

Getting Started with CATIA Version 5



•Let's review the following key features:

- ✍ Multi-document support
- ✍ Standard and specific menus & toolbars (File, Edit, Insert, ...)
- ✍ Standard manipulation (Copy-Paste, Drag & Drop, ...)
- ✍ Intuitive (Highlight, co-pilot, cursor shapes, ...)
- ✍ Contextual menu (**M**ouse **B**utton 3)
- ✍ Specification Tree (including all technological features, constraints, relationships, ...)
- ✍ Workbenches and associated toolbars ...

For these tutorials you will need to install a material catalogue:

Due to the Demonstration restrictions, the data you can use must be flagged. In order to use the material catalogue, you have to replace the code file by the flagged one. Please do the following:

- ✗ Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\French directory*
- ✗ Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\German directory*
- ✗ Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\Japanese directory*
- ✗ Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials directory*

- ✗ Answer Yes in order to replace the old catalogue*

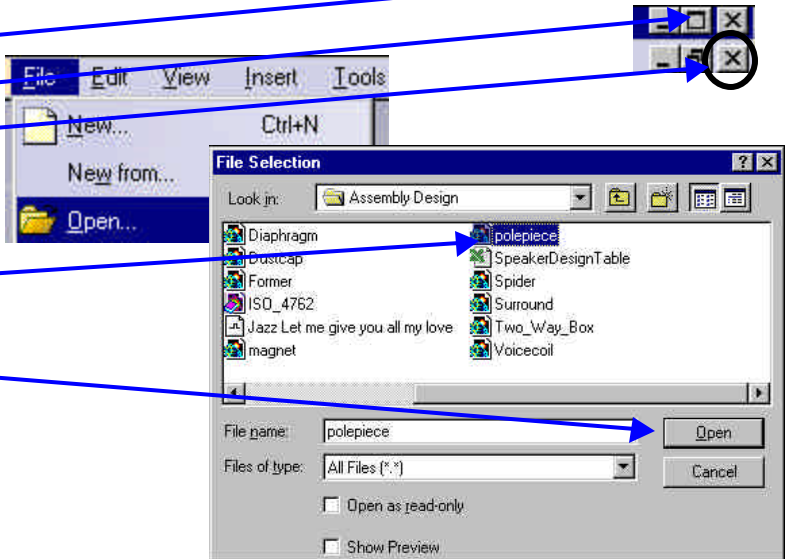
CATIA User Interface

- IMPORTANT**

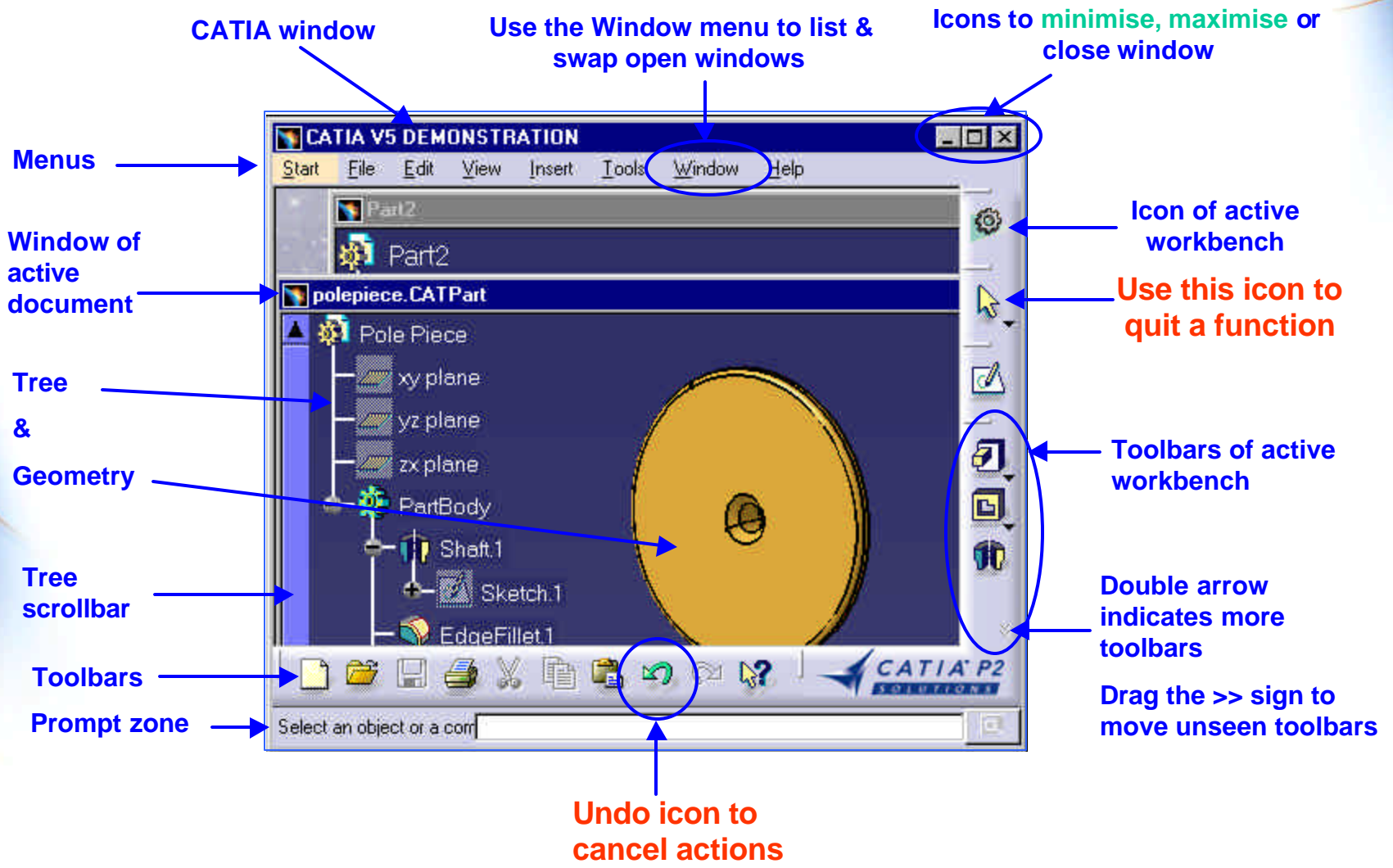
- ✍ The next pages are very important to use CATIA correctly.
- ✍ You MUST READ these pages and refer to them if you don't remember how to use some of the functionalities.

- ✍ First, open a CATIA file.

