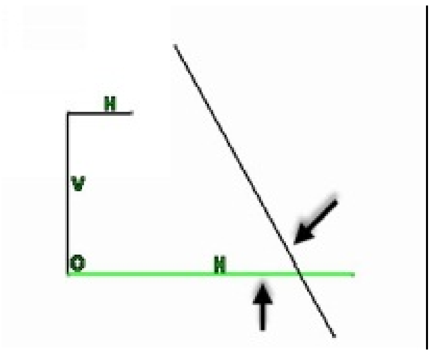
You can also convert it back to a standard sketch element by right clicking on it and selecting **Geometrical Element**.



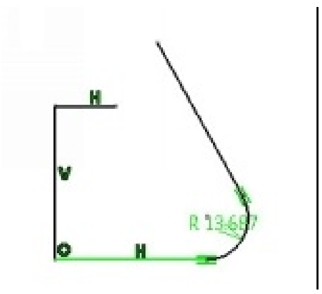

##  The Corner command

This command rounds a sharp corner created by intersection of two lines, arcs, circles, and rectangle or polygon vertices.

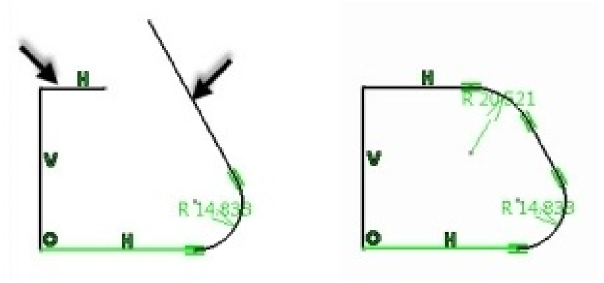
1. On the **Operation** toolbar, click the **Corner** icon (or) click **Insert > Operation > Corner > Corner**.
2. Select the intersecting elements to add a corner.



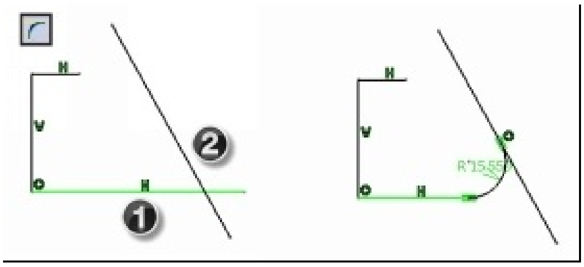
3. Type-in a radius value in the **Radius** box available on the **Sketch tools** toolbar.
4. Press Enter.



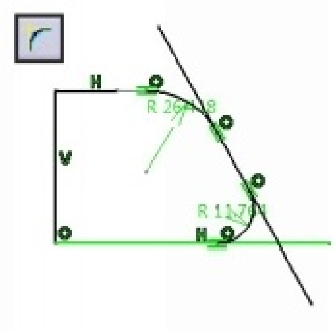
The elements to be cornered are not required to touch each other.



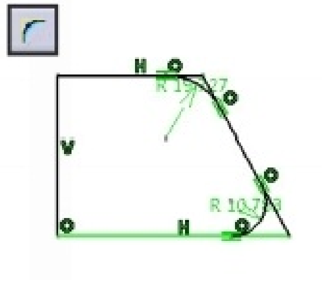
By default, the elements are automatically trimmed or extended to meet the end of the new corner radius. You can use the **Trim first Element** option on the **Sketch tools** toolbar, if you want to trim only the first element.

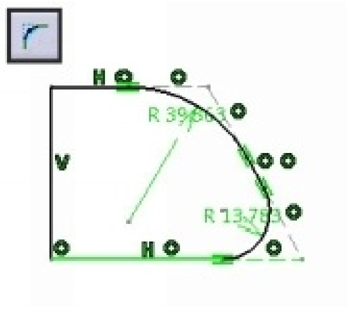


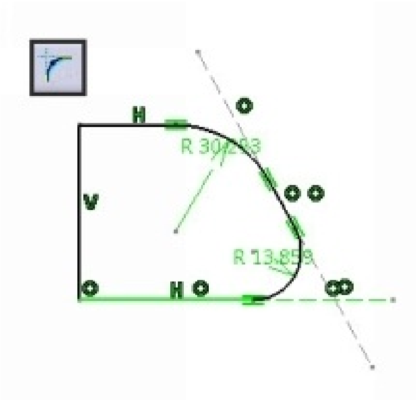
Use the **No Trim** option on the **Sketch tools** toolbar, if you do not want to trim or extend the elements as necessary.

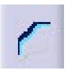


The other trim options are:



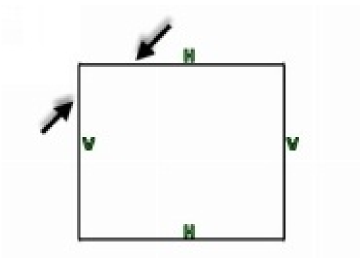




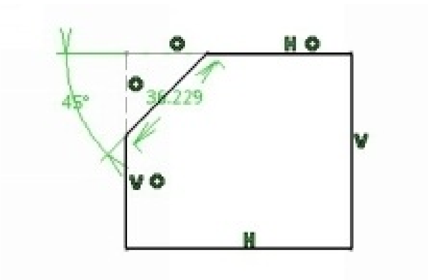
##  The Chamfer command

This command replaces a sharp corner with an angled line.

1. On the **Operations** toolbar, click the **Chamfer** icon.
2. Select the select the elements' ends to be chamfered.



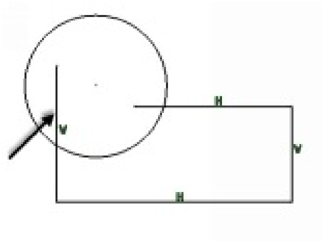
3. Type-in the chamfer angle and length in the **Angle** and **Length** boxes on the **Sketch Tools** toolbar, respectively.
4. Press Enter to create the chamfer.

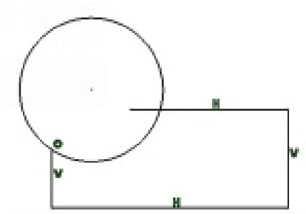


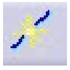

##  The Quick Trim command

This command trims the end of an element back to the intersection of another element.

1. On the **Operations** toolbar, click **Relimitations** drop-down > **Quick Trim**.
2. Click on the element to trim.





##  The Break command

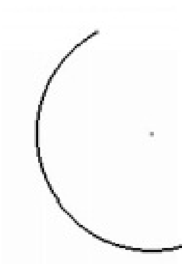
This command breaks a sketch element at a selected point.

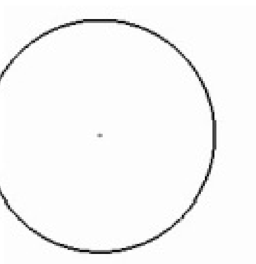
1. On the **Operations** toolbar, click **Relimitations** drop-down > **Break**.
2. Select the element to break.
3. Click to define the break point.

##  The Close Arc command

This command closes the open arc.

1. On the **Operations** toolbar, click **Relimitations** drop-down > **Close Arc**.
2. Click on an arc to convert it into a circle.

