DASSAULT SYSTEMES

CATIA V5 Training
Foils

CATIA V5 Fundamentals

Version 5 Release 19
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EDU_CAT_EN_V5F_FI_V5R19

Introduction to CATIA

About this Course

Introduction

CATIA is a robust application that enables you to create rich and complex designs. The aim of the CATIA V5 Fundamentals course is to teach you how to build parts and assemblies in CATIA, and how to make simple drawings of these parts and assemblies. This course focuses on the fundamental skills and concepts that enable you to create a strong foundation for your designs.

Course Design Philosophy

This course adopts a process- or task-based approach to training. Rather than focus on individual features and functions, it emphasizes the process and procedure to complete a particular task. Using case studies that illustrate these processes, you will learn the necessary commands, options, and menus within the context of completing a design task.

Target audience

The target audience for this course are new CATIA V5 mechanical designers.

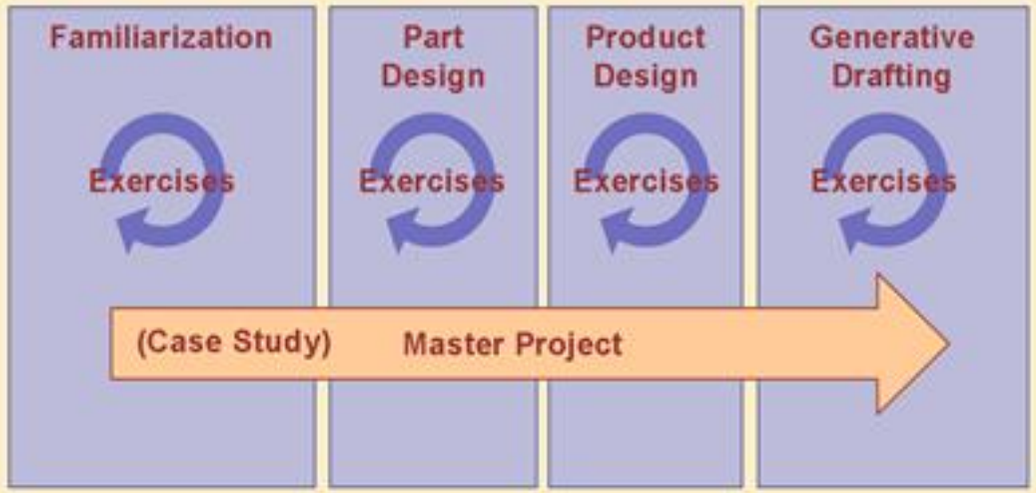
Prerequisites

Students attending this course should have:

- Experience in Mechanical design.
- Experience of using the Windows operating system

Case Study

Each lesson in this course contains a case study, which explains the skills and concepts covered in the lesson. The case study will be described at the beginning of each lesson, and the student will be able to do the case study exercise once the theory for that lesson has been covered.



Design Intent

Each case study contains a set of model requirements, known as the design intent. The first case study does not contain a design intent because you are not going to design anything. However, by the end of this lesson you should be able to:

- ✓ Change the orientation of a model.
- ✓ Change the visualization properties of a model.
- ✓ Manipulate the specification tree.
- ✓ Access the Help system of CATIA.

Stages in the Process

Each lesson consists of steps. You will go through the following steps to introduce yourself to CATIA:

1. Understand the CATIA software.
2. Open CATIA.
3. Understand the CATIA interface.

Understand the CATIA Software

In this section, you will learn about PLM and how it relates to CATIA. This section also introduces the key features of CATIA.

Use the following steps:

1. **Understand the CATIA software.**
2. Open CATIA.
3. Understand the CATIA interface.

PLM – Product Lifecycle Management

Defining Product Lifecycle Management –

A strategic business approach that applies a consistent set of business solutions in support of the collaborative creation, management, dissemination, and use of product definition information across the extended enterprise from concept to end of life – integrating people, processes, business systems and information

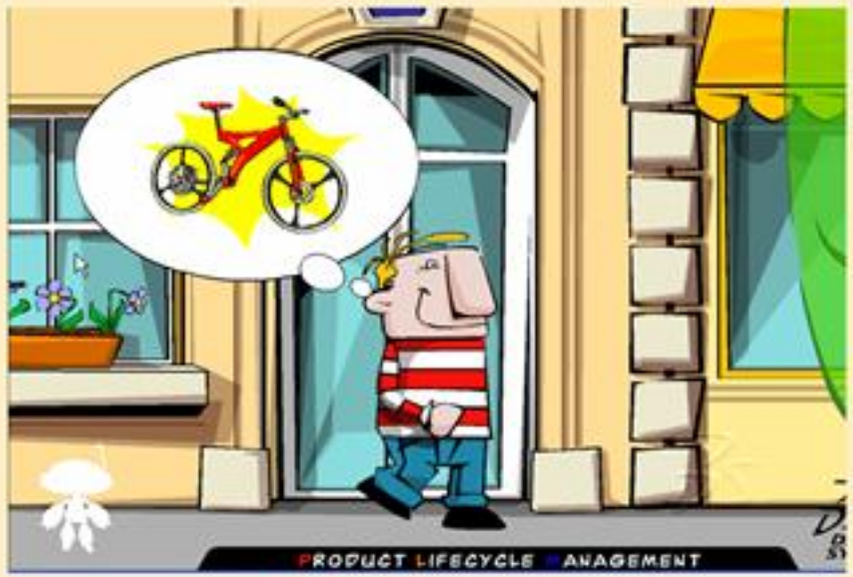
CIMdata®

Source: CIMdata Inc.

PLM in Practice

- PLM breaks down the barriers in technology that have so far limited the interaction between people who design products and people who build, sell, buy, and use them.
- Using the collaborative power of Dassault Systemes Solutions, people can now experience PLM with organizations beginning to create and develop innovative product design, while reducing cycle times, streamlining manufacturing, and cutting production costs.
- Let's see a real-life example of how PLM can change people's experiences as innovators, manufacturers, and consumers…

Searching for a new bike!

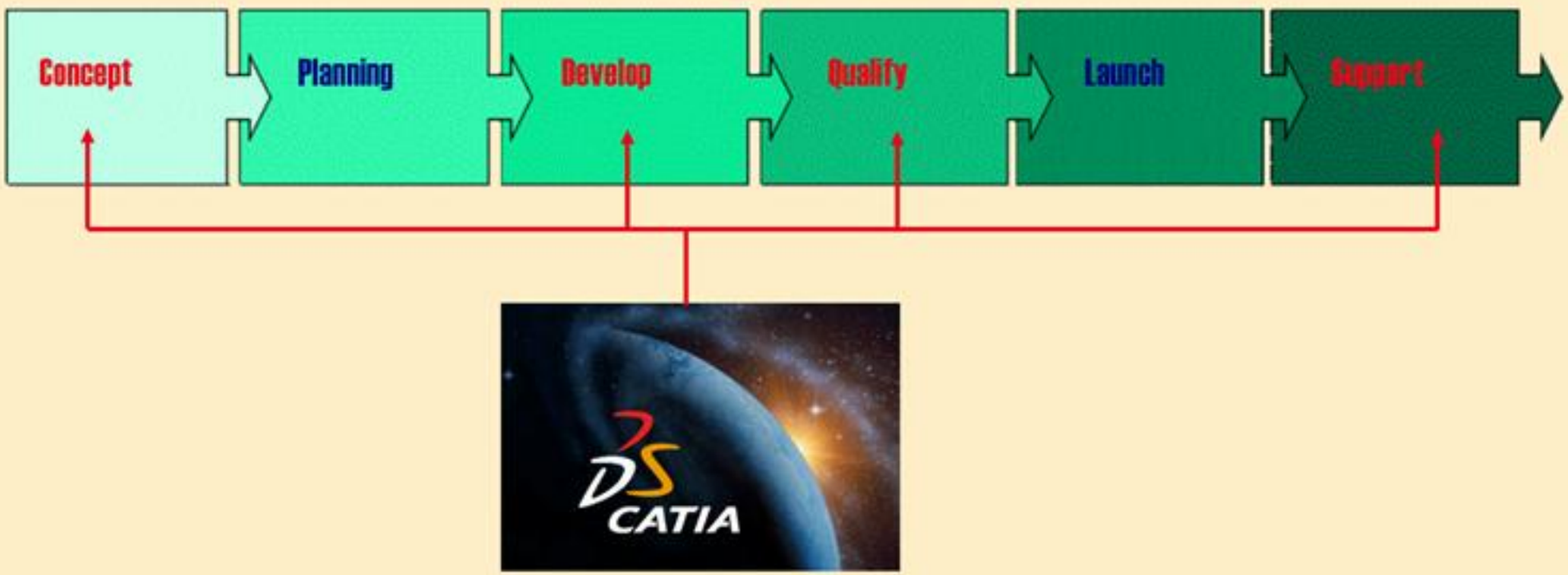


CATIA Within the PLM Solution

For the following functions CATIA plays an integral part of the PLM solution:

CATIA Coverage

From Concept To Realization…

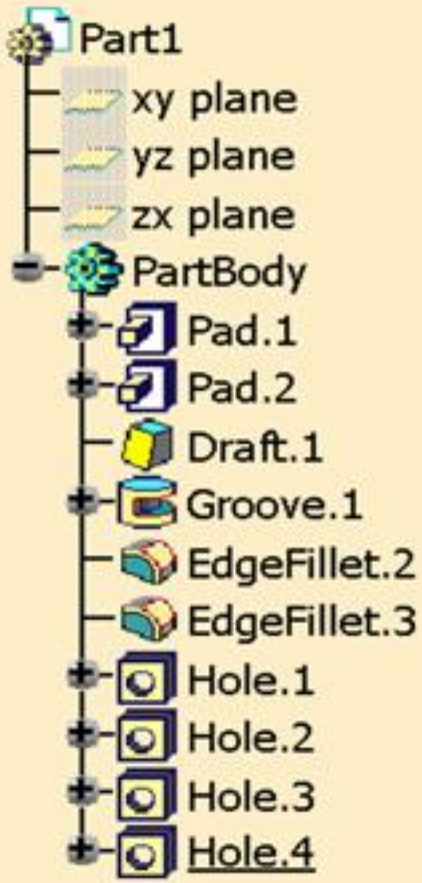


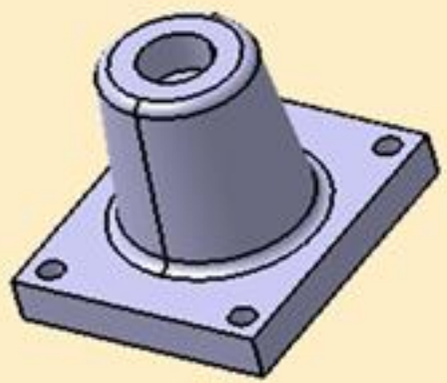
What is CATIA V5?

CATIA is mechanical design software. It is a *feature-based, parametric solid modeling* design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create *fully associative* 3-D solid models with or without *constraints* while utilizing automatic or user-defined relations to capture *design intent*. To clarify this definition, the italic terms above will be further defined.

What do these words mean?

- Feature Based
- Parametric
  - Driving Dimensions
  - Relations
- Solid Modeling
- Fully Associative
- Constraints
- Design Intent





Design Intent (1/3)

- Consider the design intent before and during the modeling of the part.

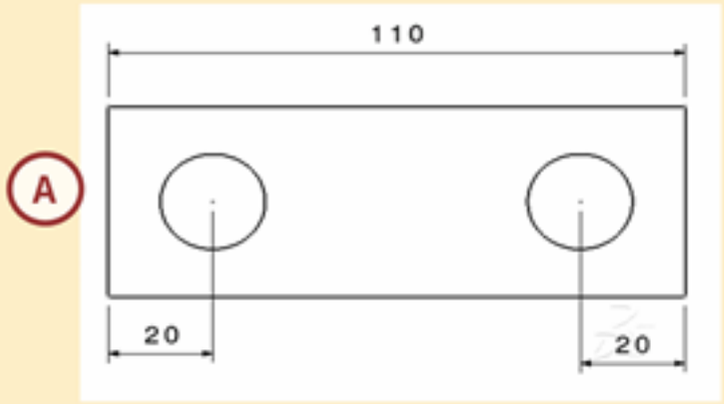
- The captured (modeled) design intent will effect how the model behaves when it is changed during it's life cycle.

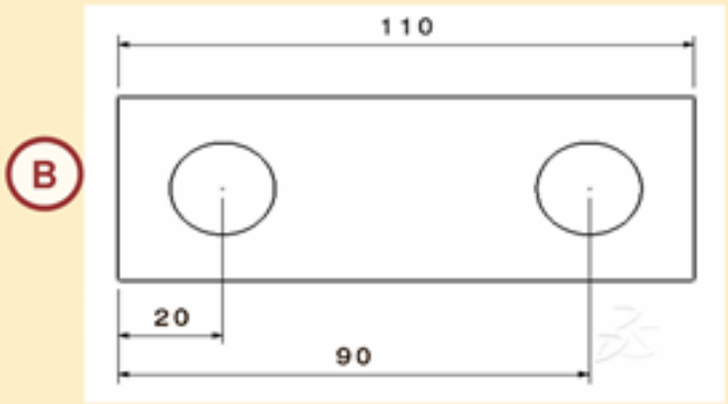
- The way a solid model of a part is built can affect many aspects:
  - Flexibility to changes
  - Stability during the change process
  - Resource requirements to compute a new result

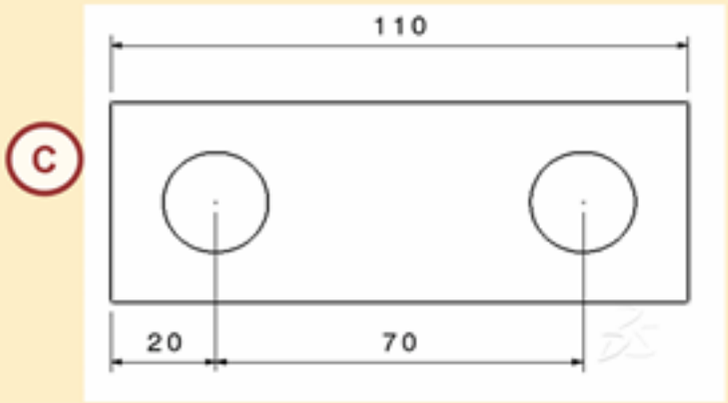
- Factors contributing to the design intent:
  - Automatic (Implicit) Relations
  - Equations
  - Additional Relations
  - Dimensioning

Design Intent (2/3)

Examples of Design Intent:

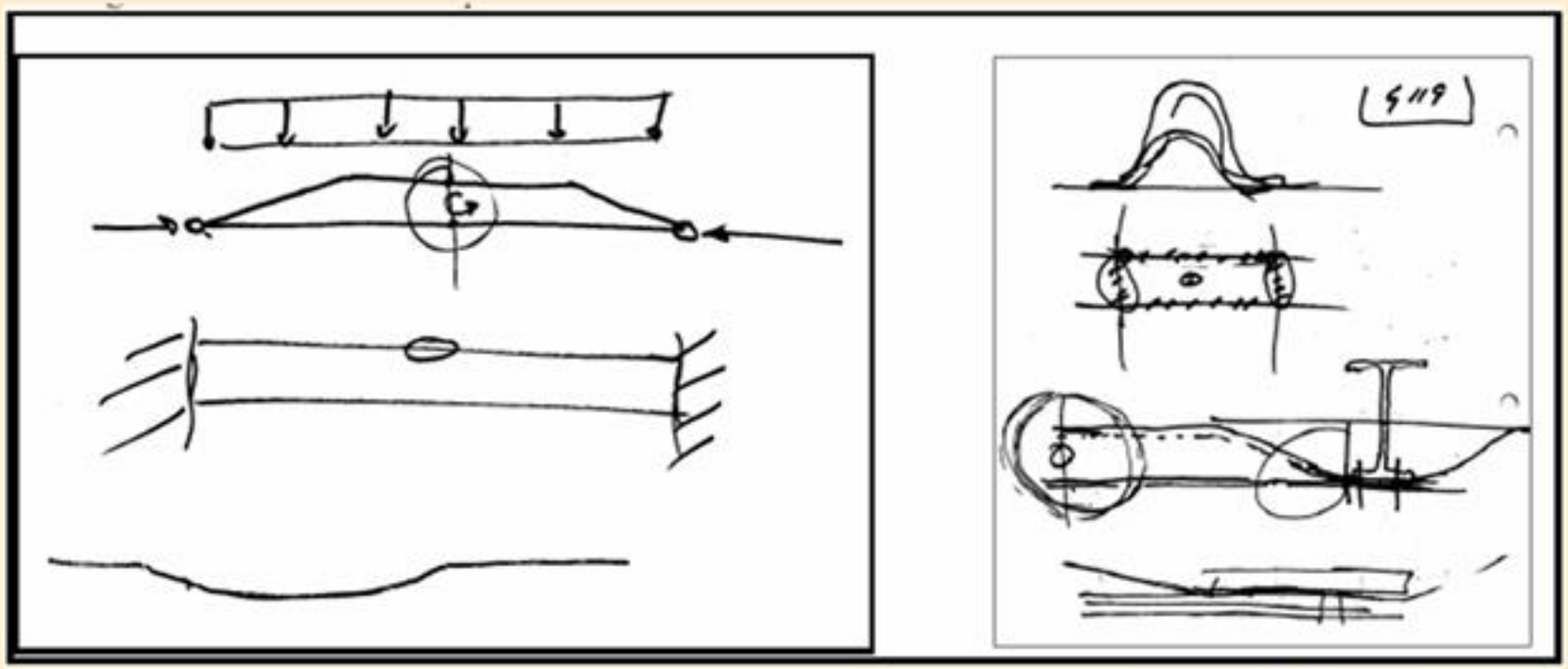






Design Intent (3/3)

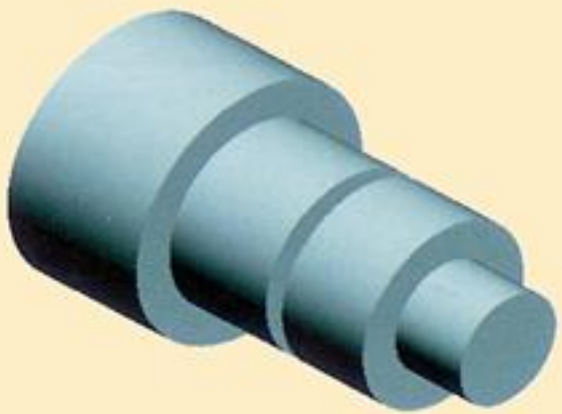
Through pre-planning, you can become very efficient in solid modeling. You can create a solid model much easier and quicker than your peers. It also provides a clear vision for any modifications.



Provided by the Boeing Company

How Features affect Design Intent (1/2)

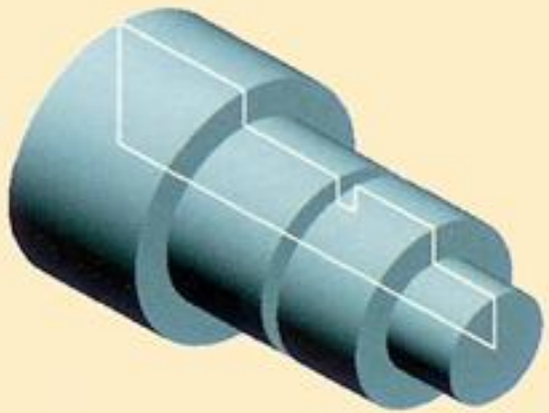
- Design intent is affected by more than just how a sketch is dimensioned.
- The choice of features and the modeling methodology are also important.



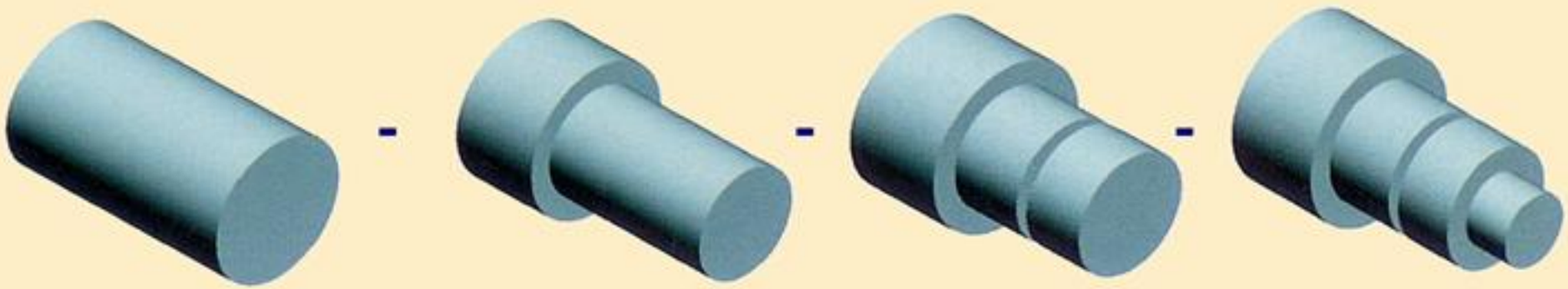
"Layered" Approach

How Features affect Design Intent (2/2)

"Potter's Wheel" Approach



"Manufacturing" Approach

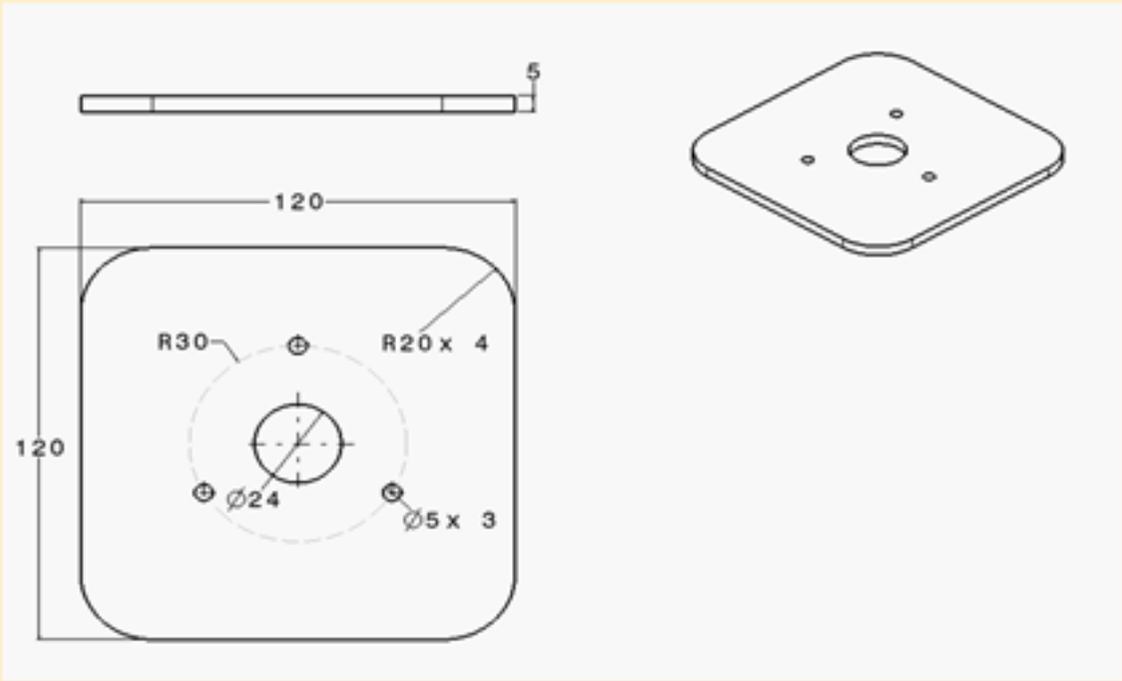


Exercise Overview

You will practice what you have learned by working through a simple self assessment and a short exercise.

Design Intent

Number of questions	***: 6***
Duration	***: Unlimited***
Scored	***: no***
Allow to change previous answers	***: yes***



Open CATIA

In this section, you will learn how to open CATIA in a Windows environment.

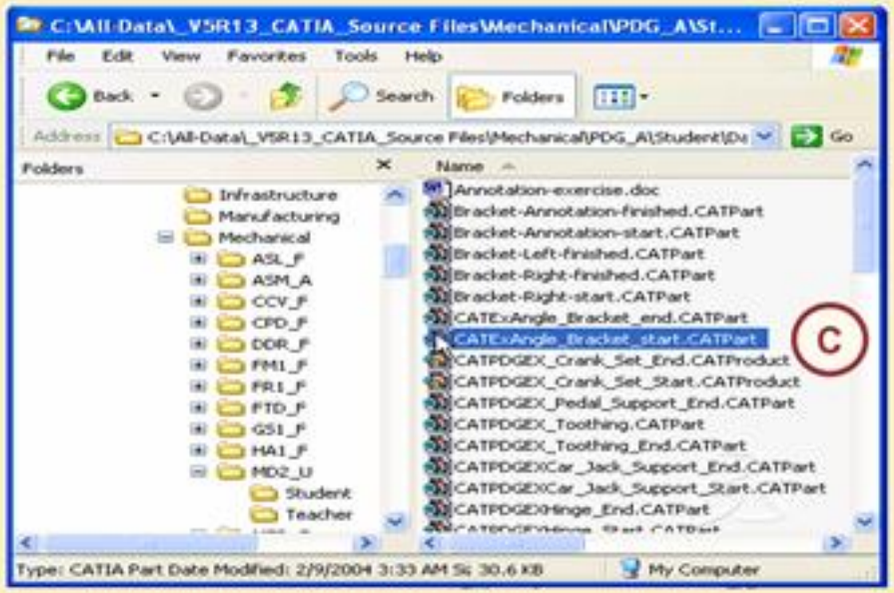
Use the following steps:

1. ✔ Understand the CATIA software.
2. **Open CATIA.**
3. Understand the CATIA interface.

Starting CATIA Using the Start Menu

In a Windows environment, you can start the CATIA application in several ways:

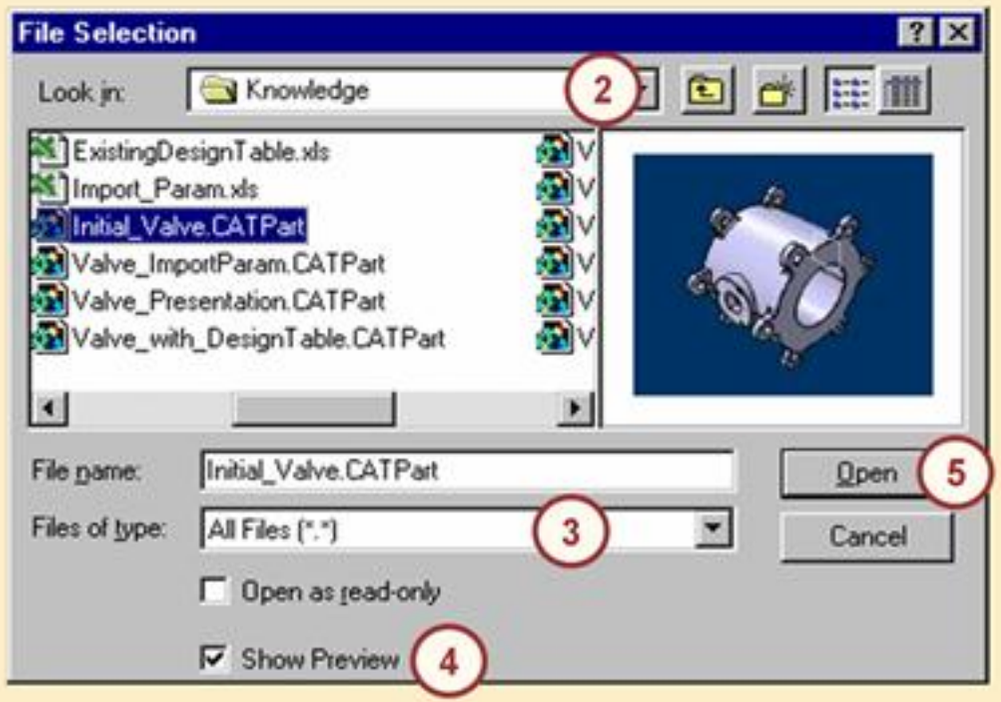
A. Select **CATIA** from the **Start > Programs > CATIA** menu.
B. Double-click the CATIA icon on your Windows desktop.
C. Double-click on an existing CATIA document.



Opening an Existing Document

Use the following steps to open an existing document within CATIA:

1. Select **File > Open**, or click the Open icon.
2. Browse to the file's location using the Look in list of the File Selection dialog box.
3. In the Files of type list, select the type of document you want to open.
4. Select the Show Preview checkbox to see a preview of the selected file.
5. Click Open.



Note that if you are working in an environment where CATIA V5 is connected to a Product Data Management (PDM) system, CATIA documents will not be stored as files on a local disk, but will be managed by the PDM system. The interface will differ from that shown above.