- ✍ Launch CATIA if it is not already launched
- ✍ Click on the Close button
- ✍ Maximise the CATIA window
- ✍ Close the Product1 window
- ✍ Open the file polepiece.CATPart under the ... \DATA\Assembly Design\ directory
- ✍ Select the polepiece.CATPart file
- ✍ Click on the Open button



CATIA User Interface



CATIA User Interface

- **Dialogue boxes provide parameters for the definition of features. They are standardised across all functions.**

EX: if you double-click on a hole you will get the **Hole Definition** dialogue box.

Click the question mark and select a field to get help
Click the cross to close the dialogue box

The image shows a screenshot of the 'Hole Definition' dialog box in CATIA. The dialog box has a title bar with a question mark icon and a close icon. It contains several tabs: 'Extension', 'Type', and 'Thread Definition'. The 'Extension' tab is active. The dialog box is divided into several sections: 'Up To Last' (a dropdown menu), 'Diameter' (a text field with '0.591in'), 'Depth' (a text field with '2.37in' and a lock icon), 'Limit' (a text field with 'No selection'), 'Offset' (a text field with '0in'), 'Axis' (a section with a 'Reverse' button and a 'Normal to surface' checkbox), 'Bottom' (a section with a 'Flat' dropdown and an 'Angle' text field with '120deg'), and 'Positioning Sketch' (a section with a sketch icon). At the bottom of the dialog box are three buttons: 'OK', 'Apply', and 'Cancel'. Blue arrows point from text labels to these UI elements.