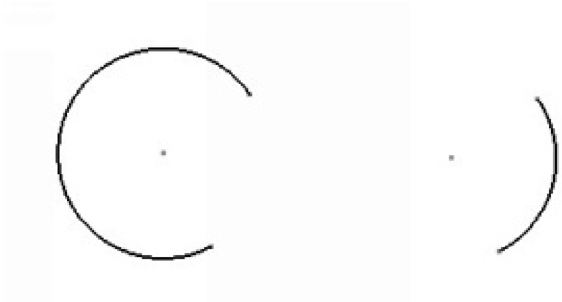




##  The Complement command

This command shows the complementary side of an arc.

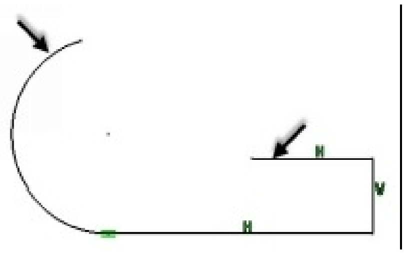
1. On the **Operations** toolbar, click **Relimitations** drop-down > **Complement**.
2. Click on an arc to show the complementary side of it.

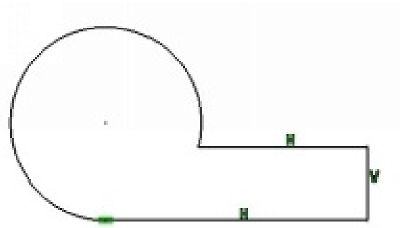


##  The Trim command

This command trims and extends elements to form a corner.

1. On the **Operations** toolbar, click **Relimitations** drop-down > **Trim**.
2. Select two intersecting elements. The elements will be trimmed and extended to form a closed corner.

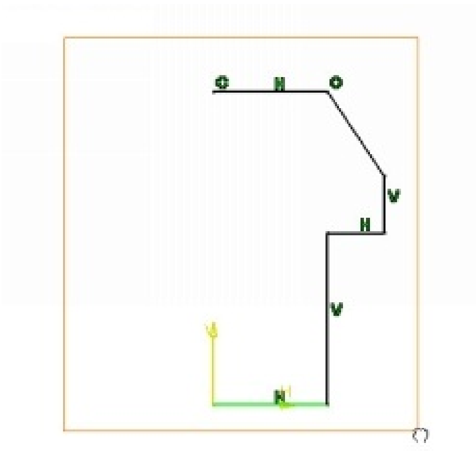




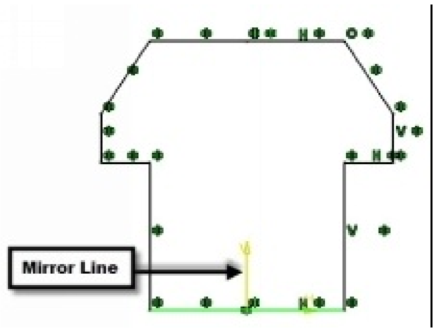
##  The Mirror command

This command creates a mirror copy of the selected sketch elements.

1. On the **Operations** toolbar, click **Transformation** drop-down > **Mirror**.
2. Drag a selection box and select the elements to mirror.



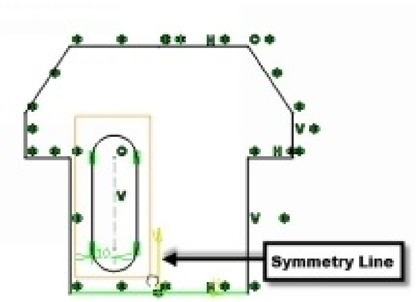
3. Click on a line or axis to define the mirror line.

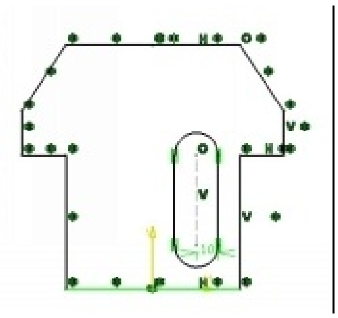


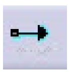

##  The Symmetry command

This command creates a mirror image of selected sketch elements without copying them.

1. On the **Operations** toolbar, click **Transformation** drop-down > **Symmetry**.
2. Click on the element to mirror (or) drag a selection box to select multiple elements at a time.
3. Click on the line or axis about which the element will be mirrored.

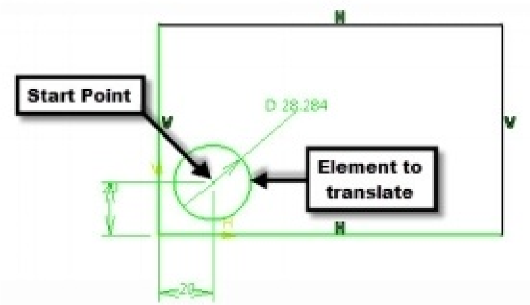


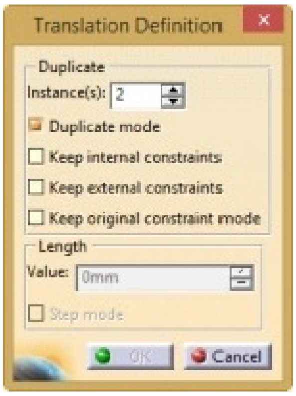
##  The Translate command

This command relocates one or more elements from one position in the sketch to any other position you specify.

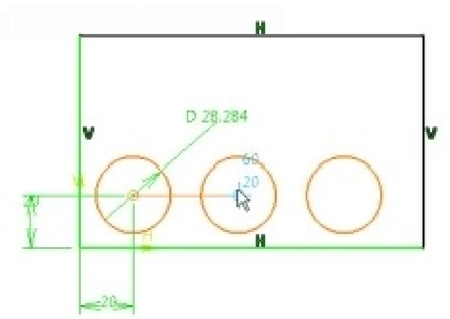
1. On the **Operations** toolbar, click **Transformation** drop-down > **Translate**.
2. Click on the elements to translate.
3. Click to define the start point of the translation.

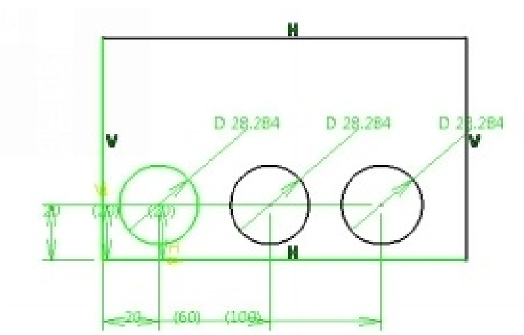



4. On the **Translation Definition** dialog, check the **Duplicate mode** if you want to copy and move the selected element(s). Next, type-in the number of instances to be created in the **Instance** box.




5. On the **Translation Definition** dialog, check the **Keep internal constraints** and **Keep external constraints** options to copy the constraints of the selected element as well. Check the **Keep original constraint mode** option, if you want to copy the element with it original constraints.
6. Move the pointer and click to define the translation distance (or) type-in a value in the **Length** box on the **Translation Definition** dialog.