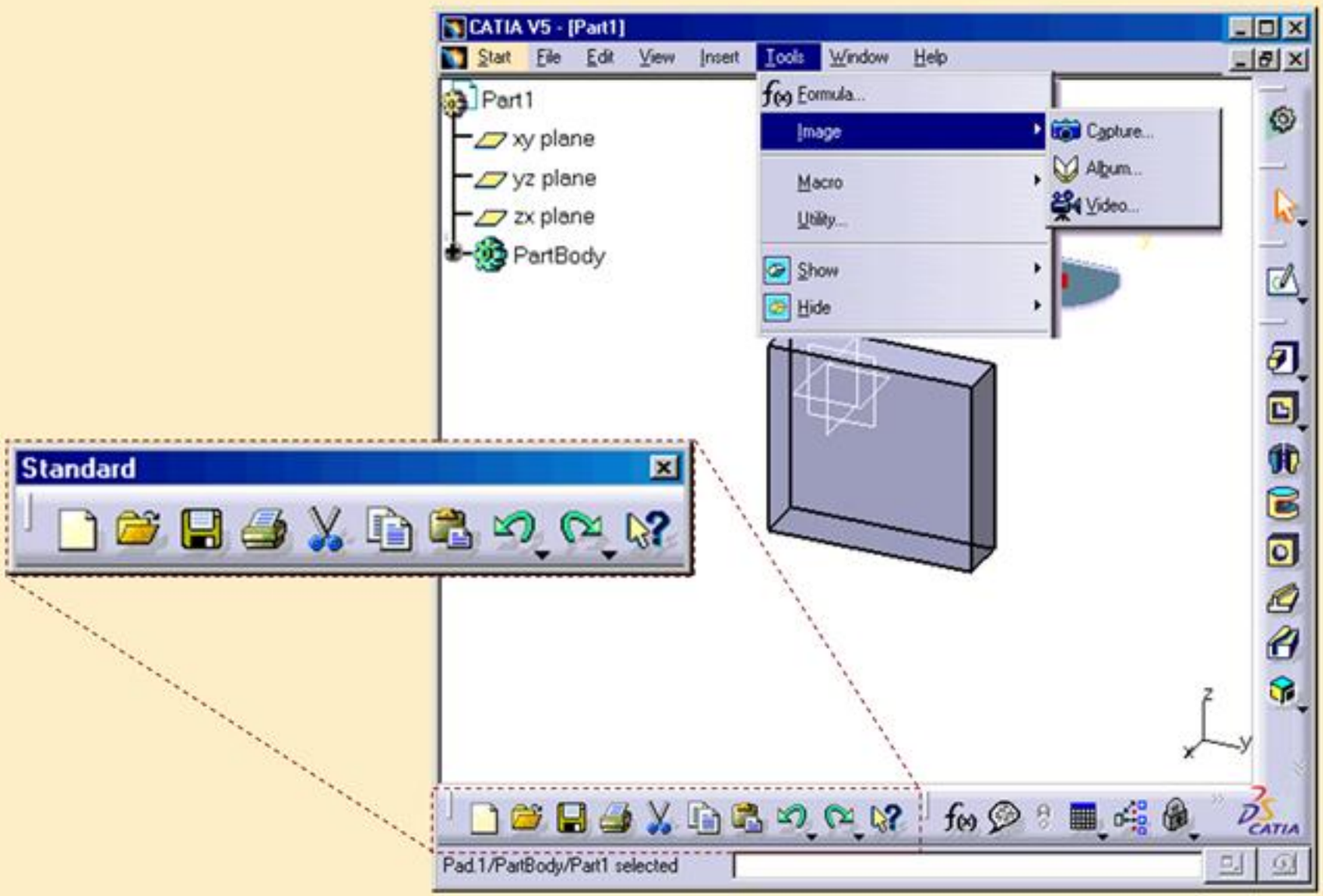
Understand the CATIA Interface

In this section, you will become familiar with the CATIA User interface.

Use the following steps:

- ✔ 1. Understand the CATIA software
- ✔ 2. Open CATIA
- 3. **Understand the CATIA Interface**

The Windows Philosophy (1/3)



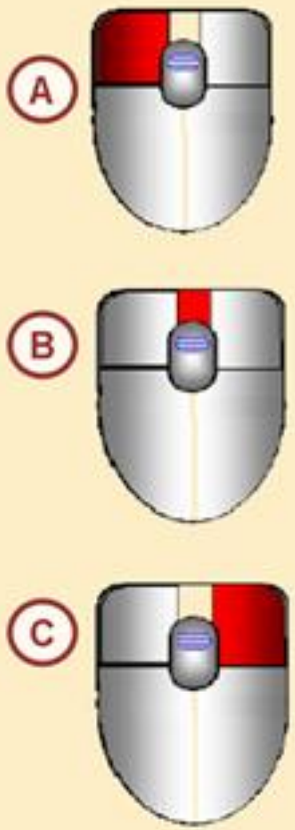
The Windows Philosophy (2/3)

CATIA V5 uses a 3 button mouse:

A. Left Mouse Button (Number 1) – used for the selection

B. Middle Mouse Button (Number 2) or the thumb wheel – used for indication or pointing

C. Right Mouse Button (Number 3) – used for the displayed of a contextual menu



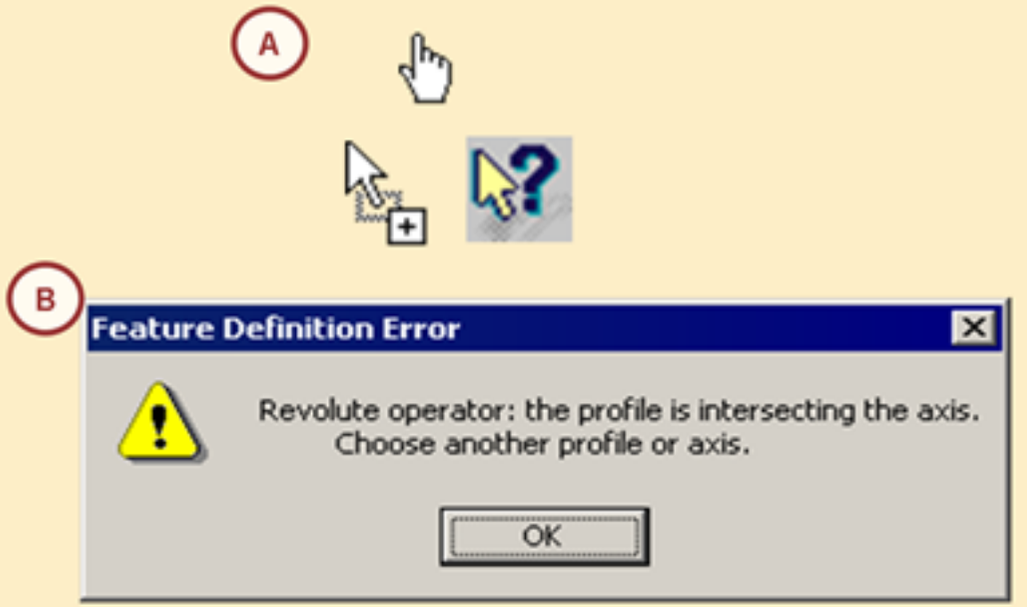
The Windows Philosophy (3/3)

CATIA provides various levels of system feedback to the user

A. Mouse pointer – Indicates different status

B. Message dialog boxes – Convey information about the progress, failure, or result that occurred

C. Tool tips / Help Messages – Tool tips and a short help message is available



Introduction to V5 Documents

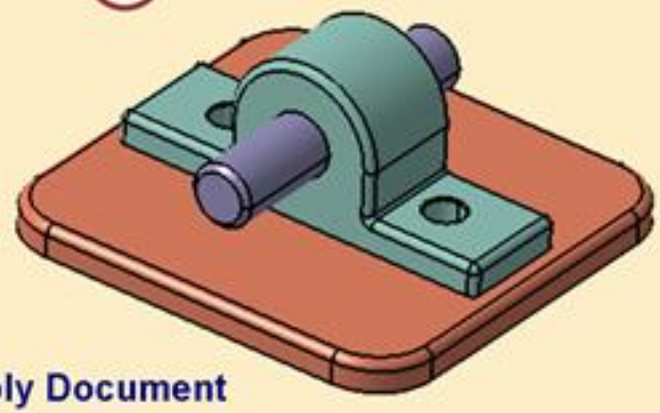
- CATIA has a wide variety of documents that can be created, modified and saved.

A

Part Document

(.CATPart)

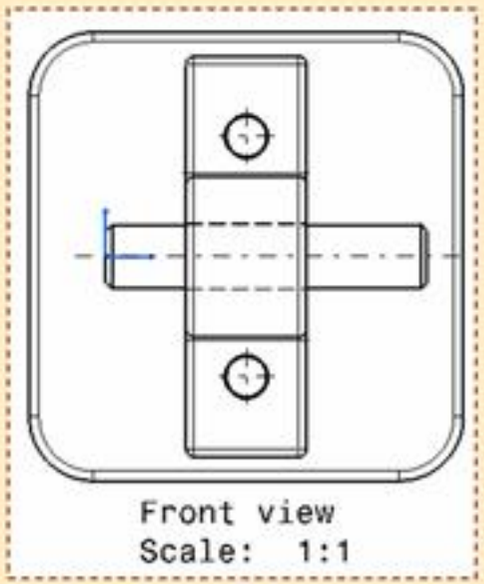
B



Assembly Document

(.CATProduct)

C



Drawing Document

(.CATDrawing)

The Workbench Concept

- CATIA assigns workbenches to document types
- Each workbench contains a set of tools that is dedicated to perform specific tasks:

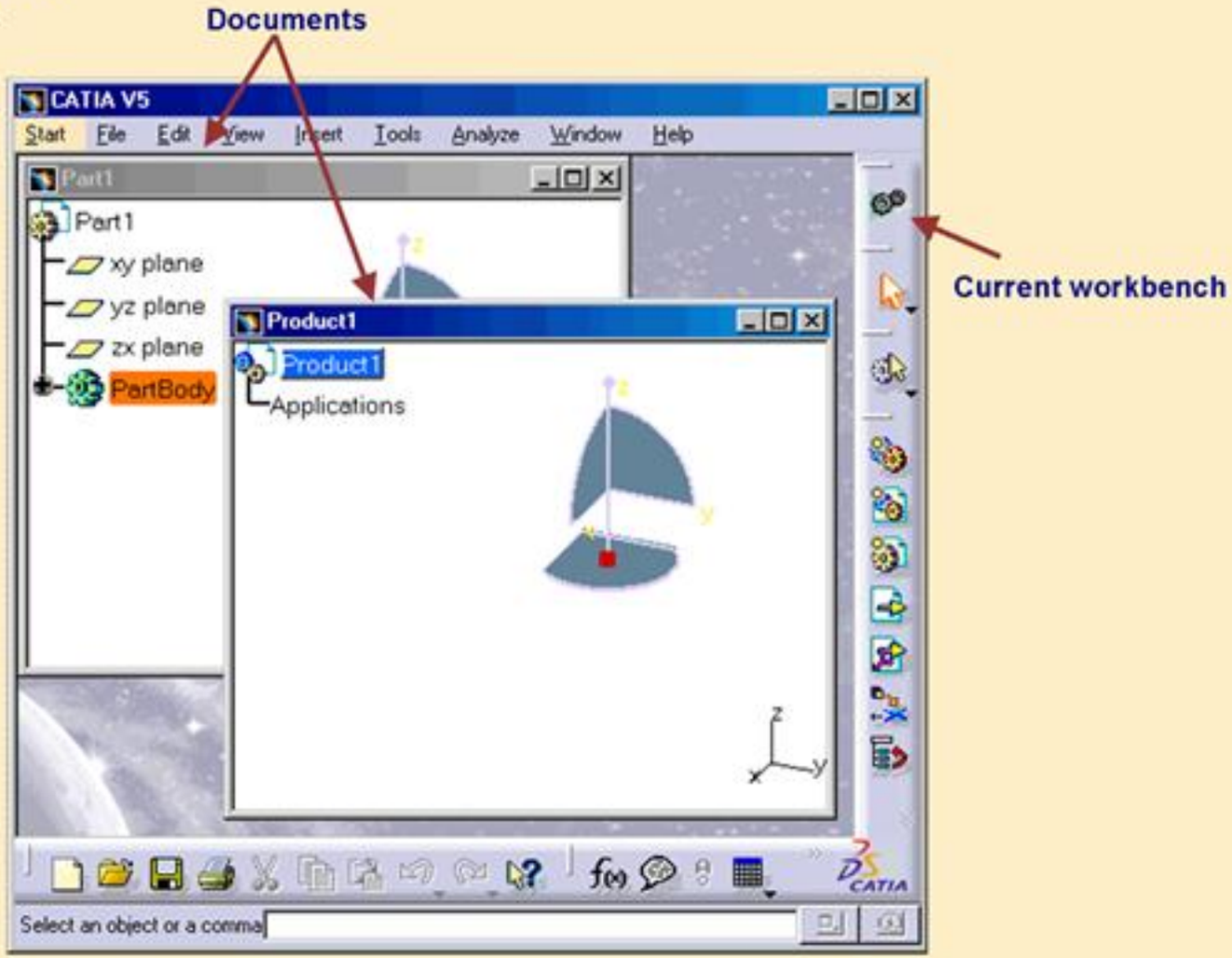

Part Design:


Sketcher:


Wireframe & Surface:


Generative & Interactive Drafting:

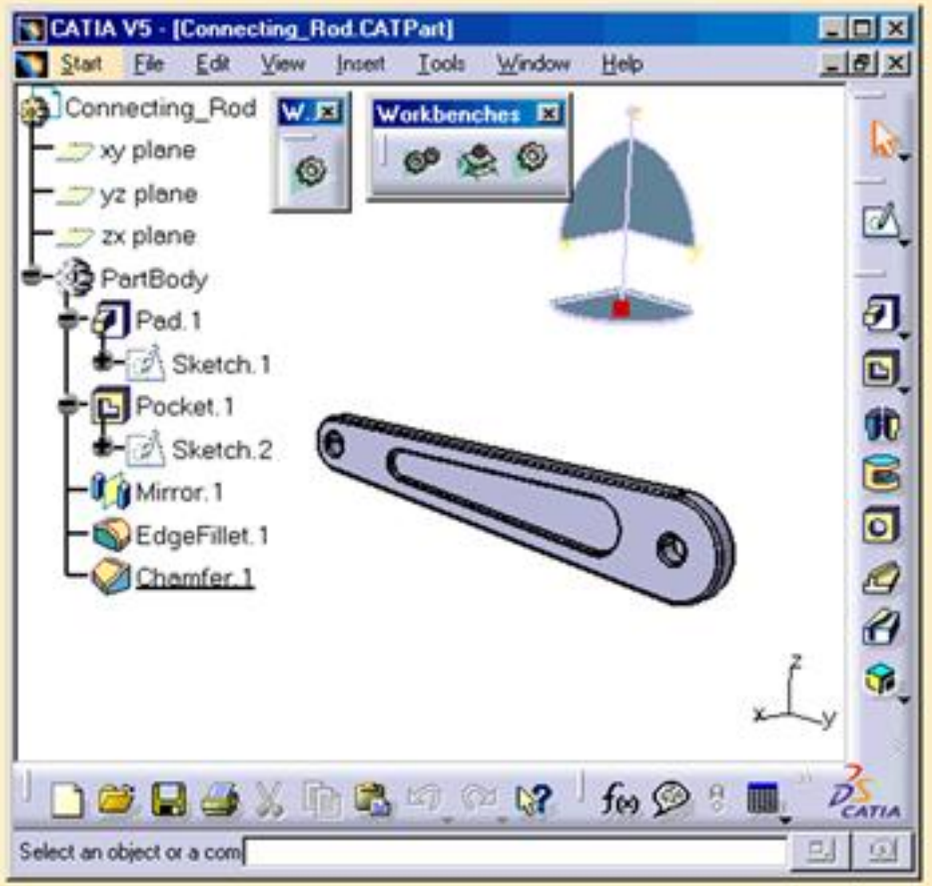

Assembly Design:



CATIA User Interface (1/2)

CATIA's user interface adopts the Windows interface, and contains the following key features.