Name of the task →

Select a tab to define particular attributes →

Select an option from the drop down list →

Key in a value or click up and down arrows →

Fields not selectable (locked) →

Select button to activate the option →

Click check box to activate the option →

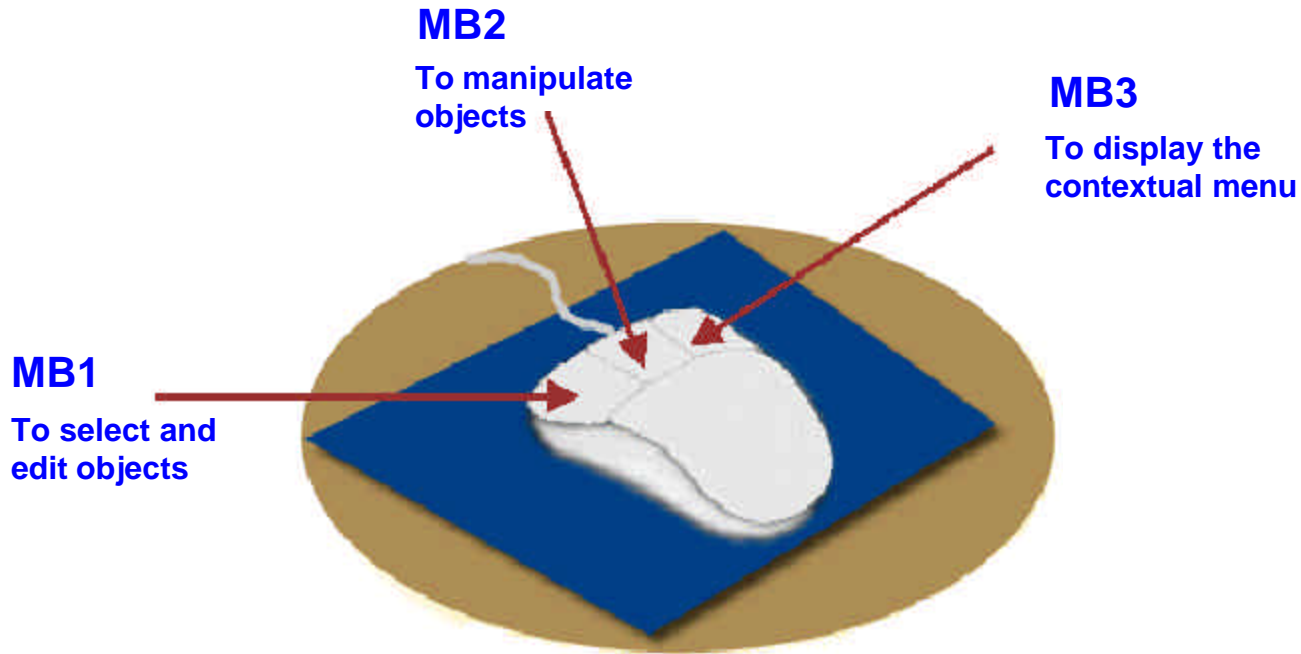
Inactive field →

Click OK to confirm →

Click Apply to pre-visualise the result →

Click Cancel to leave the function →





•Using the Mouse Function




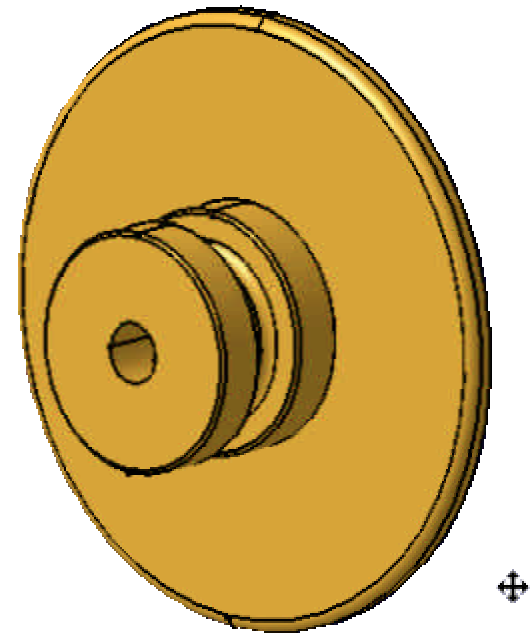
***MB* means Mouse Button**

•Panning Objects

 **It's important you practice using the mouse.**

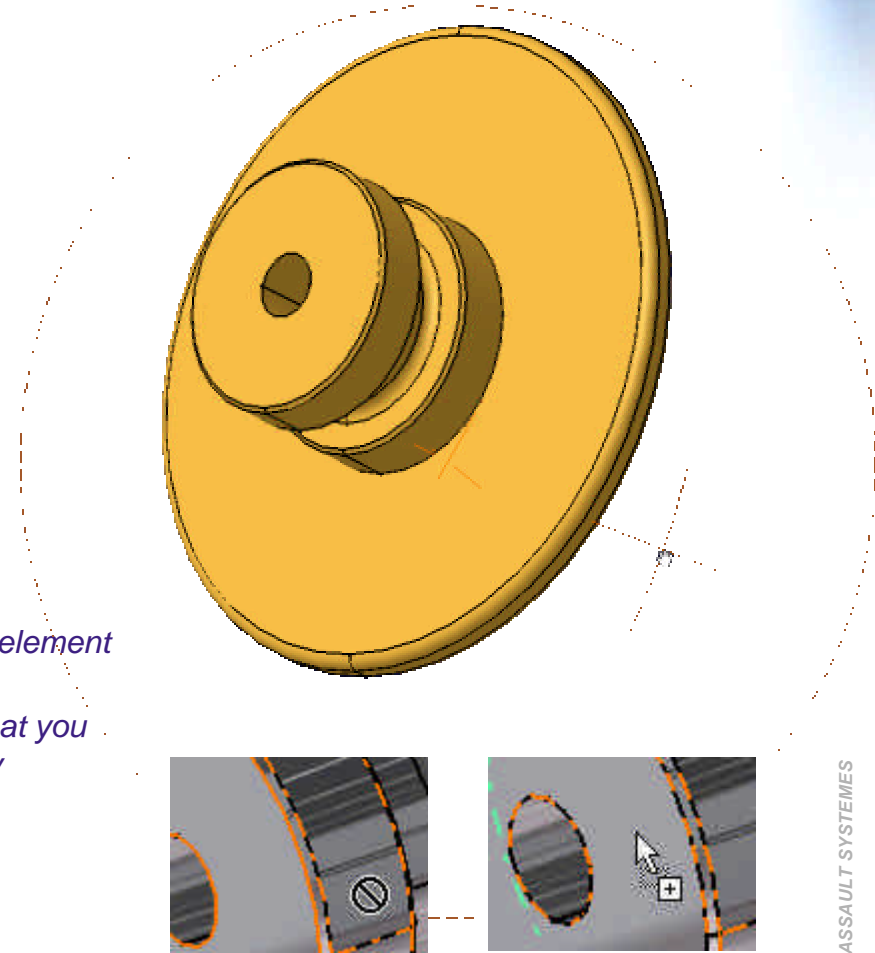
-  1. Position the cursor anywhere on the screen
-  2. Press and Hold Mouse Button 2 (MB2)
-  3. Move the mouse where you want to drag the part
 -  The cursor will change to a cross

 If the part disappears, get it back with the Fit-All-In icon in the View toolbar.



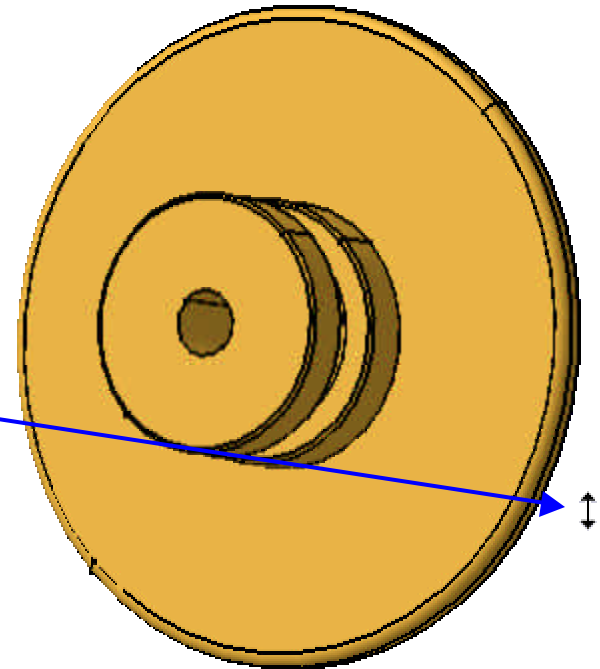
•Rotating Objects

1. Position the cursor anywhere on the screen
 2. Press and Hold mouse button 2 (MB2)
 - The axis appears in the centre of the screen
 3. Press and Hold mouse button 1 (MB1) while keeping MB2 pressed
 - The cursor changes to a hand and a red circle appears representing a virtual space ball
 4. Move the mouse to rotate the part
 - The cursor will change to a cross
- To change the rotation axis, click once on MB2 on the element that you want at the centre of the screen.
- P.S.: if you see a « + » or a Forbidden sign it means that you have not held MB1 and you will duplicate the geometry



•Zooming

1. Position the cursor anywhere on the screen
2. Press and Hold mouse button 2 (MB2)
 - ☞ An axis and a circle appear in the centre of the screen
3. Press and release MB1 while keeping MB2 pressed
 - ☞ The cursor changes to a double arrow
4. Move the mouse up to zoom in
5. Move the mouse down to zoom out



☞ To zoom step by step click on + and – icons in the View toolbar



•Moving the Tree

Using the Scrollbar

- When and only when the size of the tree exceeds the window you can use the scrollbar to display the tree downward or upward

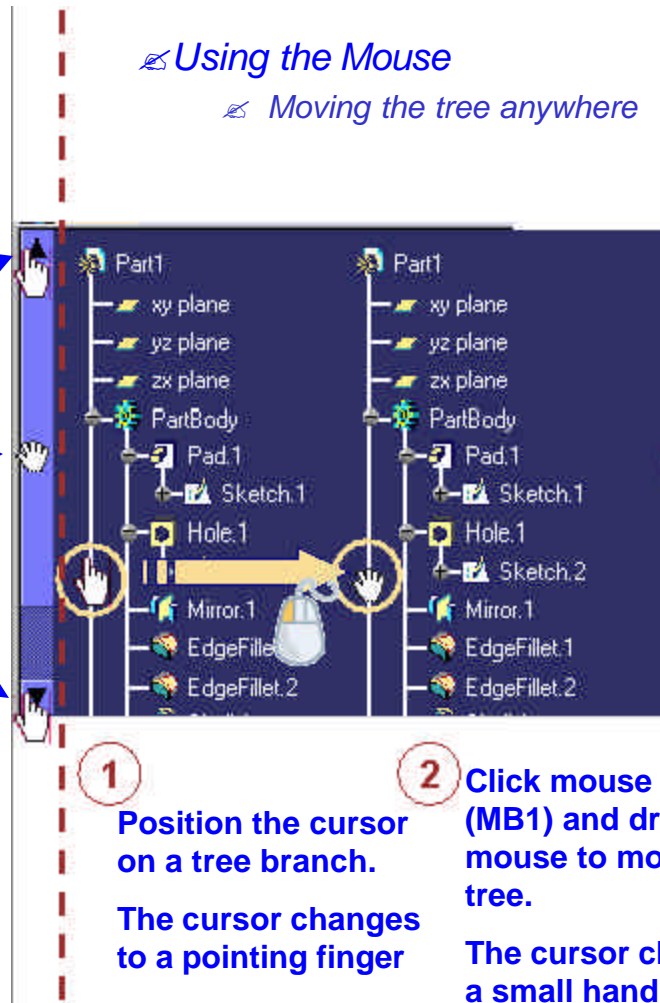
Click up arrow to display the top of the tree