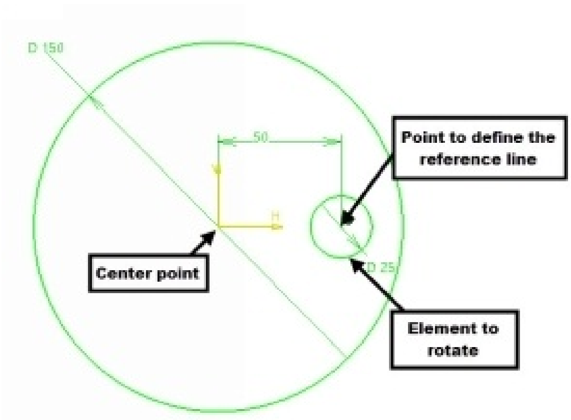


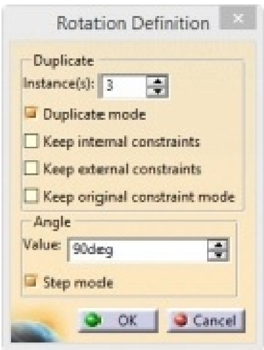
## The Rotate command

This command rotates the selected elements to any position.

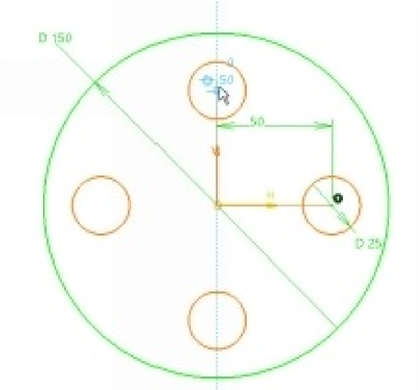
1. On the **Operations** toolbar, click **Transformation** drop-down > **Rotate**.
2. Select the elements to rotate.
3. Click to define the center point of the rotation.
4. Move the cursor and click to define a reference line for rotation angle.

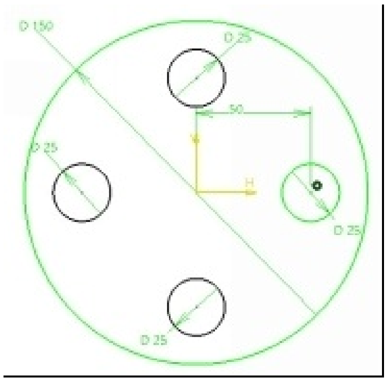



5. On the **Rotation Definition** dialog, check the **Duplicate mode** if you want to copy and rotate the selected element(s). Next, type-in the number of instances to be created in the **Instance** box.




6. On the **Rotation Definition** dialog, check the **Keep constraints** option to copy the constraints of the selected element as well.
7. Move the pointer and click to define the rotation angle (or) type-in a value in the **Value** box on the **Rotation Definition** dialog.

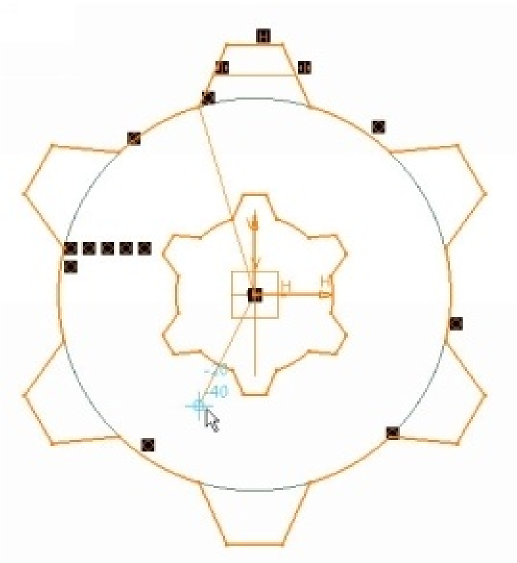



## The Scale command

This command increases or decreases the size of elements in a sketch.

1. On the **Operations** toolbar, click **Transformation** drop-down > **Scale**.
2. Select the elements to scale.
3. Select a base point.
4. Scale the size of the selected elements by moving the pointer or typing-in a scale value in the **Value** box on the **Scale Definition** dialog.

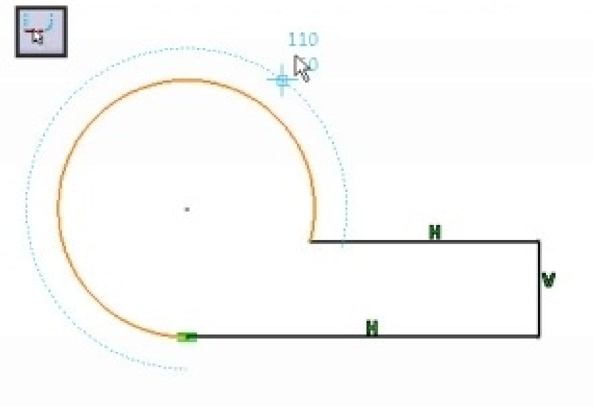


## The Offset Curve command

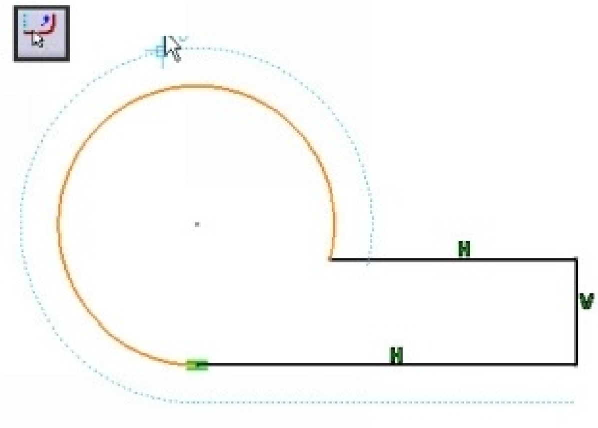
This command creates a parallel copy of a selected element or chain of elements.

1. On the **Operations** toolbar, click **Transformation** drop-down > **Offset**.
2. Click on the sketch element to offset.

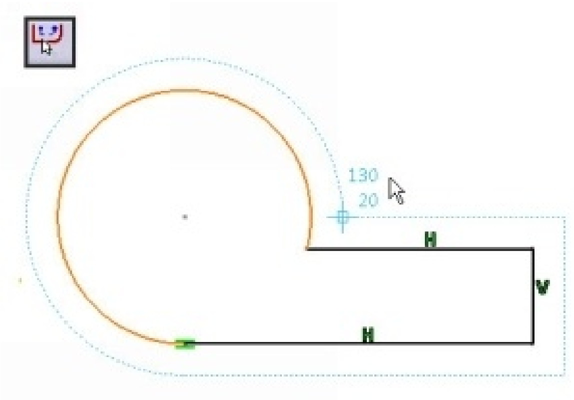
Use the **No Propagation** option on the **Sketch tools** toolbar to select a single element.