- Separate workbenches and their respective toolbars
- Easy navigation from one workbench to another
- Standard and specific menus & toolbars (File, Edit, Insert...)
- Standard manipulations (Copy-Paste, Drag-and-Drop, Edit in Place...).
- Intuitive (highlighting, copilot, pointer shapes...).
- Multi-document support
- Contextual menu (MB3) support
- Specification tree, which includes technological features, constraints, and relationships



CATIA User Interface (2/2)

Below is the layout of the elements of the standard CATIA application:

A. Menu Commands

B. Specification Tree

C. Window of the active document

D. Filename and extension of current document

E. Icons to maximize, minimize, and close the window

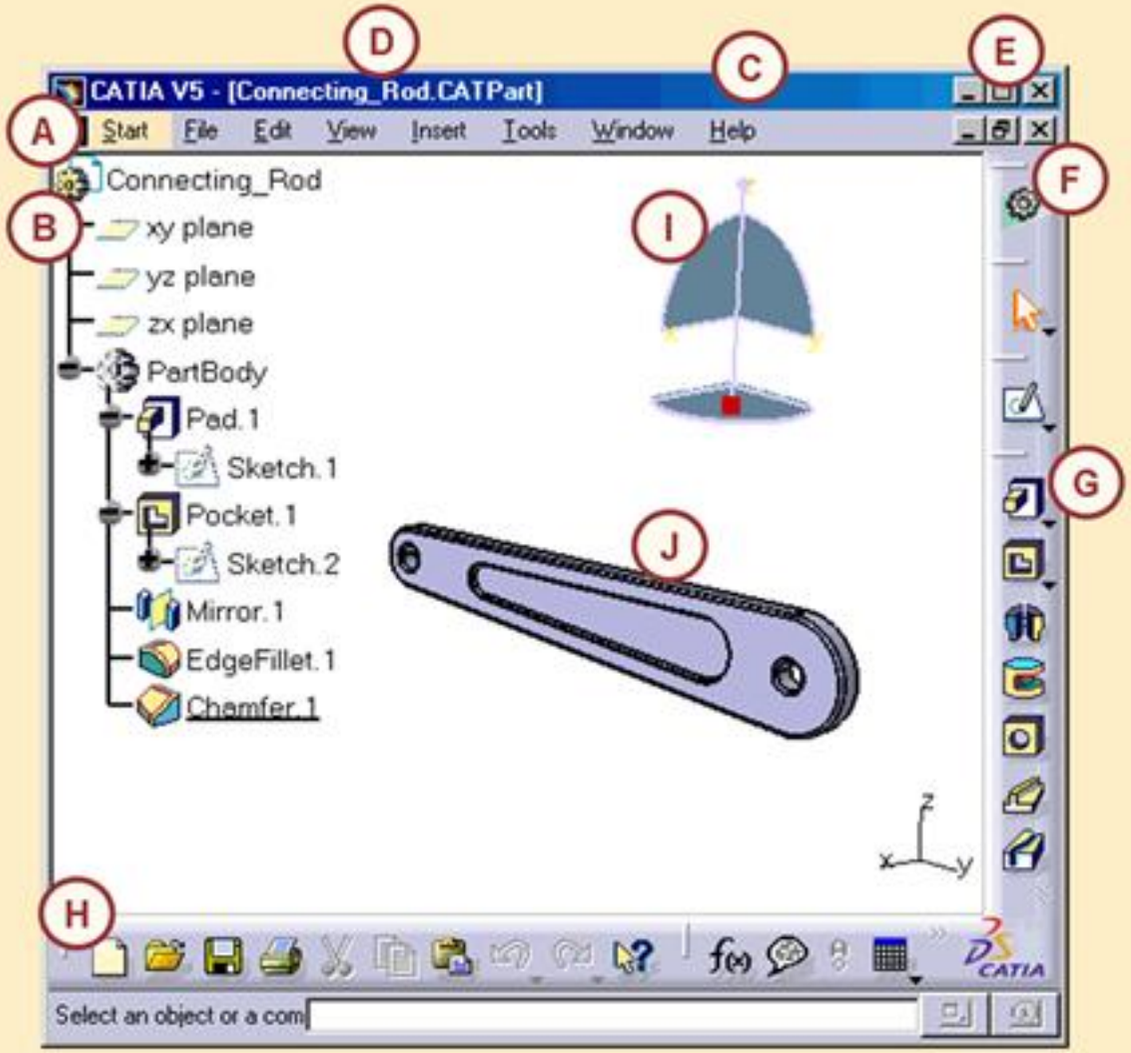
F. Icon of the active workbench

G. Toolbars specific to the active workbench

H. Standard toolbar

I. Compass

J. Geometry area

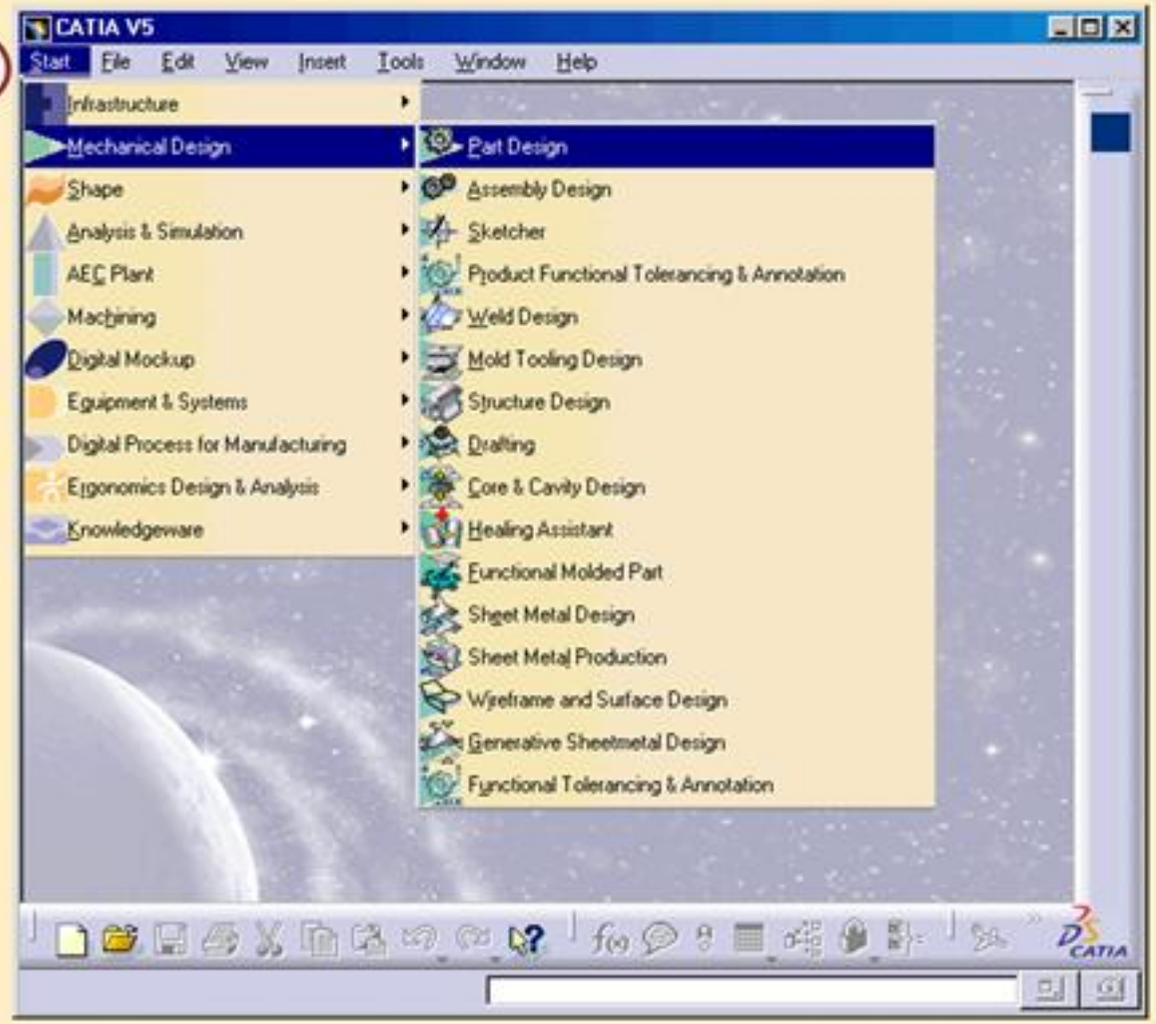


Workbenches (1/2)

Workbenches contain various tools that you may need to access while creating a part. You can switch between any primary workbench using the following two ways:

A. Use the **Start** menu to open the required workbench.

B. Click **File > New** to create a new document of a particular file type. The associated workbench automatically opens.



Workbenches (2/2)

You can create an direct access to a favorite workbench in the Start menu as follows:

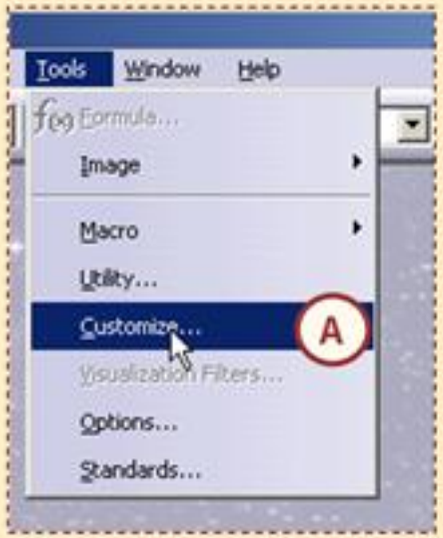
A. Click **Tools > Customize** to access the dialog box.

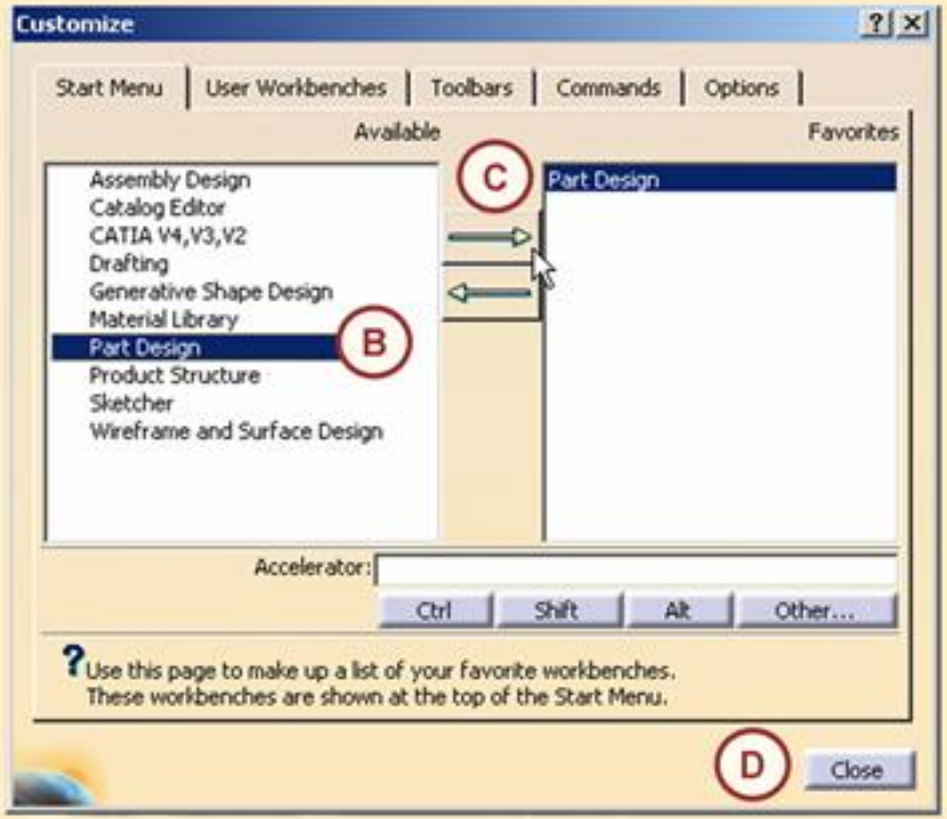
B. Using the Start Menu tab, select the workbench from the list.