Drag the bar up and down as you wish

Click down arrow to display the bottom of the tree

Using the Mouse

- Moving the tree anywhere



CATIA User Interface

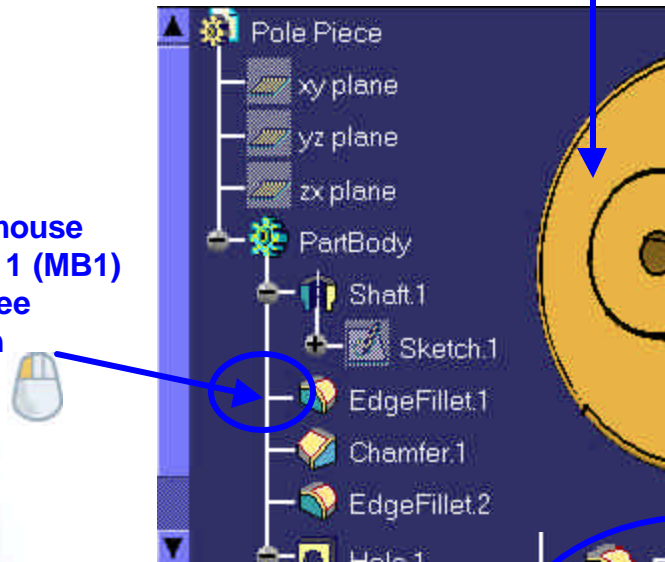
• Adjusting and Expanding the Tree

To adjust tree size:

Geometry dimmed while adjusting the size of the tree



1
Click mouse button 1 (MB1) on a tree branch



2
Zoom in and out, as explained two pages before



3
To reactivate the geometry, click again on a tree branch

⚡ Note that you can use the scrollbar to display the top or bottom of the tree

CATIA User Interface

©2001 DASSAULT SYSTEMES-R2

CATIA Documents

When working in CATIA you can create, modify and save geometries within documents

Documents are containers

Different Types of Documents

The type of document depends on the workbench you use to create and modify a geometry :

 **Assembly Design** → .CATProduct

 **Part Design**

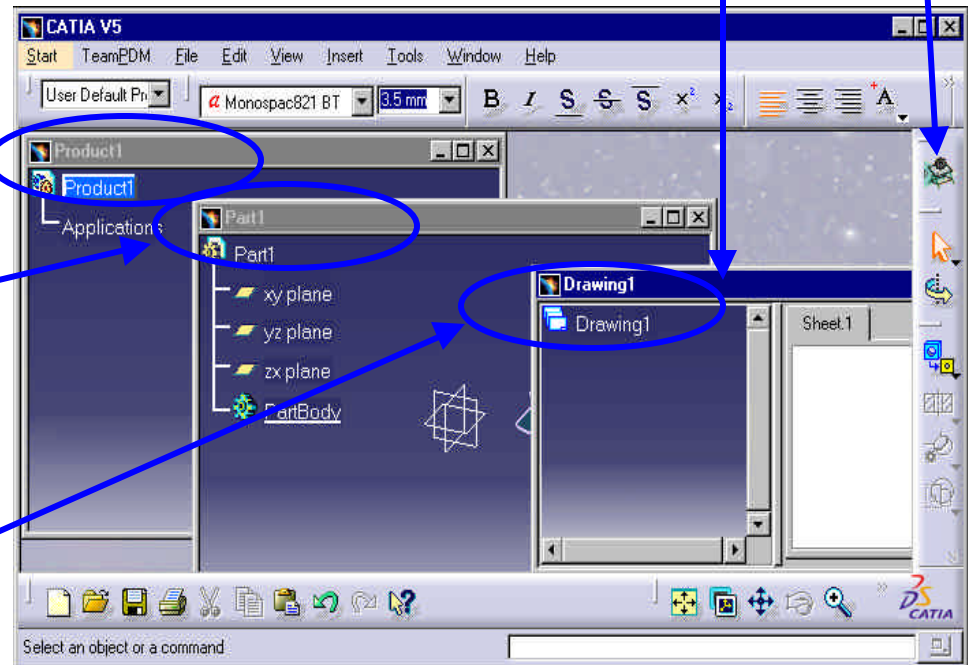
 **Sketcher**

 **WireFrame & Surface**

.CATPart

 **Generative & Interactive Drafting** → .CATDrawing

Current workbench
Current document

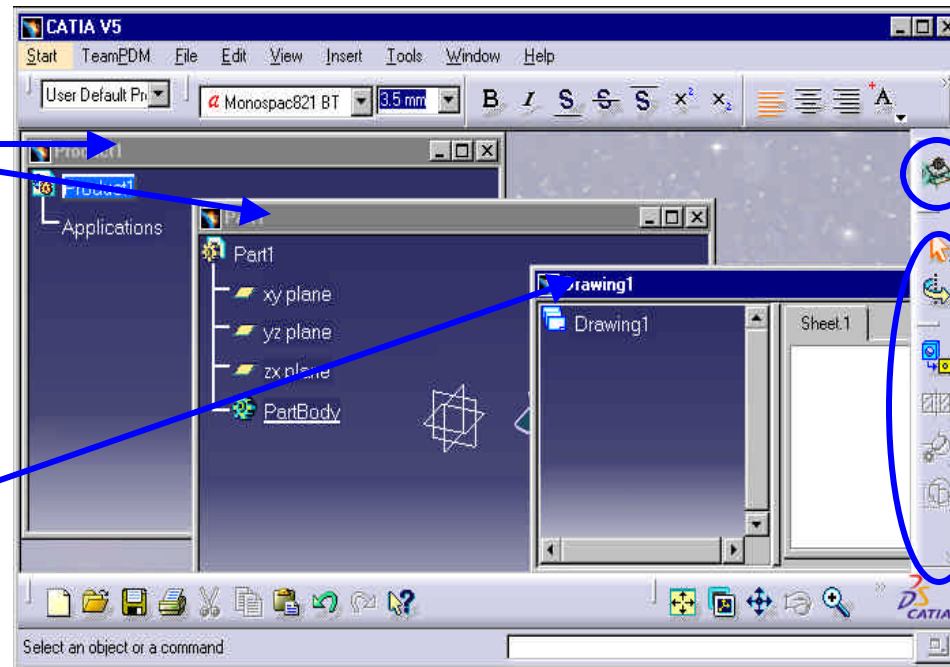


•Terminology

- ✦ A Product Structure is a way to structure and **organise** your products logically. You can navigate within the structure and work on its different components
- ✦ A Document is a file including data you can create and manage with the associated workbench (specific extension)
- ✦ A Workbench is a set of tools that allows you to create and manage your geometry

Inactive documents have a grey title bar

Current document (blue title bar)



Current (or active) workbench



Workbench toolbars...