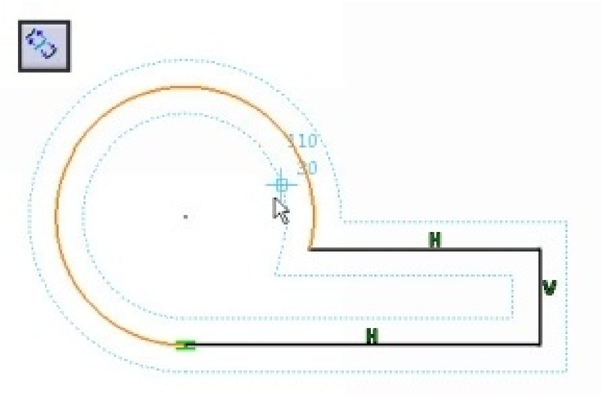

Use the **Tangent Propagation** option to select tangentially connected elements.



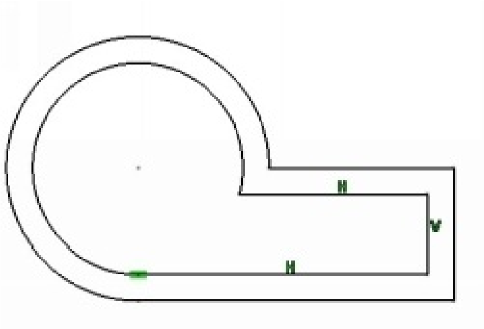

Use the **Point Propagation** option to select all the connected elements in a single click.




Use the **Both Sides** offset option to offset the sketch elements on both sides.



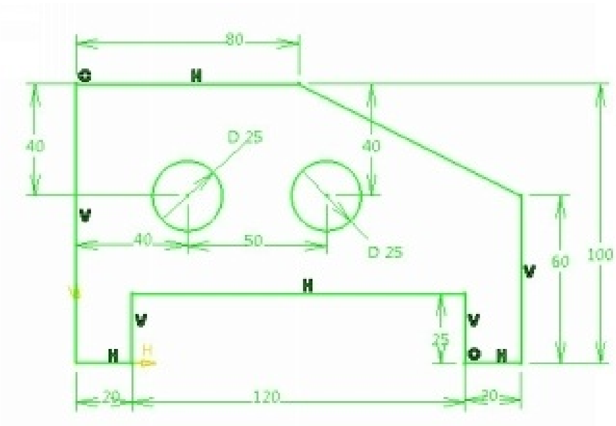

3. Type-in a value in the **Offset** box available on the **Sketch tools** toolbar.
4. If you want to create more than one offset copy, then type-in a value in the **Instance** box available on the **Sketch tools** toolbar.
5. Press Enter to create the offset copy.



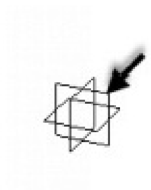
# Examples

## Example 1

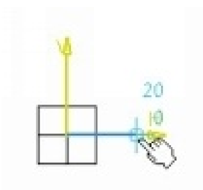
In this example, you will draw the sketch shown below.



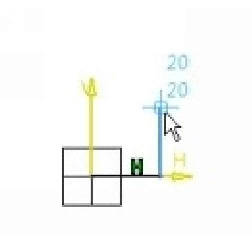

1. Start **CATIA V5-6R2015** by clicking the **CATIA V5-6R2015** icon on your desktop.
2. On the **Standard** toolbar, click the **New** icon.
3. On the **New** dialog, click **List of Types > Part** and click **OK**.
4. Click **OK** on the **New Part** dialog.
5. Click **Sketch** icon on the **Sketcher toolbar** (or) **Insert > Sketcher > Sketch** on the Menu.
6. Click on the YZ plane to start the sketch.



7. On the **Sketch tools** toolbar, deactivate the **Snap to Point** icon
8. On the **Profile** toolbar, click the **Profile** icon.
9. Click on the origin point to define the first point of the line.
10. Move the pointer rightwards and click.

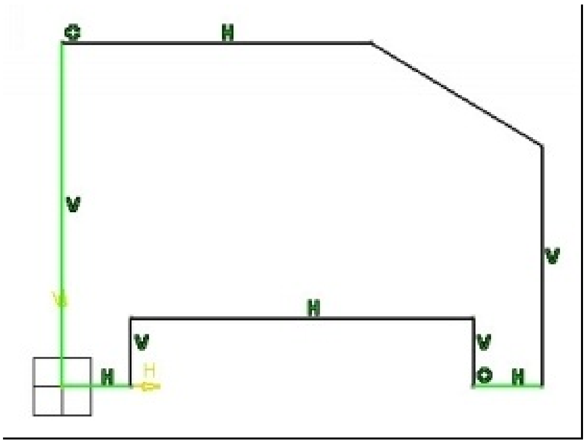


11. Move the pointer upwards and click.

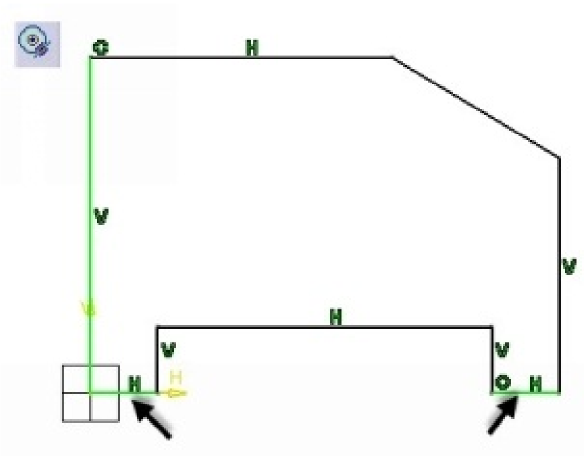


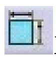
12. Move the pointer rightwards and click.

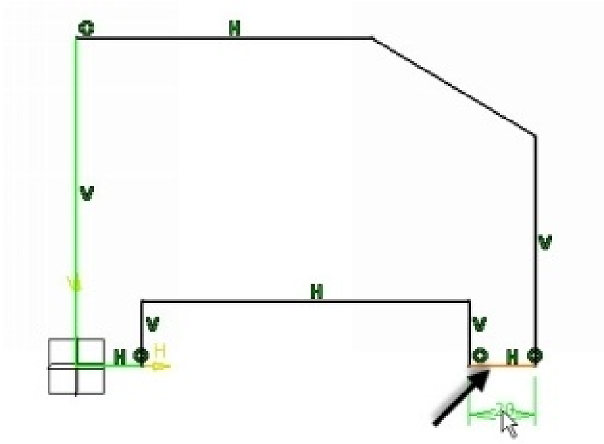

13. Create a closed loop by selecting points, as shown below.



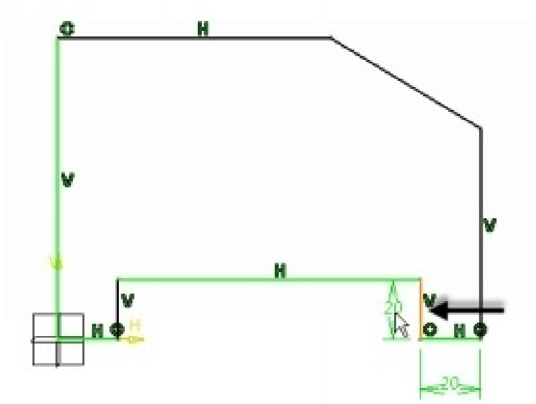

14. On the **Constraints** toolbar, click the down arrow next to the **Constraints** icon and select the **Contact Constraint** icon (or) **Insert > Constraint > Constraint Creation > Contact Constraint** on the Menu.
15. Click on the two horizontal lines at the bottom; they become collinear.