C. Select the arrow to add the workbench to the Favorites list.

D. Click Close to exit.

E. The workbench will be displayed when the **Start** menu is next selected.

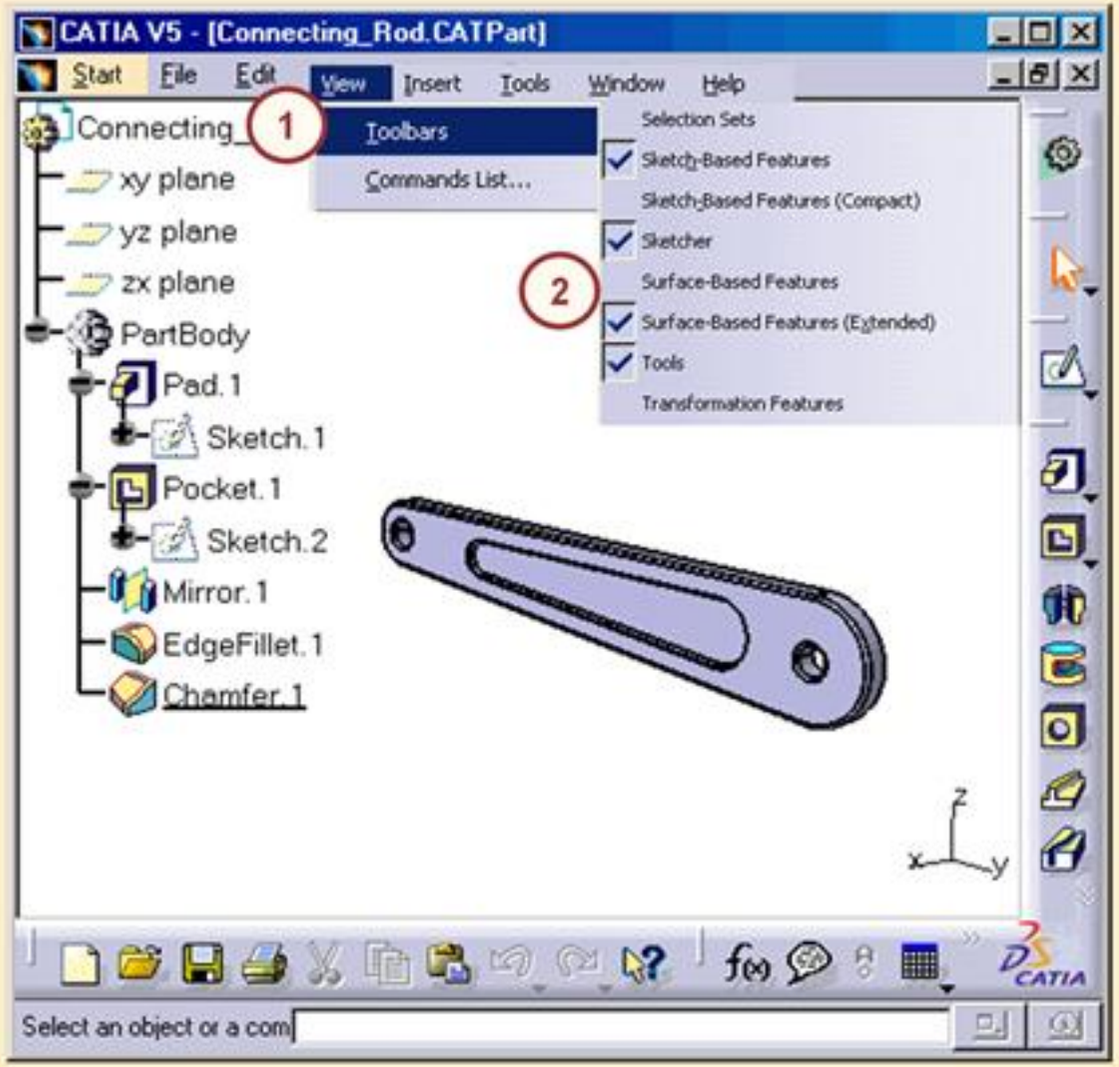




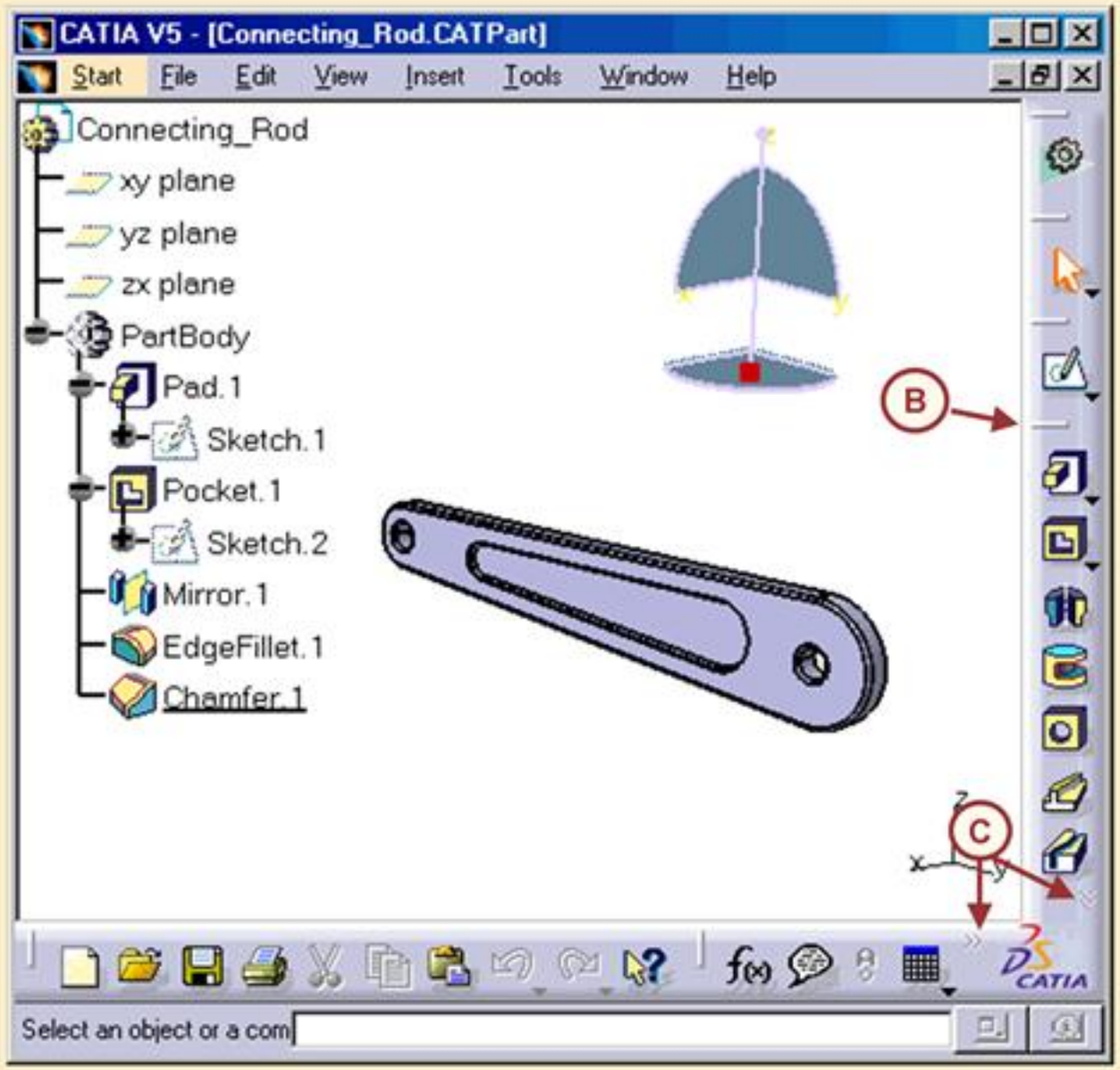
Menus and Toolbars (1/2)

To activate/deactivate a toolbar use the following steps:

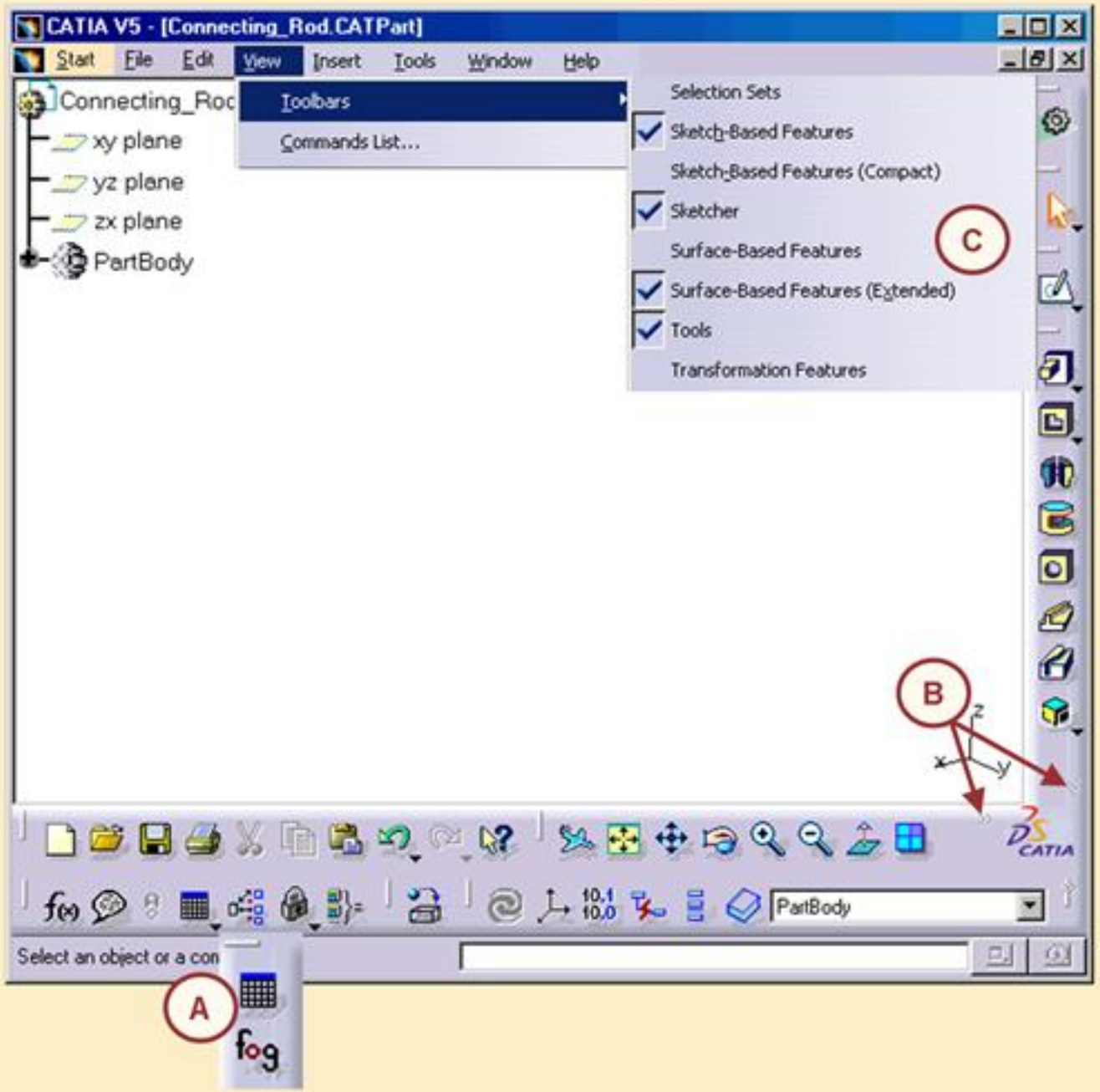
1. Click **View > Toolbars** menu.
2. A complete list of toolbars is shown with respect to the current workbench.
3. Activated toolbars have a checkmark beside them.
4. Select a particular toolbar to activate or deactivate the required toolbar.



Menus and Toolbars (2/2)

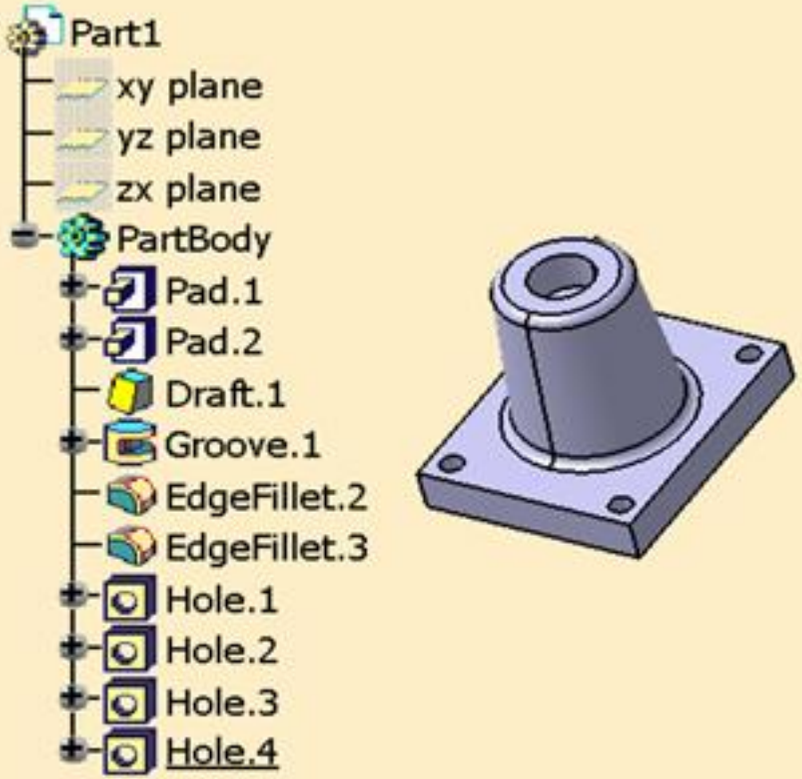


What to Do When a Tool Cannot be Found



The Specification Tree

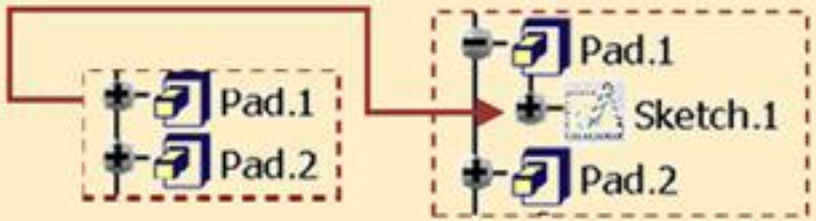
- CATIA V5 provides a specification tree, which keeps the hierarchy of features, constraints, process, and assembly information for a CATIA document.
- Can edit, re-order or removed specifications.
- The specification tree can suppress certain features, and information by temporarily removing them from consideration for the model.



Manipulating the Specification Tree

CATIA allows you to manipulate the specification tree in a manner similar to the geometric area.
Refer to the list below for more information:

Manipulation	Action	Description
Hide the tree	F3	Press the <F3> key to hide or show the tree.
Activate/Deactivate the tree	Or Shift+ F3	Click anywhere on the tree structure or press <Shift> and <F3> simultaneously to activate or deactivate the tree.
Move the tree		Drag and drop the tree structure using the left mouse button.
Expand/Collapse a node of the tree		Click on the [+] sign to expand the node, and on the [–] sign to collapse it.



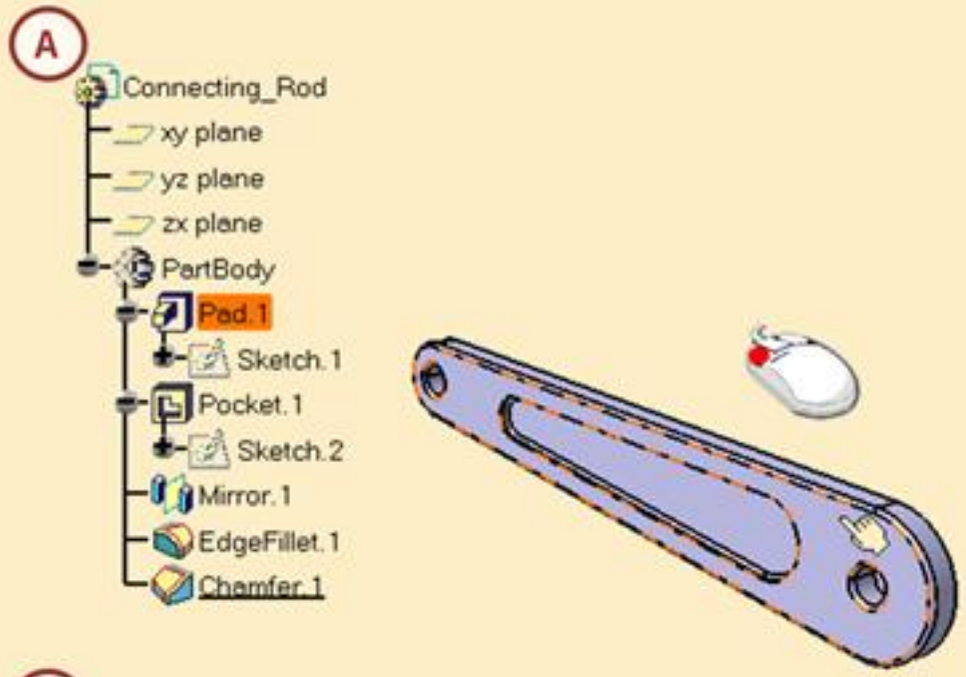
Selecting Objects With the Mouse

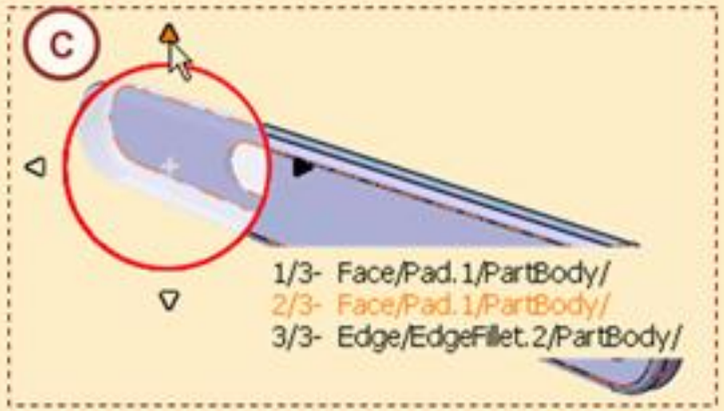
Ways to Select Objects:

A. Simple Selection

B. Multi-Selection

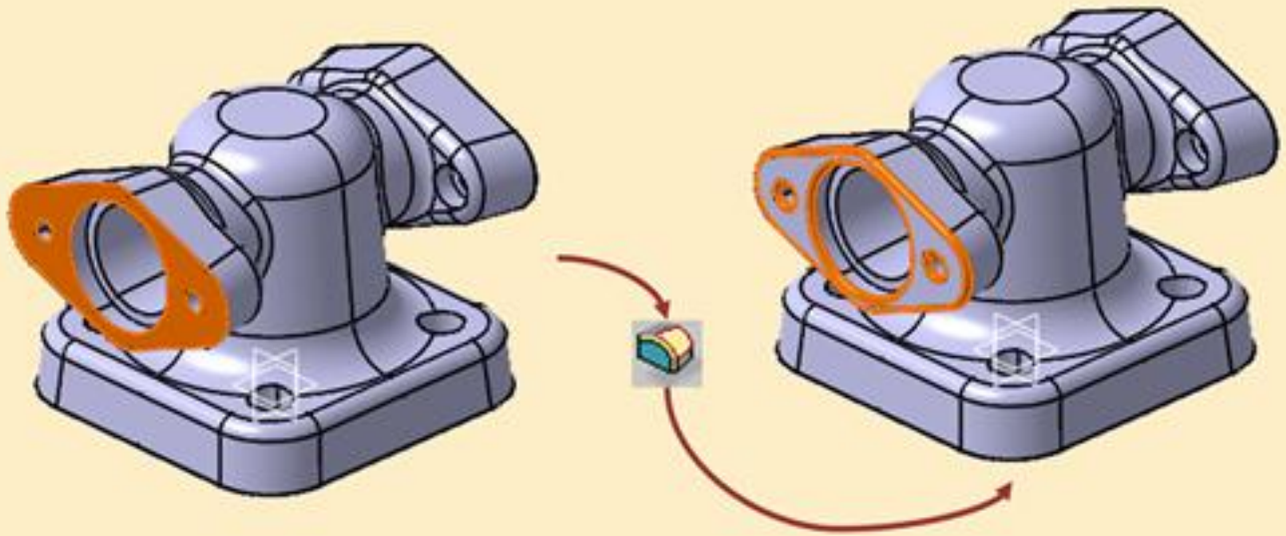
C. The Pre-selection Navigator



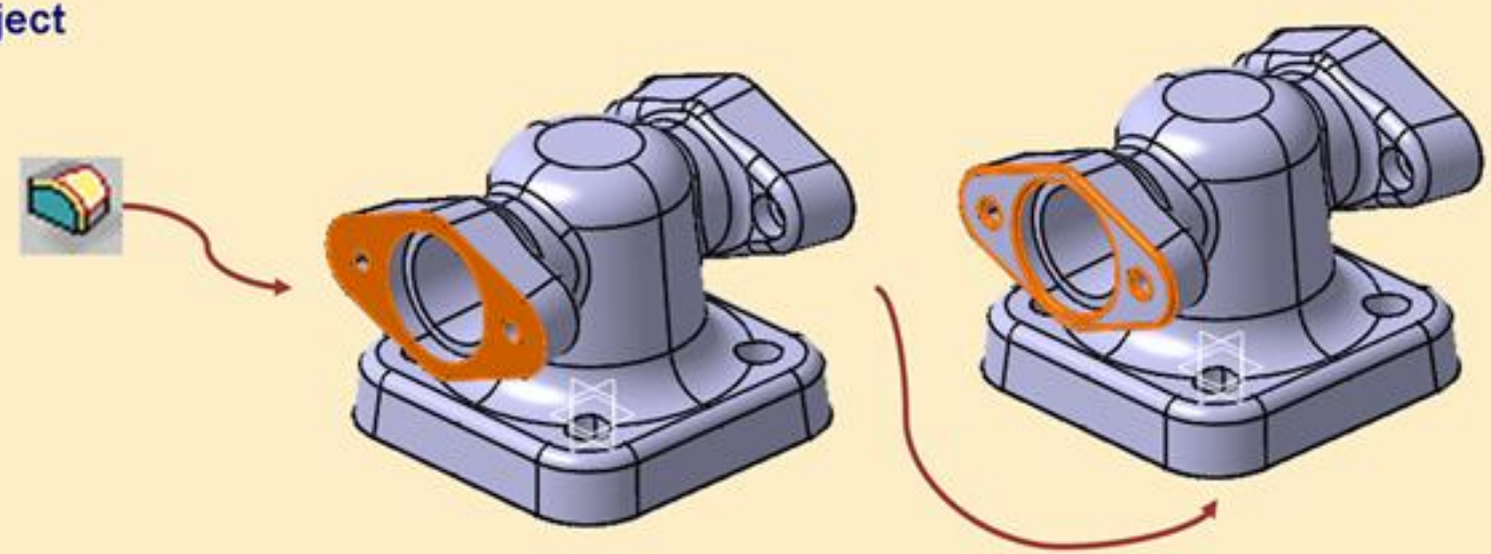


The Object/Action and Action/Object Approaches

Object / Action



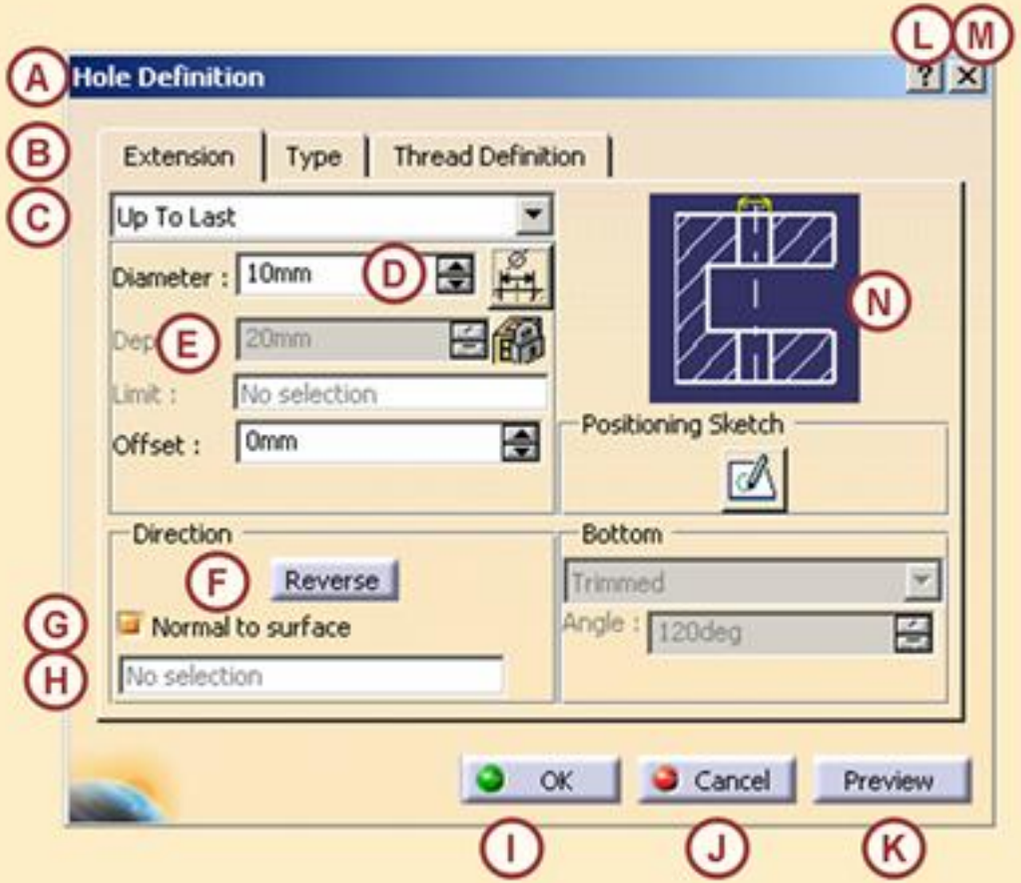
Action / Object



Using Dialog Boxes

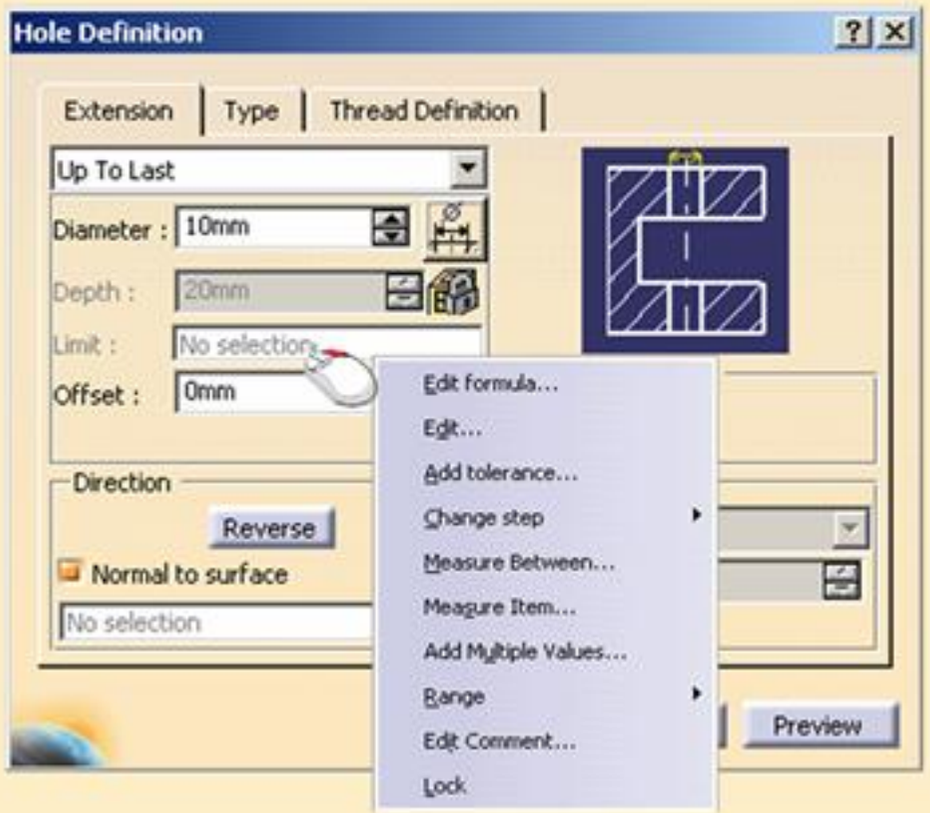
Dialog boxes provide parameters for the definition of features:

A. Name
B. Option Tabs
C. Drop down list
D. Increment arrows
E. Locked fields
F. Buttons
G. Check boxes
H. Inactive fields
I. OK button
J. Cancel button
K. Preview button
L. Help
M. Close
N. Visual Aid Assistants



Using Dialog Boxes and Right-click

Pressing the **right** mouse button (*right-clicking*) in a field



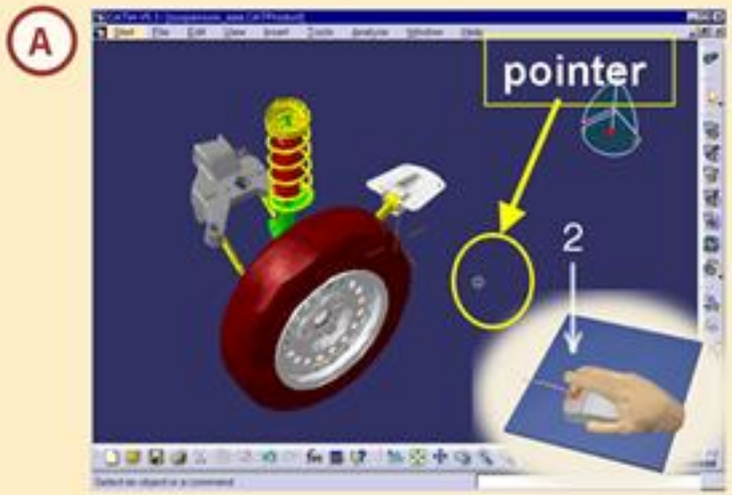
Moving Objects With the Mouse

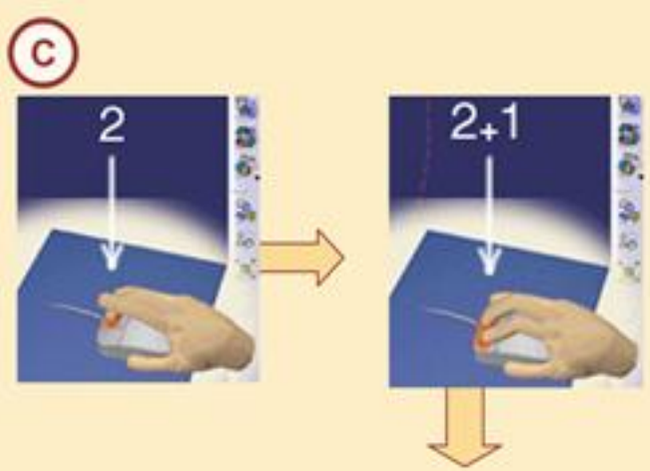
Ways of viewing the model geometry:

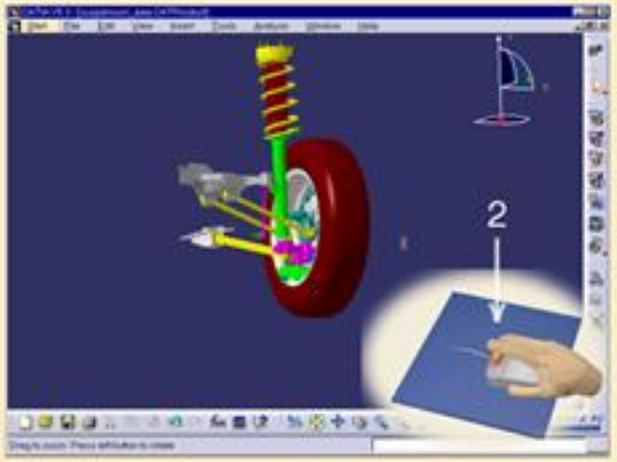
A. Panning

B. Rotating

C. Zooming

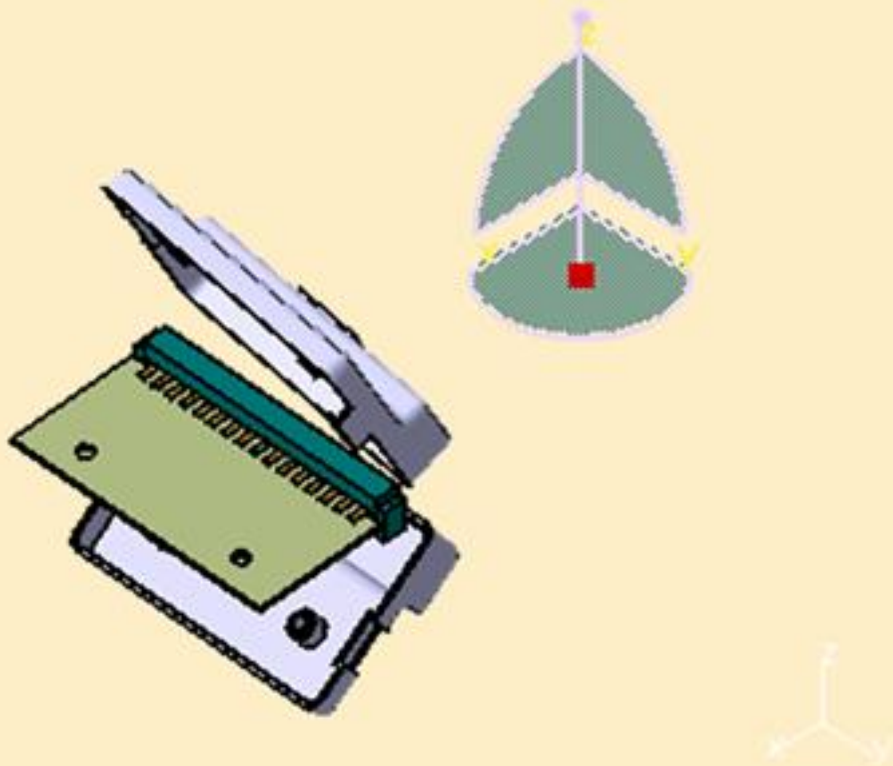






What is the Compass?