SKETCHER Workbench Presentation



• Exercise

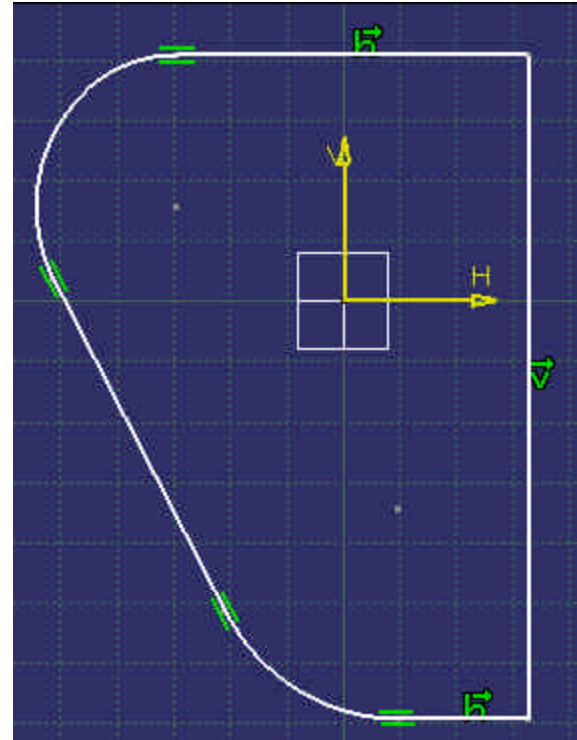
☞ Open a new Part : Start + Mechanical Design + Part Design

① Select a plane, a Solid face, or a Planar Surface to Sketch on

② Access the Sketcher Workbench 

③ Select the Polygon icon and draw a sketch as shown 

- ☞ Click and release MB1 where you want to start a line
- ☞ Click MB1 where you want to finish the line
- ☞ Hold MB1 then move the mouse and release MB1 to draw an arc
- ☞ Double-click when you want to finish your profile



☞ *If you want to exit the function click on the **Select** icon*



• Exercise

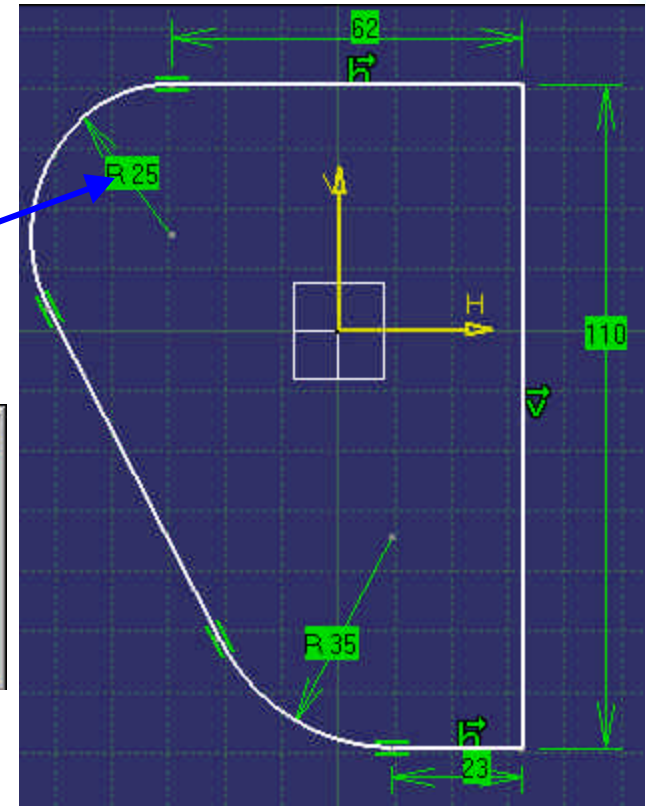
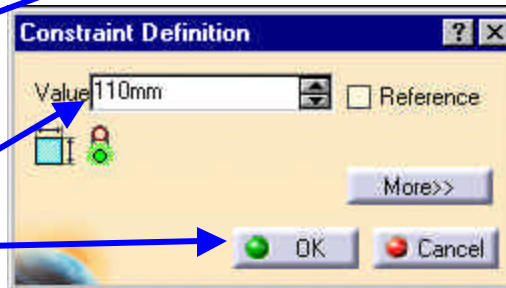
4 Select the Constraints icon and set dimensions



- To set a dimension, select a line, drag the dimension to the desired position
- Click to create it

5 Change Dimensions as shown

- To modify a Dimension, double-click on the green dimension value box.
- Change the value and click OK to validate



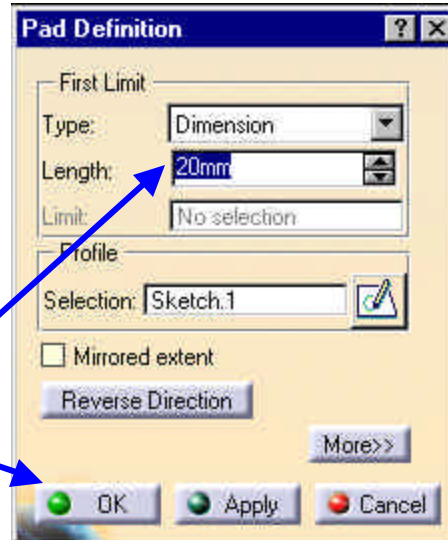
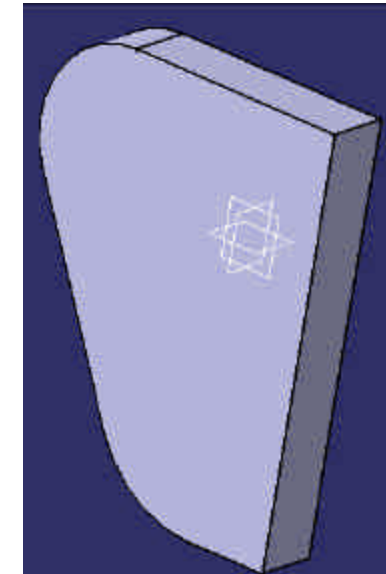
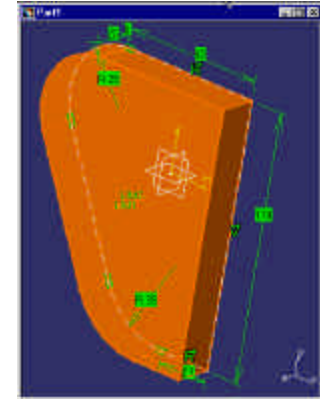
6 Exit the Sketcher