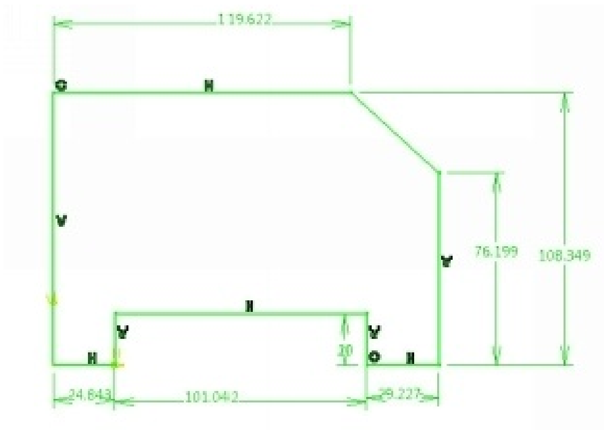

16. On the **Constraint** toolbar, double-click the **Constraint**  icon and click on the lower horizontal line.
17. Move the pointer downward and click to position the dimension.



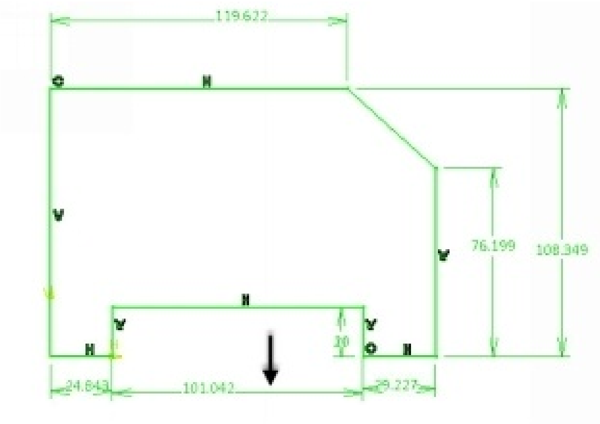

18. Click on the small vertical line.
19. Move the pointer and click to position the dimension.



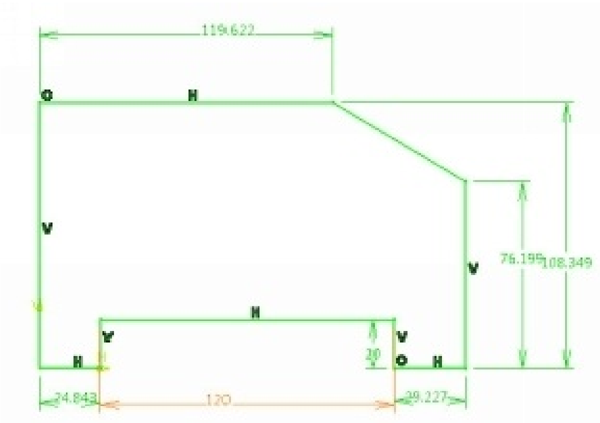

20. Likewise, create other dimensions, as shown below.



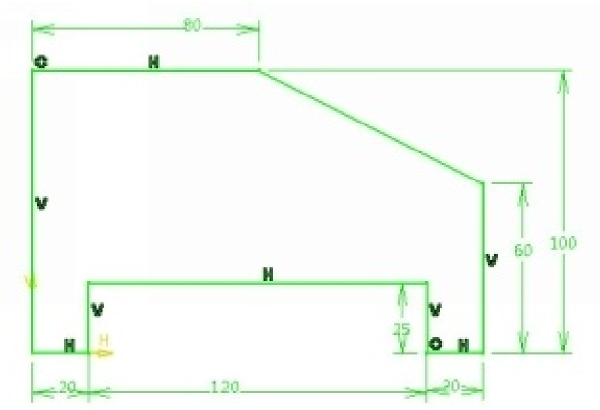

21. On the **Constraints** toolbar, click the **Edit Multi-Constraint** icon.
22. Click on the horizontal dimension, as shown in figure.

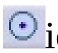


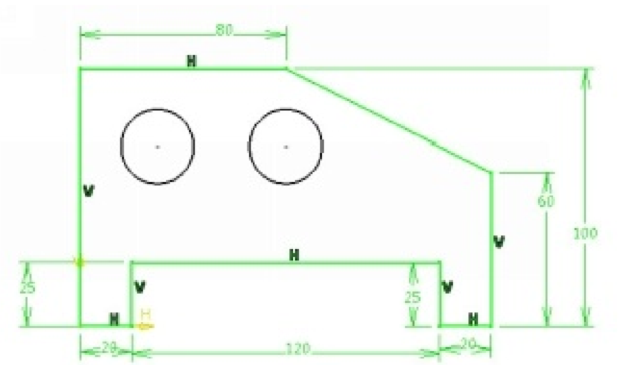

23. Type **120** in the **Current Value** box and click the **Preview** button.



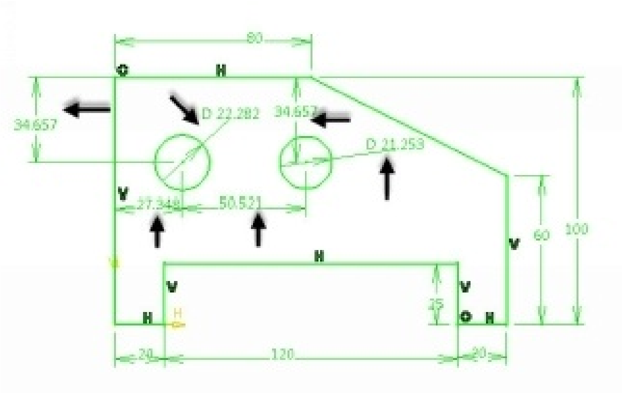

24. Likewise, change the other dimensional values. Click **OK** on the dialog.



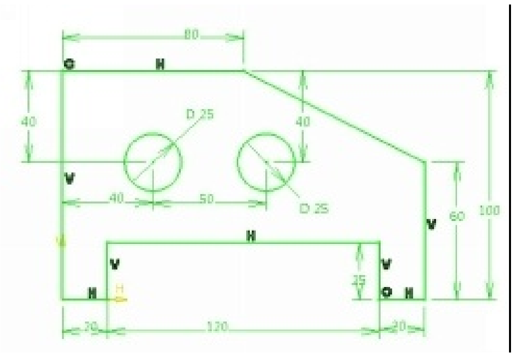

25. On the **Profile** toolbar, click **Circle**  icon.
26. Click inside the sketch region to define the center point of the circle. Move the pointer and click to define the diameter. Likewise, create another circle.