- The compass is used to physically move and manipulate objects
- It is useful within the Assembly Design, Freestyle, and Digital Mockup workbenches
- Base of the *Compass* (or privileged plane) Default is XY plane
- The default orientation of the Compass is parallel to the reference XYZ axis system
- Compass will update to reflect the new viewing angle/direction



Graphic Properties

The Graphic Properties toolbar enables you to change various graphical properties of elements displayed on the screen.
It provides tools to change the following properties:

A. Fill color

B. Transparency

C. Line thickness

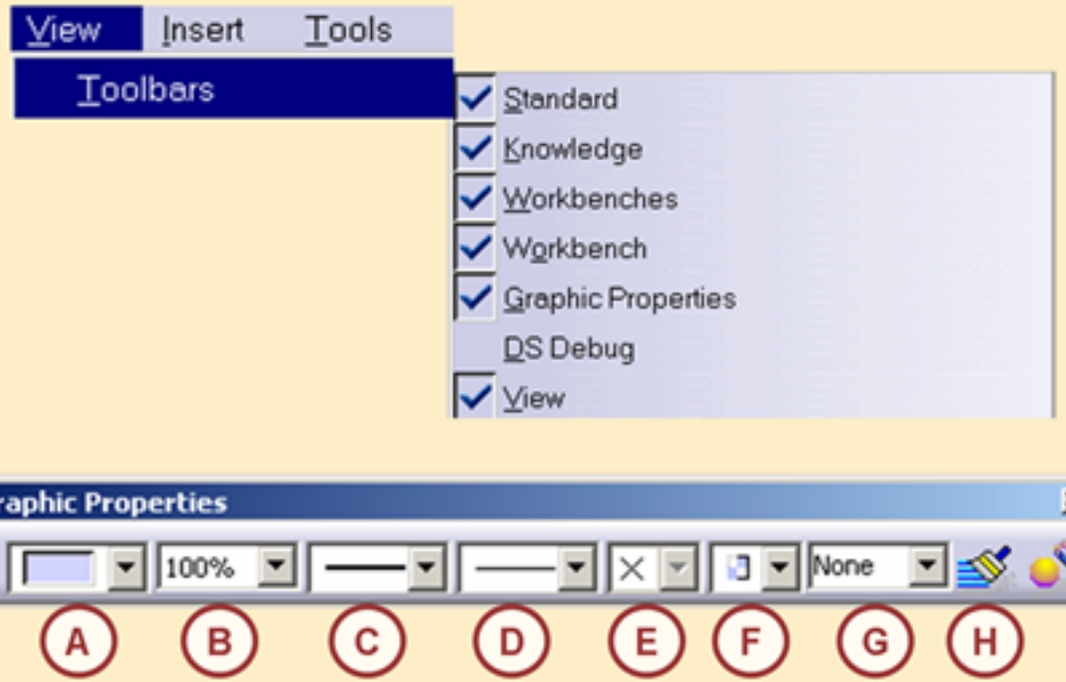
D. Line type

E. Point symbol

F. Rendering style

G. Active Layer

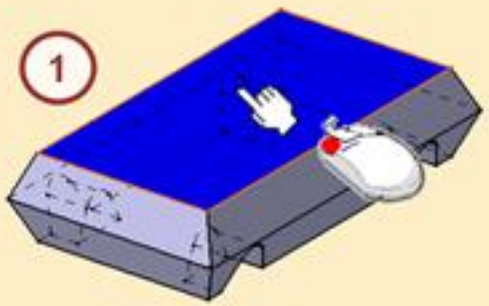
H. The Painter tool copies the graphic properties of one feature to another

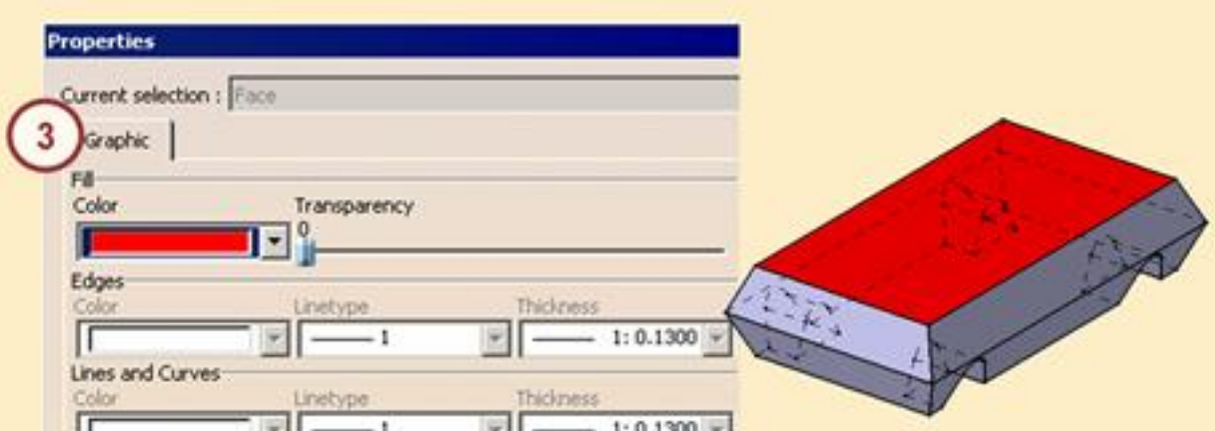


Changing the Graphic Properties

Use the following steps to change the graphic properties of an element:

1. Select the element(s) using the left mouse button.
2. Right-click and select **Properties** in the contextual menu.
3. Select the Graphic tab, and change the required graphic properties.

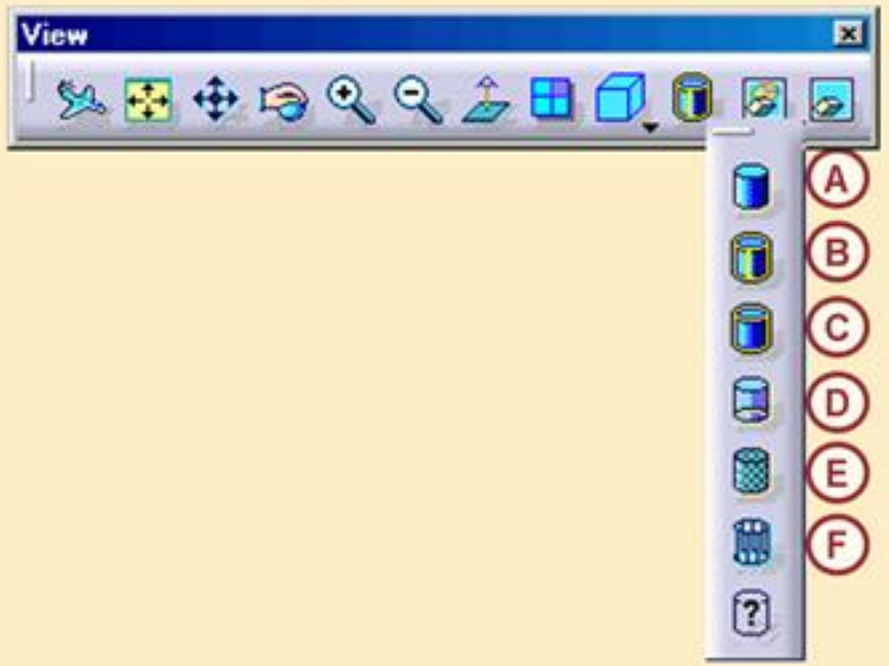


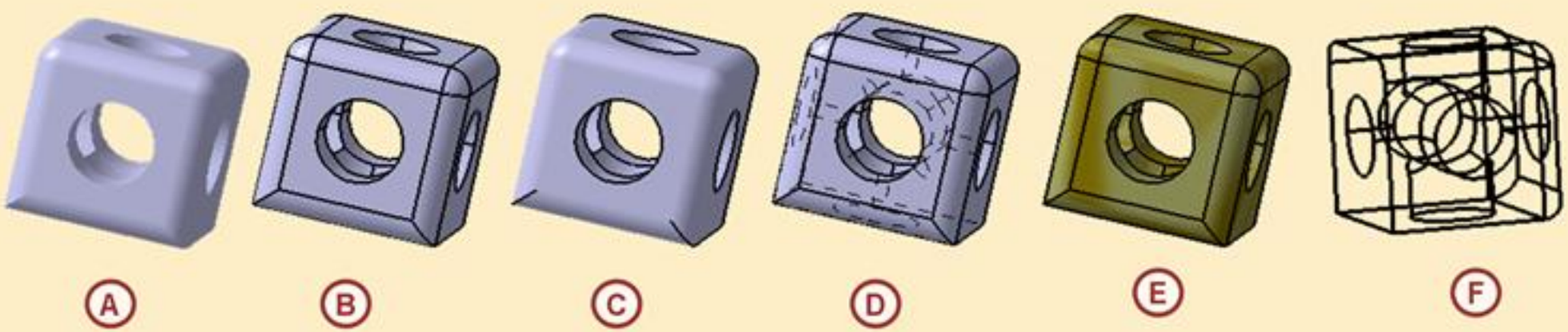


Rendering Styles

The View toolbar contains the rendering styles of:

A. Shading (SHD)
B. Shading with Edges
C. Shading with Edges without smooth Edges
D. Shading with Edges with Hidden edges
E. Shading with Material
F. Wireframe (NHR)



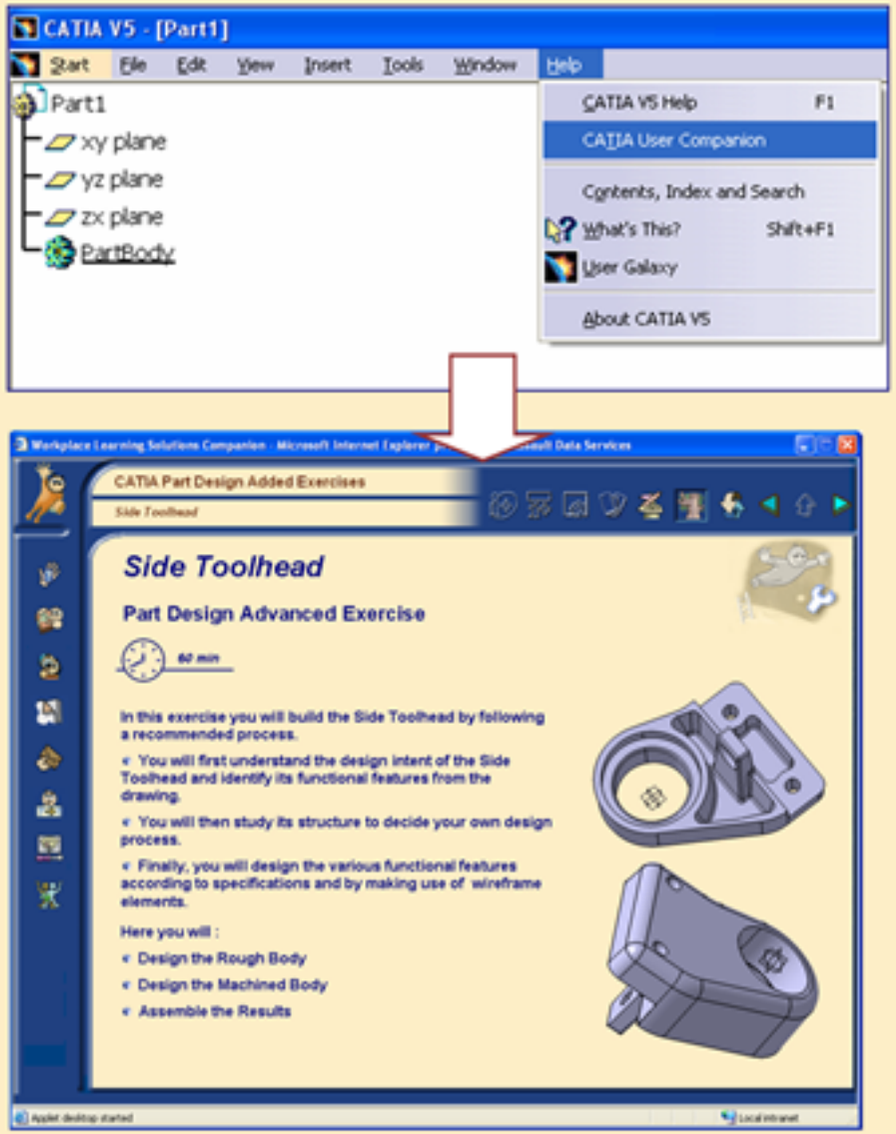


CATIA User Companion

CATIA contains many assets to help you learn the finer details of all the functionalities and tools. One such asset that focuses on workplace learning is the User Companion.

The Companion is a self-contained series of learning objects, that aims to allow you to learn at your own pace, to get some additional knowledge or some extra practice on the following topics:

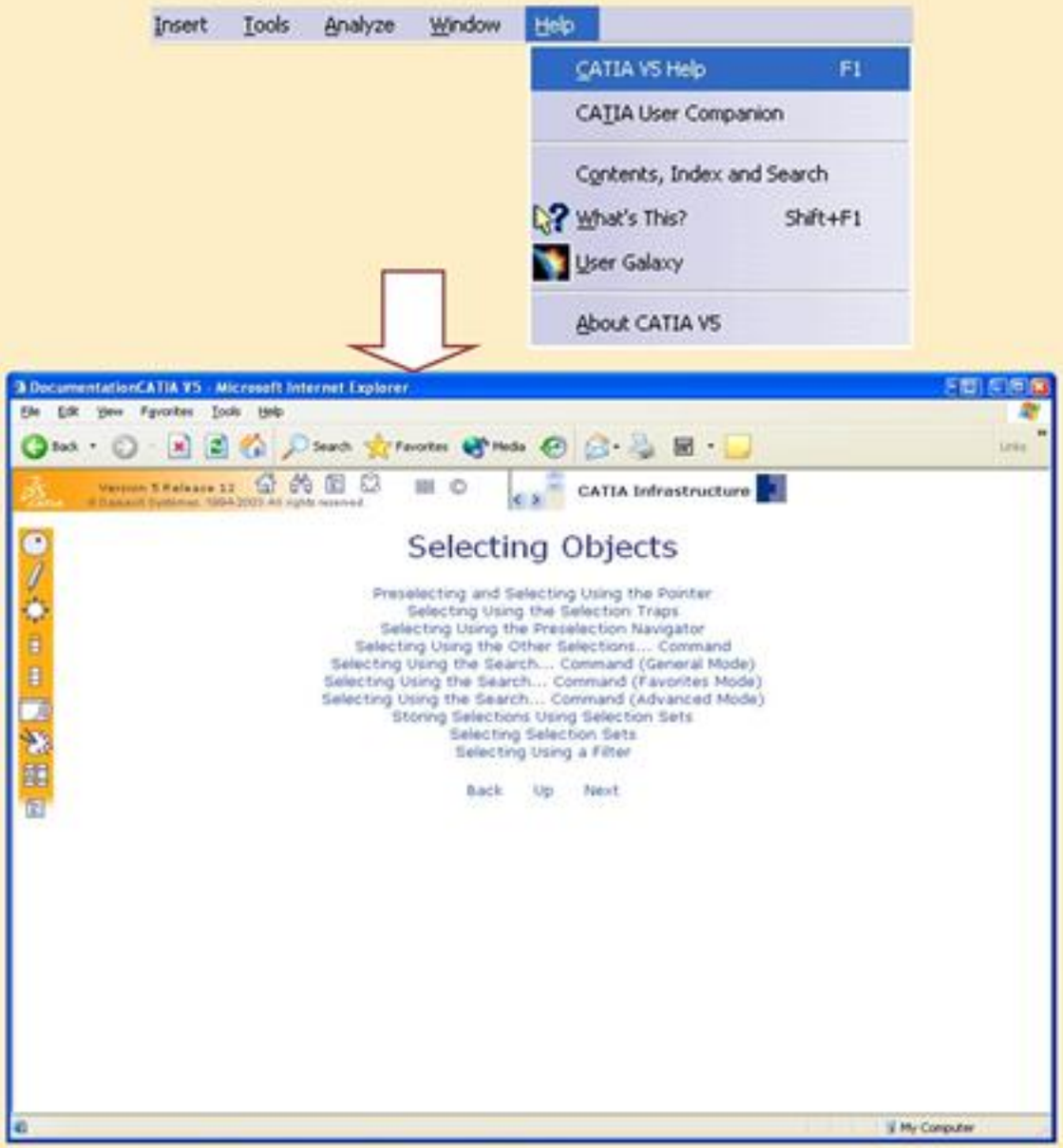
- CATIA V5 (Mechanical Design, Hybrid Design, Generative Sheet Metal Design, Analysis)
- DMU
- ENOVIA
- SMARTEAM



Help Documentation

Another asset that focuses on providing information is the online Help Documentation.