• Exercise

7 Click on the Pad icon 

Select the Sketch if necessary




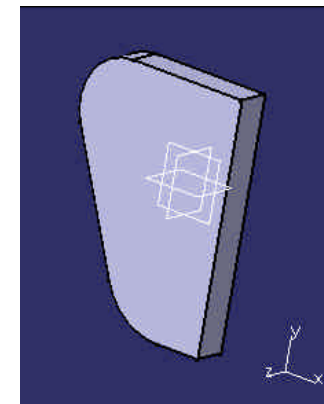
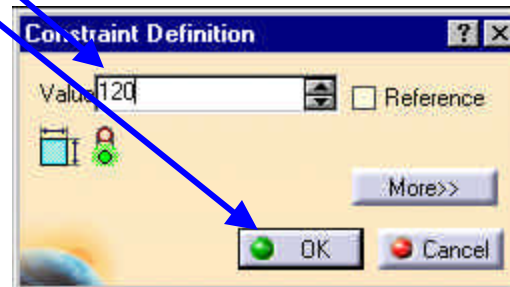
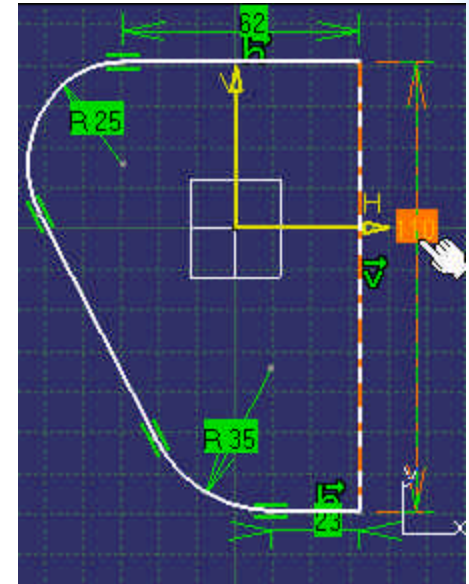
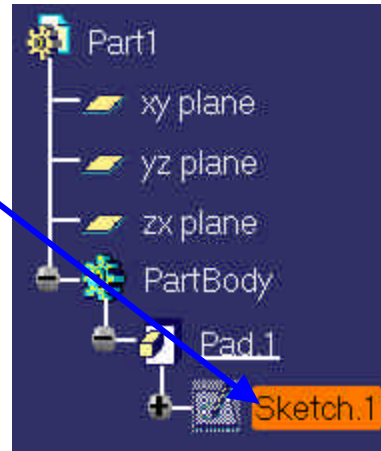
8 Click OK to validate

You can change the length value before validation

• Exercise

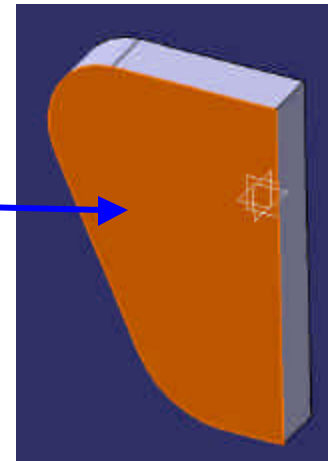
⑨ Double-click on Sketch.1 in the Tree to enter Sketcher

- ☞ Double-click on the 110 dimensions
- ☞ Change the value to 120
- ☞ Click OK to validate
- ☞ Exit the Sketcher 



• Exercise

10 Select the face as shown and enter Sketcher



11 Select the Circle icon



Click to define the circle centre

Drag the mouse to define the radius

Click to create the circle



12 Exit the Sketcher

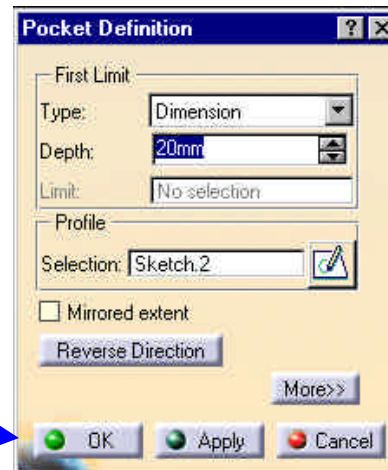


13 Select the Pocket icon

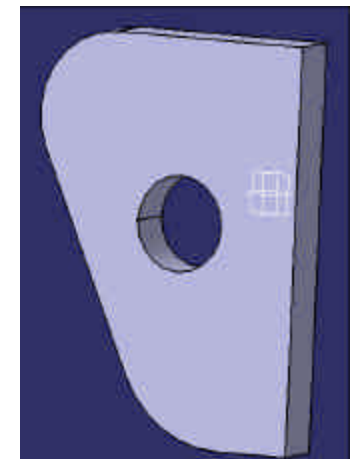


14 Click OK to validate

You can change the depth value before validation



15 Close all the windows except the CATIA one. This is the END of the Sketcher Training



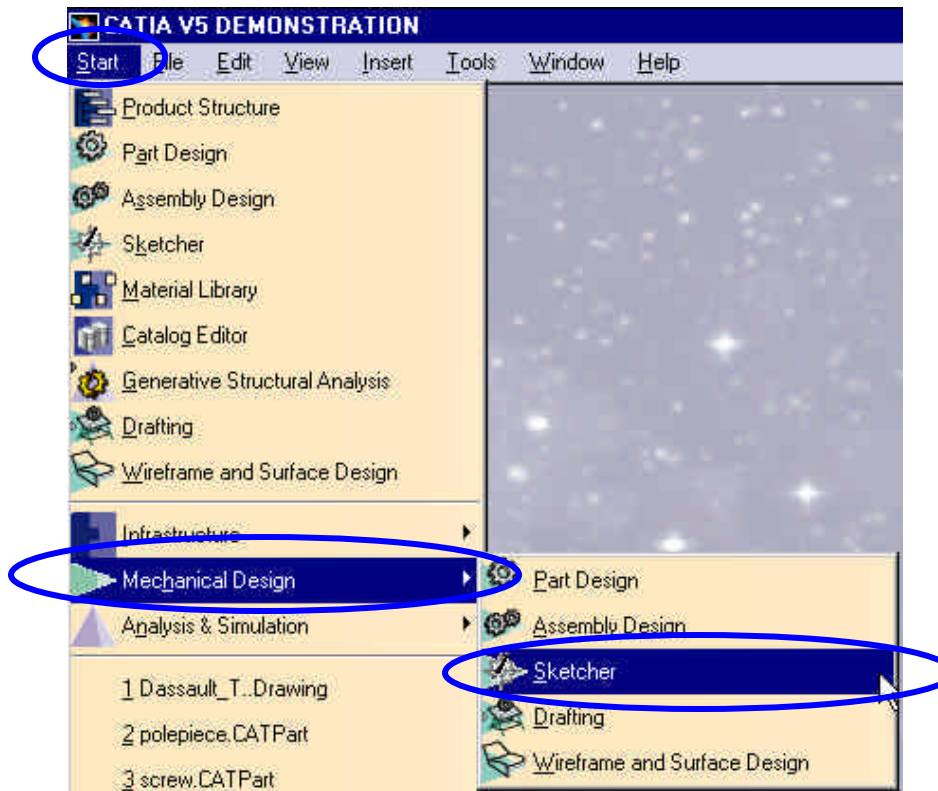
SKETCHER Workbench Presentation

- **Let's have a short Debriefing...**

- ✍ The Sketcher is a set of tools to help the user quickly generate 2D geometry
- ✍ The completed Sketch can then be used to generate solids and surfaces
- ✍ The capability to define constraints between elements in the Sketcher allows for quick modifications of the Sketch on subsequent solids or surfaces
- ✍ Others tools such as Animate Constraints enable the user to explore design alternatives