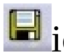


27. On the **Constraints** toolbar, click the **Constraints** icon and apply dimensions to fully constraint the circles.



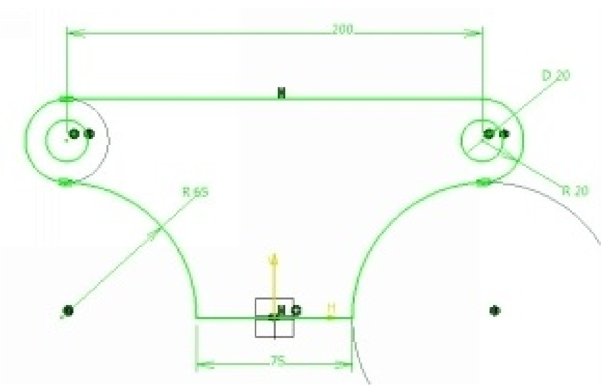

28. Activate the **Edit Multi-Constraint** command and modify the dimension values of the circles.



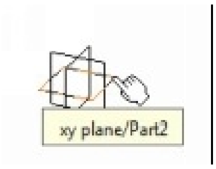

29. On the **Workbench** toolbar, click **Exit Workbench** .
30. On the **Standard** toolbar, click the **Save** icon (or) click **File > Save** on the Menu.
31. On the **Save As** dialog, type-in **C2_example1** in the **File name** box. Define the location and click **Save** to save the part file.
32. Click **Close Window** on the top right corner to close the part file.


## Example 2

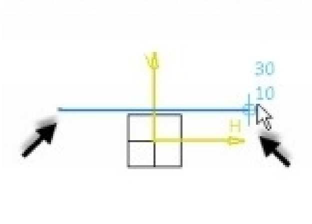
In this example, you will draw the sketch shown below.



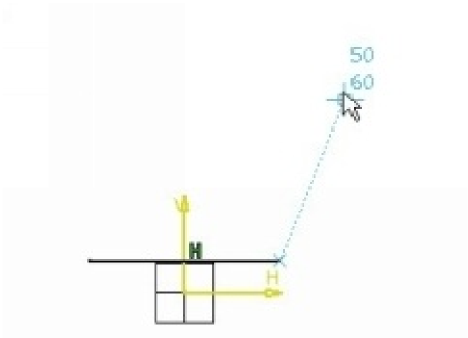
1. Start **CATIA V5-6R2015** by clicking the **CATIA V5-6R2015** icon on your desktop.
2. On the **Standard** toolbar, click the **New** icon.
3. On the **New** dialog, click **List of Types > Part** and click **OK**.
4. Click **OK** on the **New Part** dialog.
5. To start a new sketch, click the **Sketch** icon on the **Sketcher** toolbar.
6. Click on the XY Plane to start the sketch.



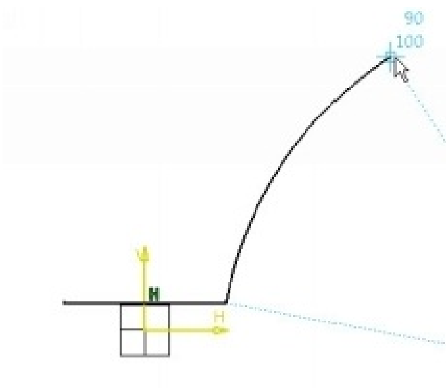

7. On the **Profile** toolbar, click the **Profile** icon.
8. Click in the second quadrant of the coordinate system to define the start point of the profile. Drag the pointer horizontally and click to define the endpoint.



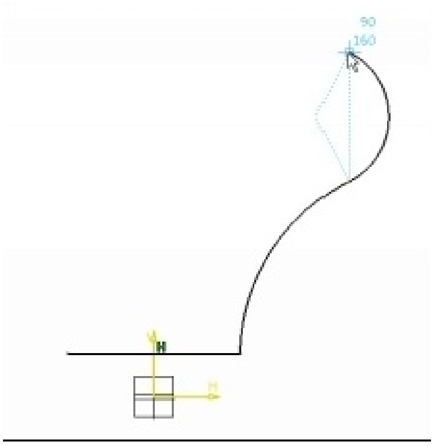

9. On the **Sketch tools** toolbar, click the **Three Point Arc**  icon.
10. Move the pointer upwards right and click to define the second point of the arc.



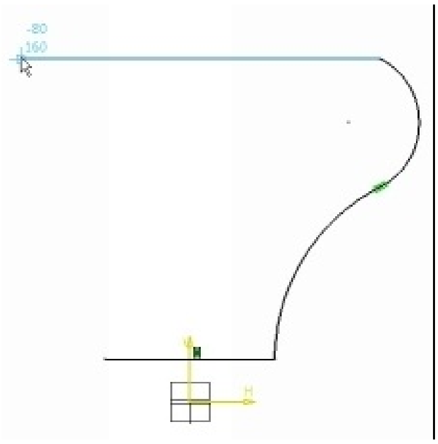

11. Move the pointer and click to define the third point of the arc, as shown.



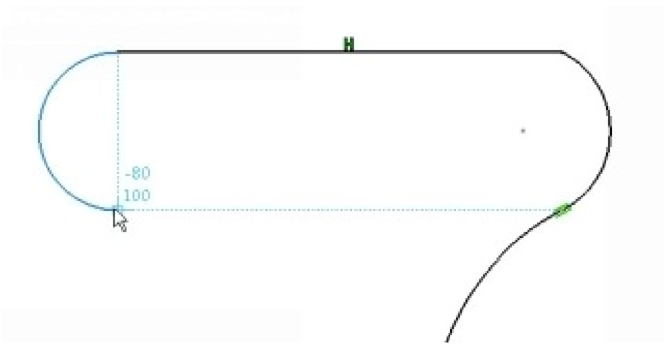

12. On the **Sketch tools** toolbar, click the **Tangent Arc**  icon.
13. Move the pointer upwards and click to create an arc tangent to the previous arc.



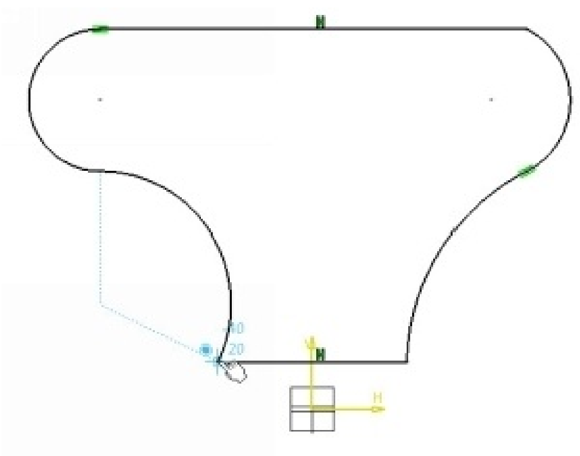
4. Move the pointer toward left and click to create a horizontal line.



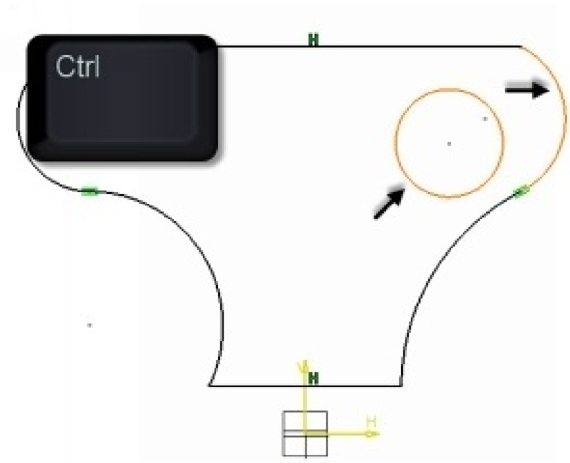
5. Click the **Tangent Arc** icon the **Sketch tools** toolbar.
6. Move the pointer downwards and click when a vertical dotted line appears, as shown below.