- **Accessing the Workbench**

- ✍ Select Start + Mechanical Design + Sketcher
- ✍ Select a plane or a face of an object



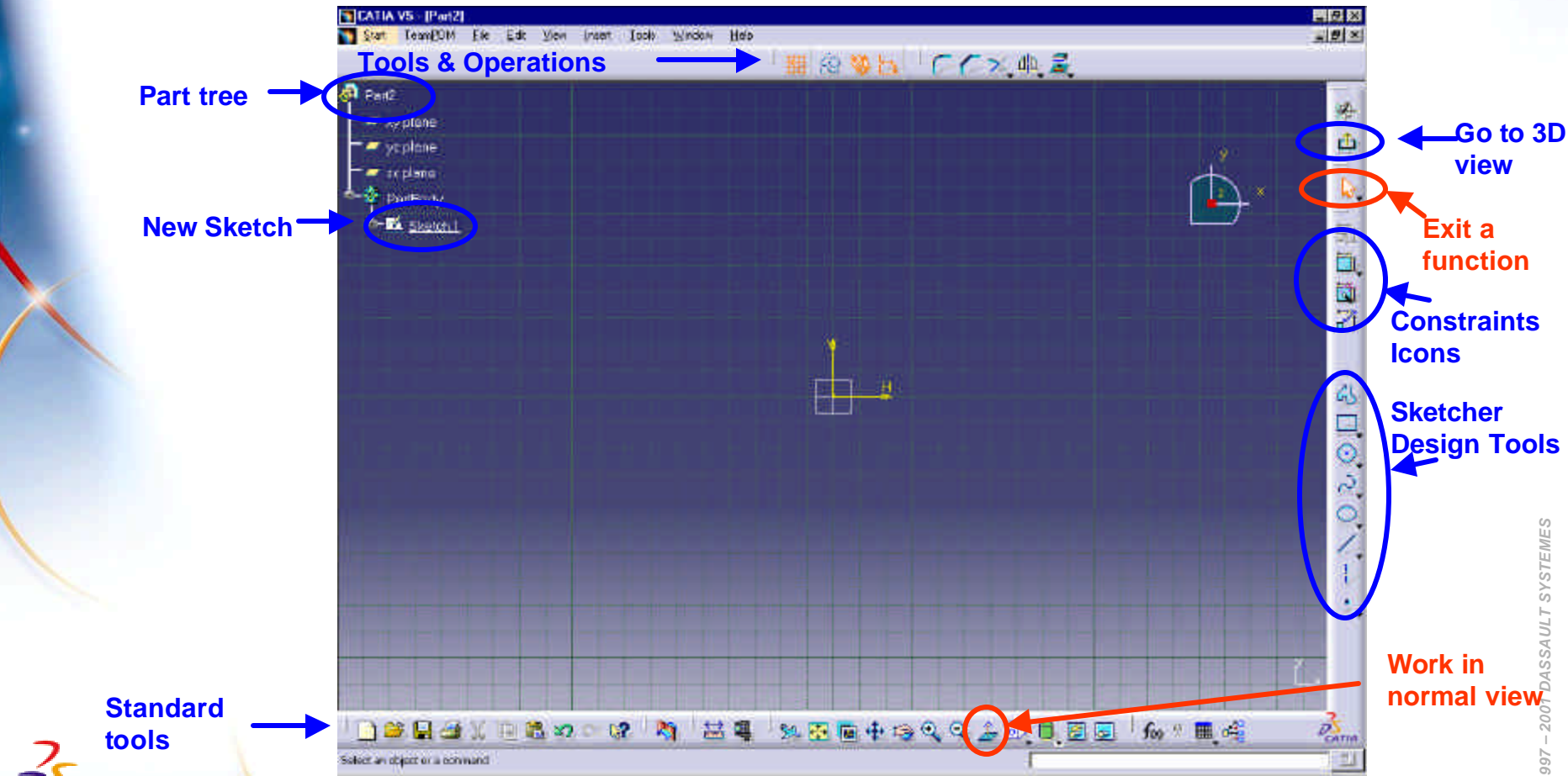
You can also access the Sketcher by selecting the Sketcher icon from any Workbench



Learning Sketcher

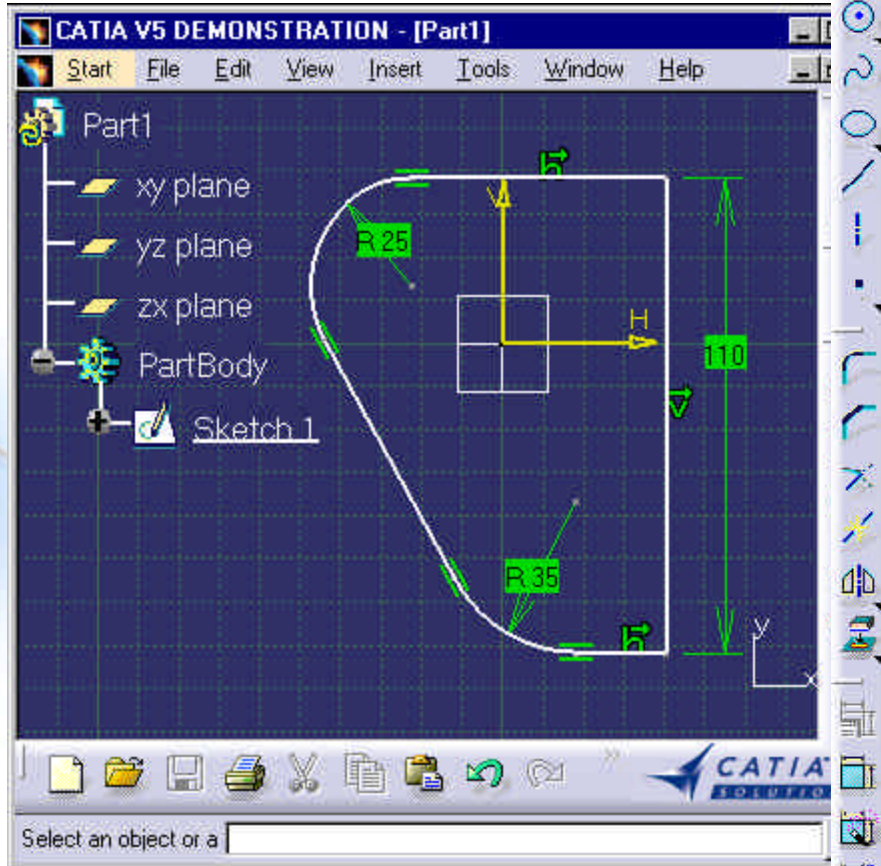
• Sketcher Workbench overview

✍ A new Sketch will appear in the Part Tree when entering the Sketcher Workbench



Learning Sketcher

• Sketcher Tools



Exit Sketcher

Profile

Rectangles, Keyholes, Polygons...

Circles, Ellipses, Arcs...

Spline

Ellipse

Line

Axis

Points...

Corner

Chamfer

Trim options...

Break

Symmetry

Projection

Constraints **dialogue box**

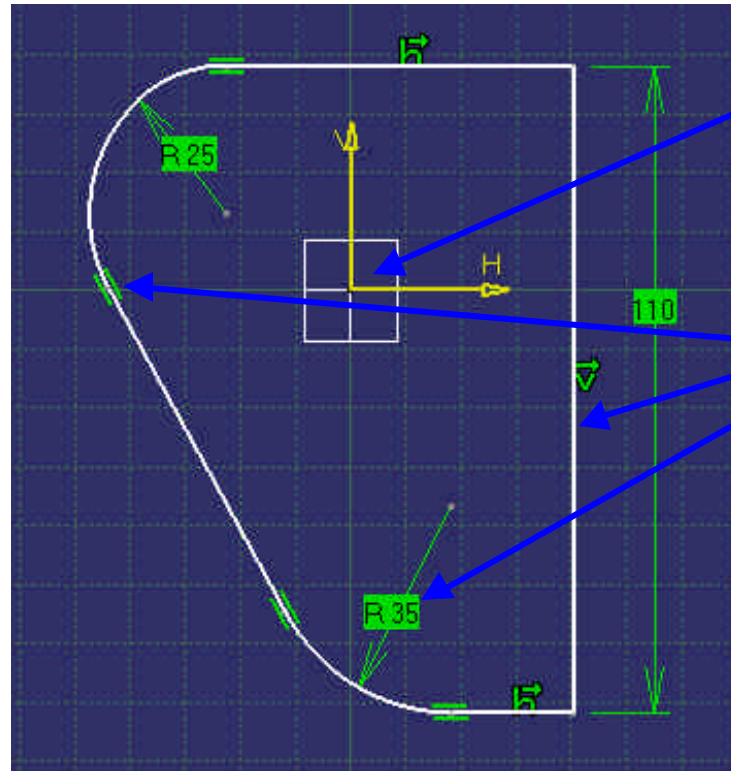
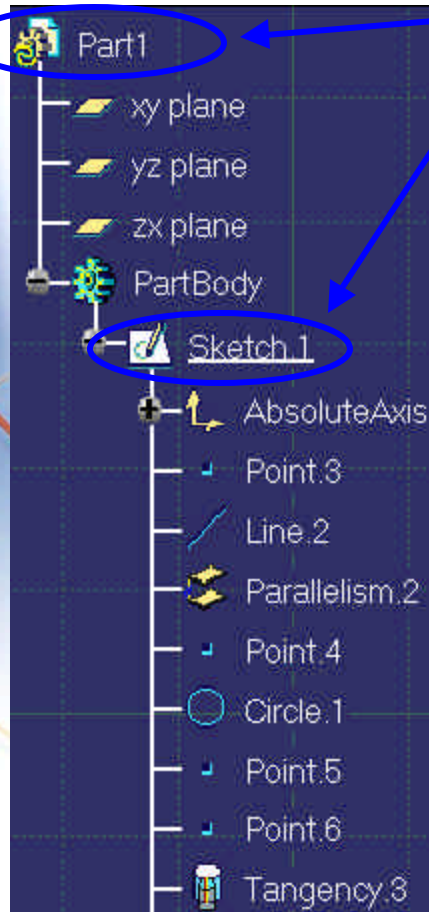
Constraints

Auto Constraints

Animate Constraints

• Terminology

The Sketch is the holding place for a group of 2D elements on a specific plane. There can be more than one Sketch using the same plane as support.



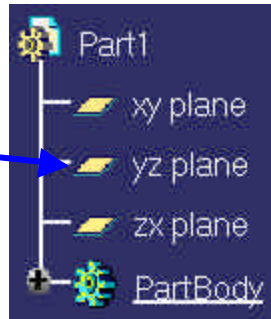
The V-H Axis is the origin of the Sketch.

Sketches generally consist of a Profile, Constraints, and Dimensions (a type of Constraint).

• General Process

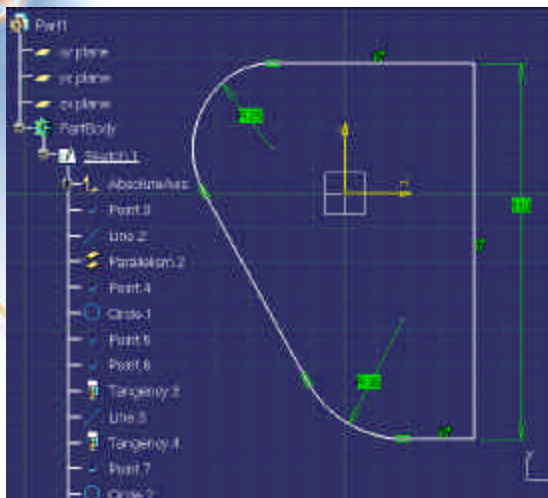
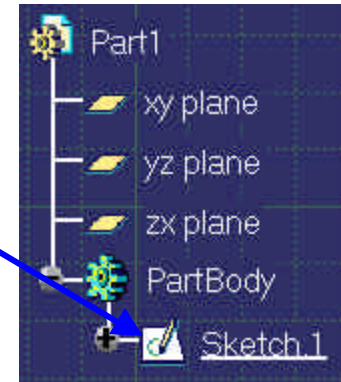
① Select a plane, a Solid face, or a Planar Surface to Sketch on

② Access the Sketcher Workbench



③

An In-Work Sketch is added to the Specifications Tree



④

Create Geometric Elements and Constraints

⑤

Exit the Sketcher



Use the Sketch to create a Solid or a Surface