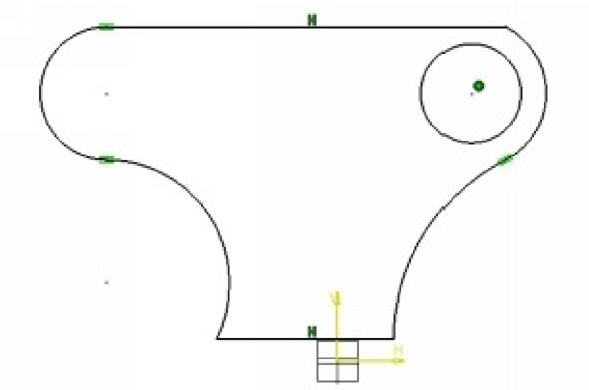

7. Click the **Tangent Arc** icon on the **Sketch tools** toolbar. Move the pointer downward right and click on the origin to close the sketch.



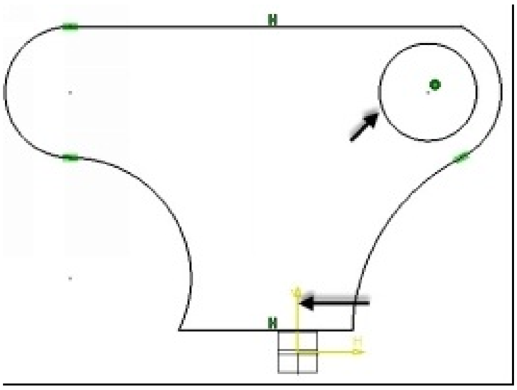
8. Press Esc to deactivate the **Profile** command.
9. Activate the **Circle** command and draw a circle on the right side.
0. Press the Ctrl key and click on the circle and the small arc.

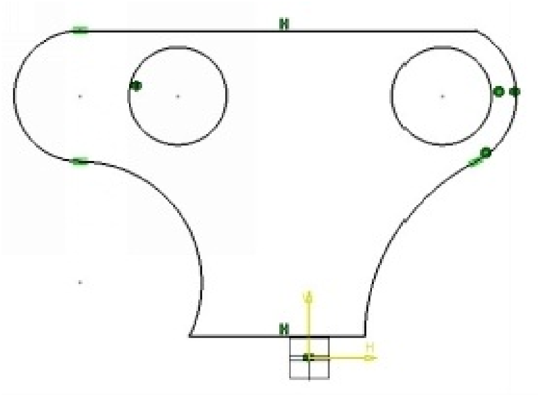



1. On the **Constraints** toolbar, click the **Constraints Defined in Dialog** icon.
2. On the **Constraints Definition** dialog, check the **Concentricity** option and click **OK**. The circle and arc are made concentric.

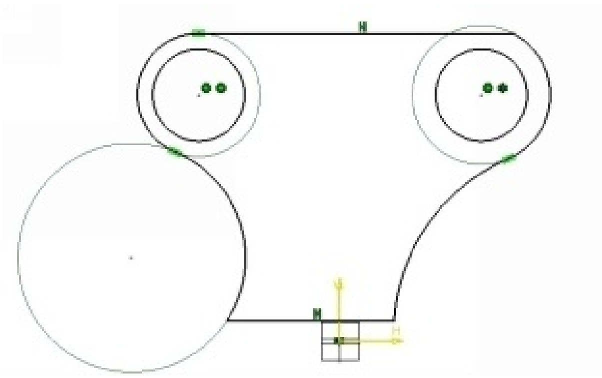


3. On the **Operation** toolbar, click the **Mirror**  icon (or) click **Insert > Operation > Transformation > Mirror**.
4. Click on the small circle and the vertical axis. A mirror copy of the circle is created.

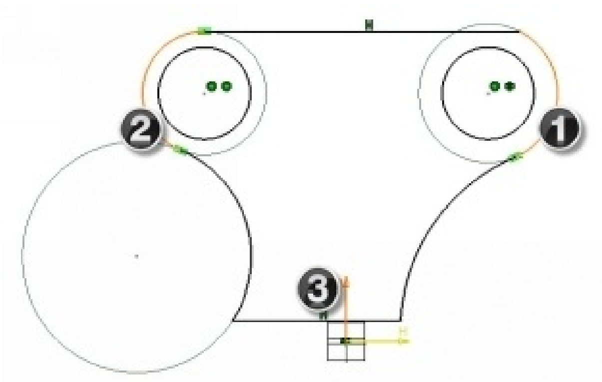




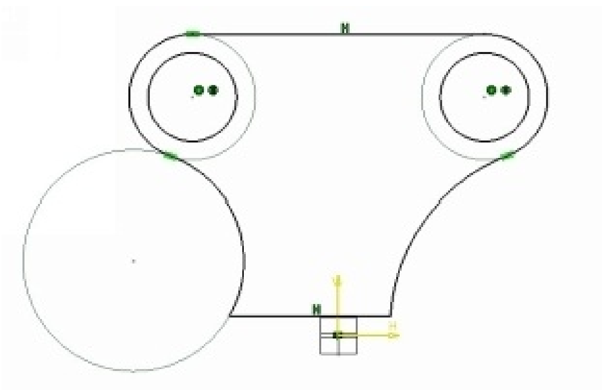
5. Apply the **Concentricity** constraint between the new circle and arc on left side.



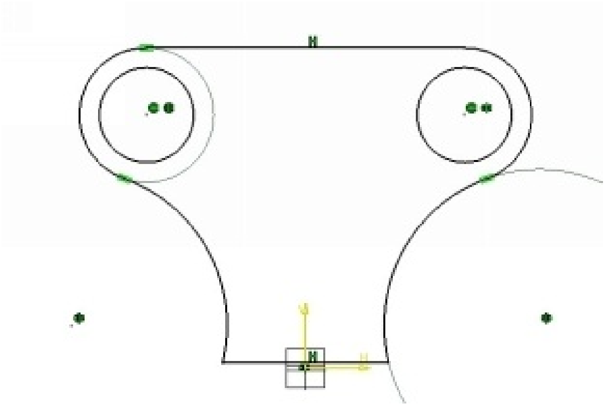
6. Press the Ctrl key and select the small arcs, and then click on the vertical axis.



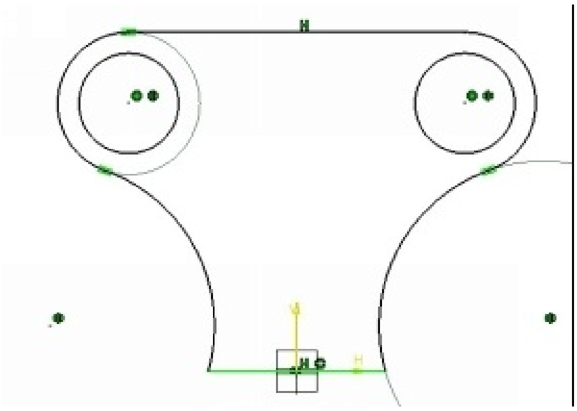

7. On the **Constraints** toolbar, click the **Constraints Defined in Dialog** icon.
8. On the **Constraints Definition** dialog, check the **Symmetry** option and click **OK**. The two arcs are made symmetric about the vertical axis.



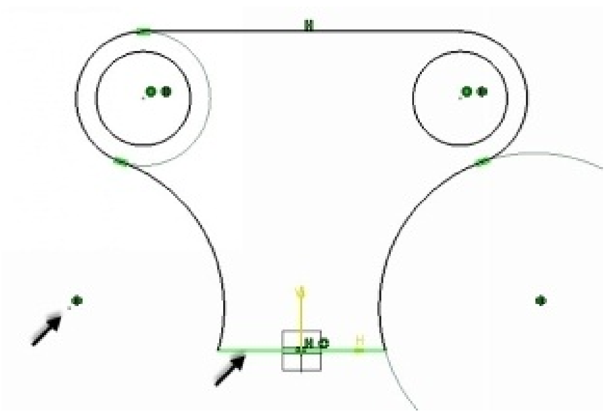
9. Likewise, make the large arcs symmetrical about the vertical axis.



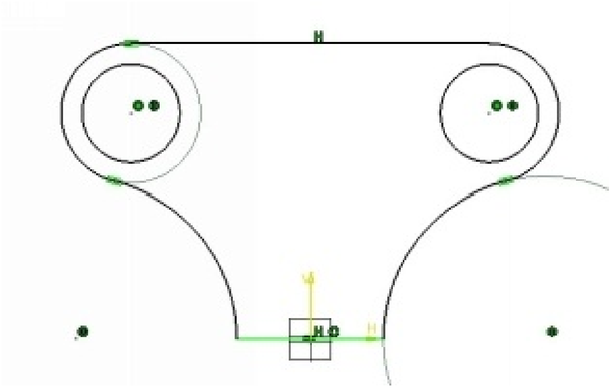
0. Press the Ctrl key and click the bottom horizontal line and the horizontal axis.
1. Activate the **Constraints Defined in Dialog** command and check the **Coincidence** option on the **Constraint Definition** dialog.
2. Click **OK** to make the horizontal line coincide with the horizontal axis.



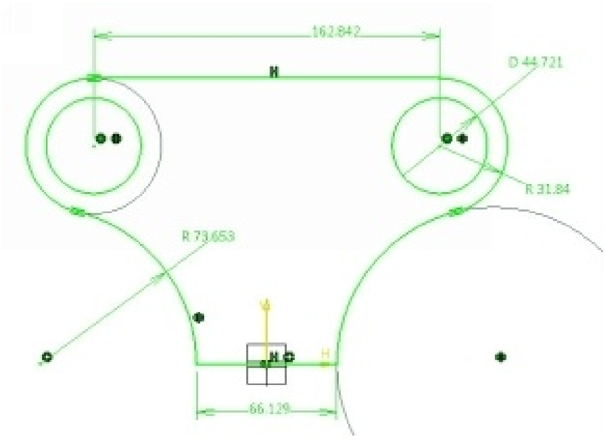
3. Press and hold the Ctrl key.
4. Click on the center point of the large arc and the horizontal line at the bottom.



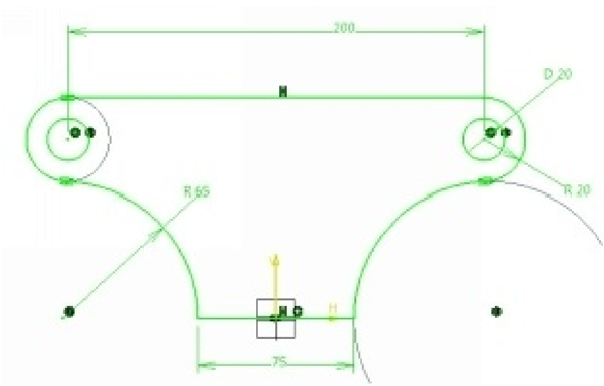
5. Activate the **Constraints Defined in Dialog** command and check the **Coincidence** option on the **Constraint Definition** dialog.
6. Click **OK** to make the center point of the large arc coincident with the horizontal line,




7. On the **Constraints** toolbar, double-click the **Constraints** icon and apply dimensions to the sketch, as shown below.




8. Activate the **Edit Multi-Constraint** command and change the dimensional values.



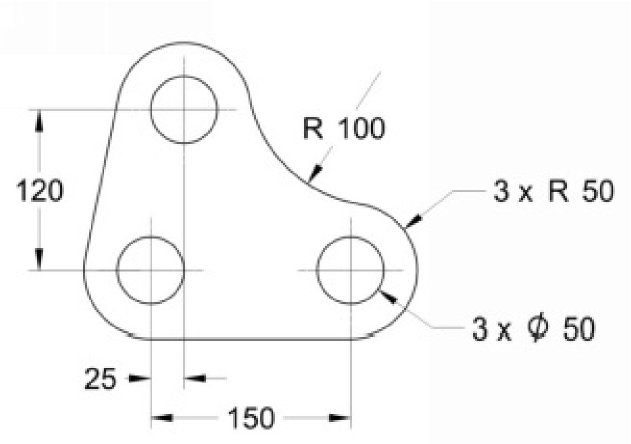

9. Click **Exit workbench** to complete the sketch.
0. To save the file, click **File > Save** on the Menu.
1. On the **Save As** dialog, type-in **C2_example2** in the **File name** box. Define the location and click **Save** to save the part file.
2. To close the file, click **File > Close** on the Menu.

# Questions

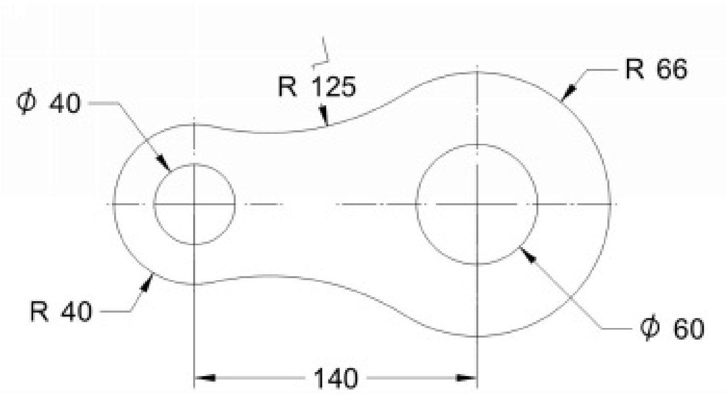
1. What is the procedure to create sketches in CATIA V5?
2. List any two sketch constraints in CATIA V5.
3. Which command creates constraints automatically?
4. Describe the method to create an ellipse.
5. How do you define the shape and size of a sketch?
6. How do you create a tangent arc using the **Profile** command?
7. Which command is used to apply different types of dimensional constraints to a sketch?
8. List any two methods to create circles.
9. How do you create fillet with an alternate solution?

# Exercises

## Exercise 1




## Exercise 2




## Exercise 3