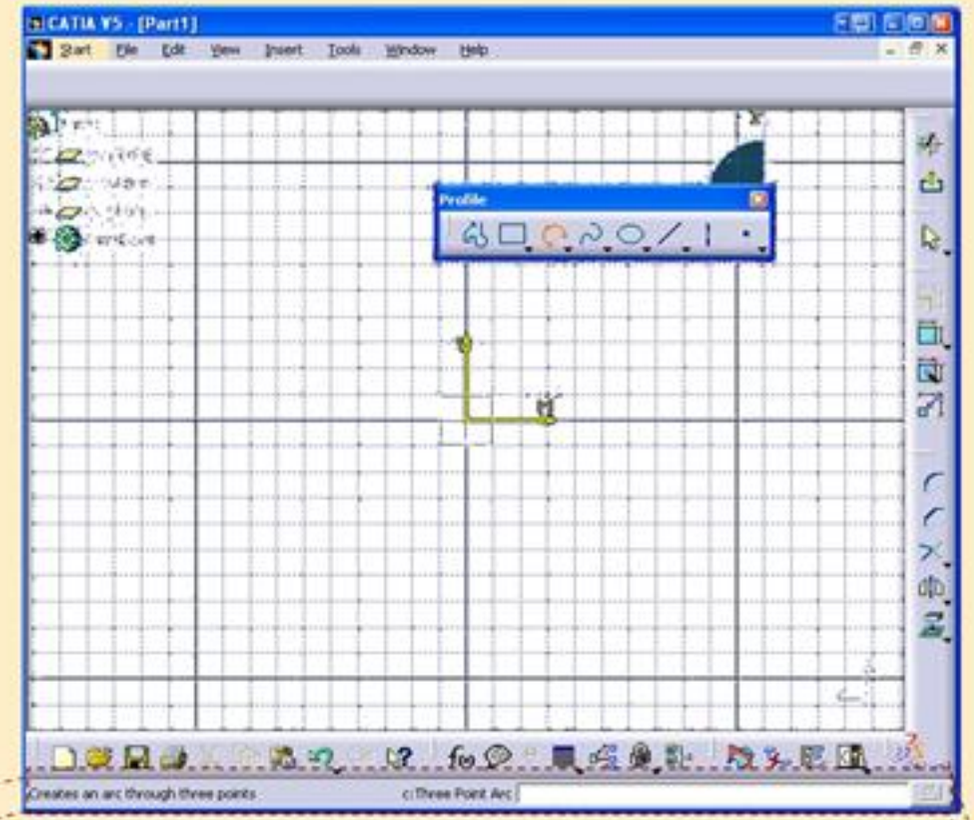
The help system is a self-contained series of HTML documents that break down the vast CATIA information into workbenches, products, and solutions.

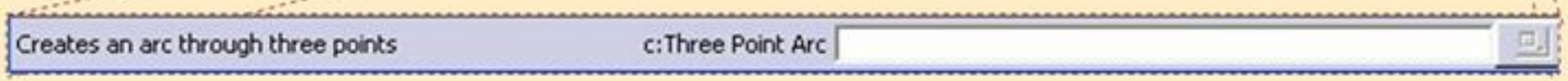


Message Bar

Another useful tool that can helps you to determine what is required, when trying to perform a command, is the Message Bar.

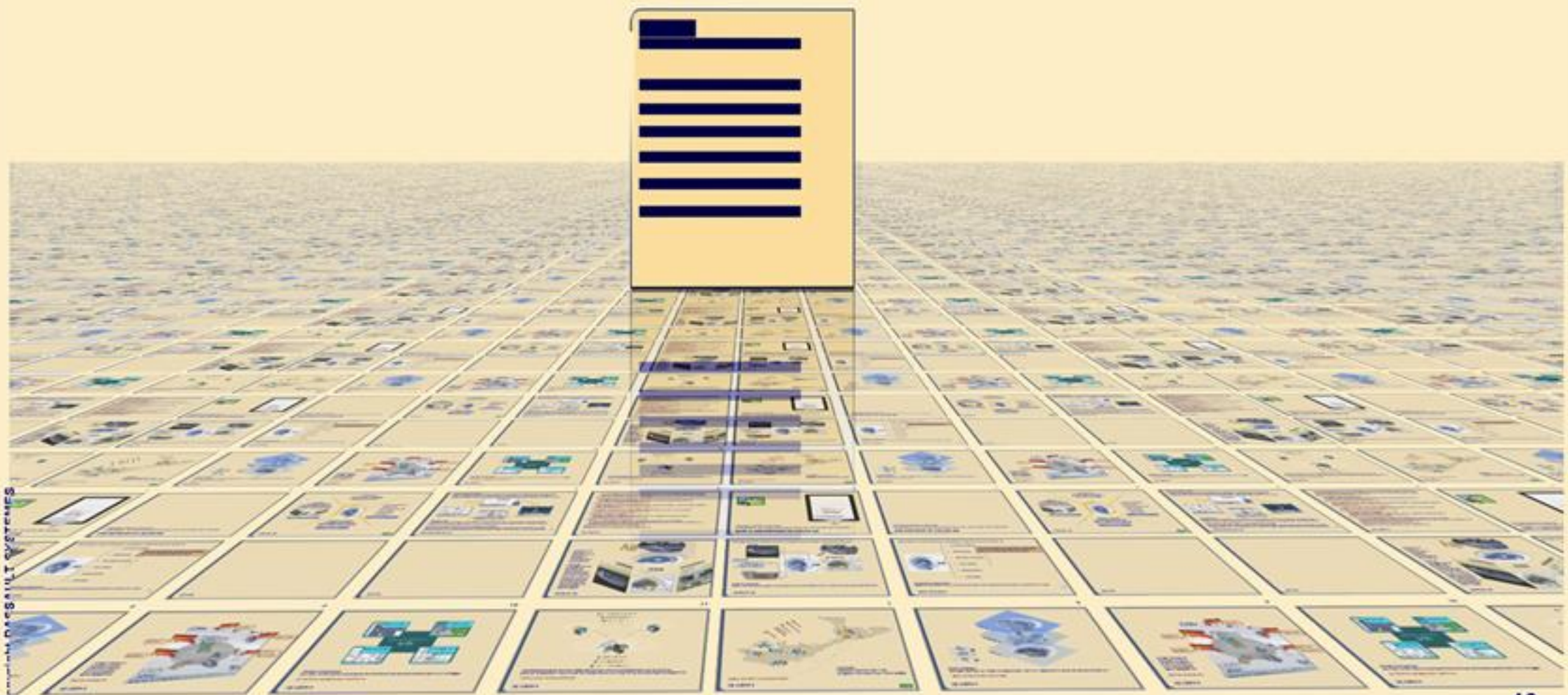
When you select a tool, CATIA uses the message bar to prompt you for the particular inputs that are needed to complete a command.





To Sum Up

In the following slides you will find a summary of the topics covered in this lesson.



Understand the CATIA software

CATIA is a mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations to capture the design intent.

- ✓ CATIA acts as the backbone for concept, product definition, manufacturing, simulation, and after-market information found within various lifecycle stages of a product.
- ✓ It provides the specifications and geometrical data related to a product across several lifecycle phases.

Open CATIA

In a Windows environment, you can start the CATIA application in several ways:

A. Select CATIA from the Start > Programs > CATIA menu.
B. Double-click the CATIA icon on your Windows desktop.
C. Double-click on an existing CATIA document.

CATIA's Scope From Concept To Realization

Understand the CATIA interface

CATIA V5 is specifically designed for the Windows operating environment, and it behaves in the same manner as other Windows applications. Traditional menus provide access to all the CATIA commands. Toolbars contain icons for quick access to the most frequently used commands.

CATIA's user interface adopts the Windows interface, and contains the following key features:

A. Separate workbenches and their respective toolbars.

B. Easy navigation from one workbench to another.

C. Standard and specific menus & toolbars (File, Edit, Insert...).

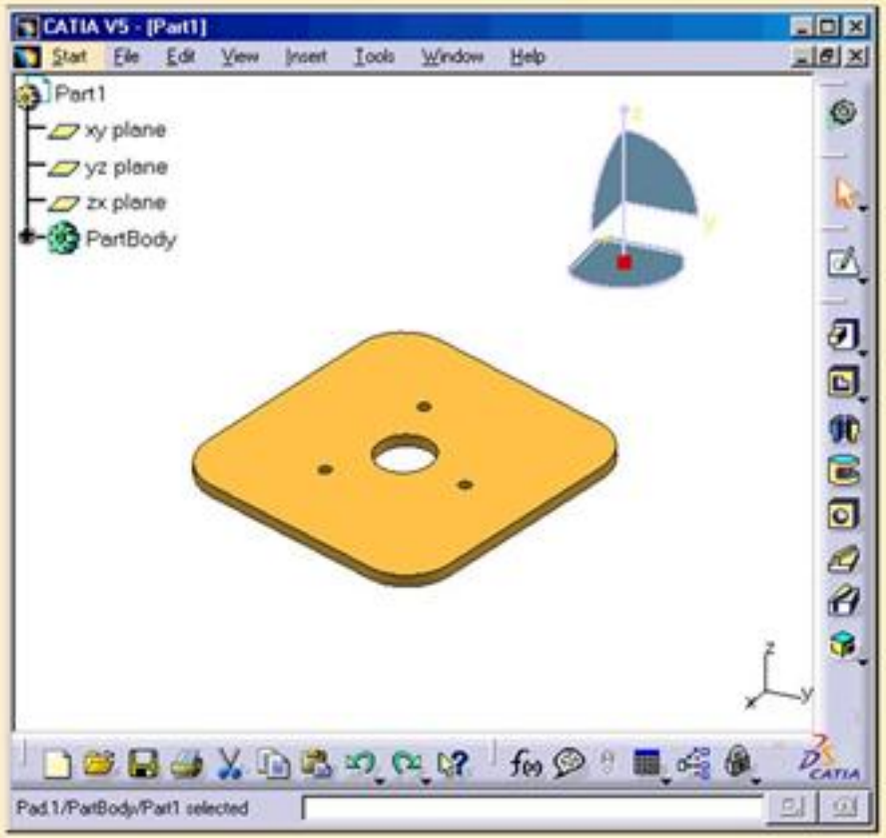
D. Standard manipulations (Copy-Paste, Drag-and-Drop, Edit in Place...).

E. Intuitive (highlighting, copilot, pointer shapes...).

F. Multi-document support.

G. Contextual menu (MB3) support.

H. Specification tree, which includes technological features, constraints, and relationships.

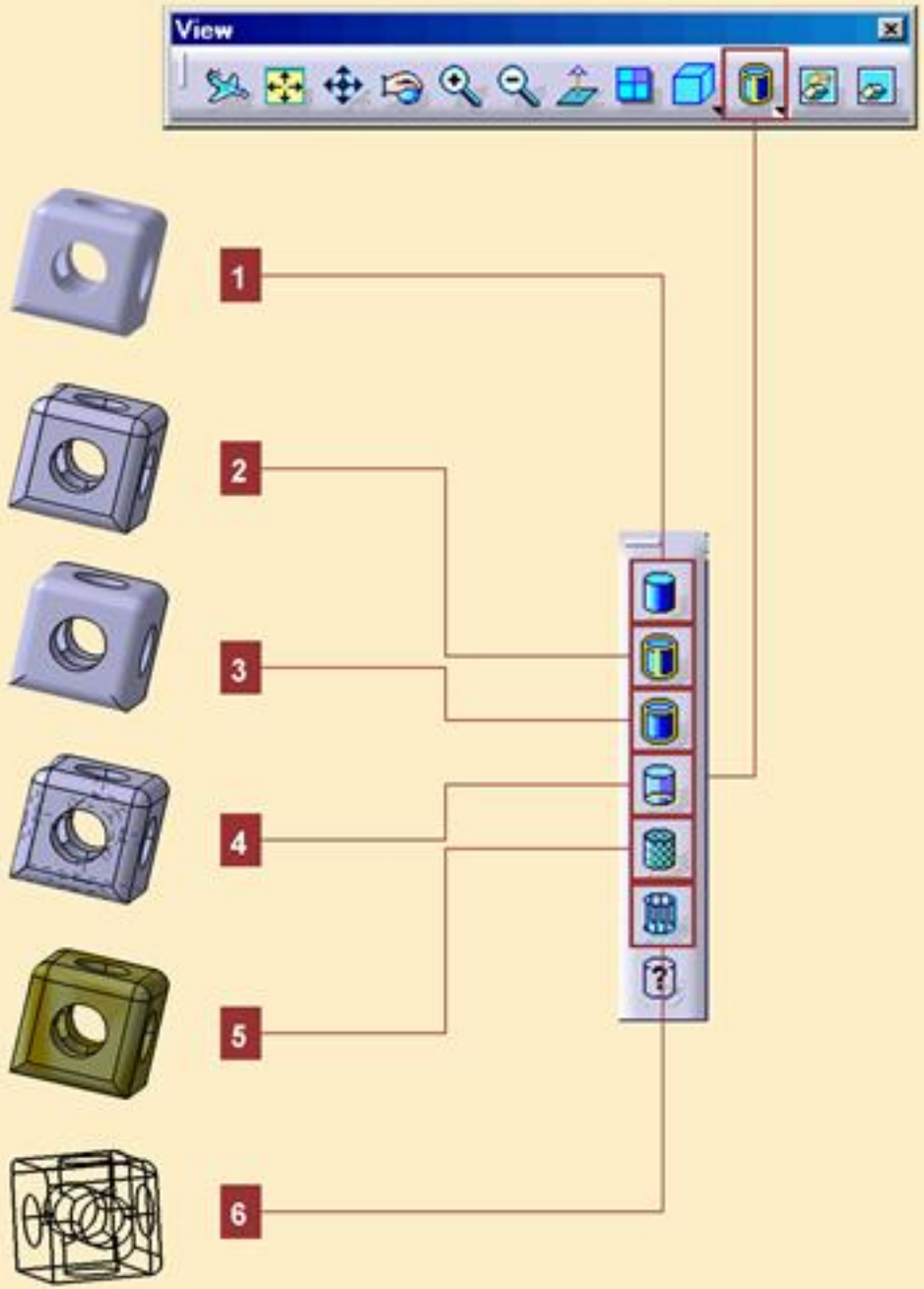


View Tools

Rendering Styles

CATIA has the ability to apply different styles of rendering to visualize the geometry and provide more clarity to the model.

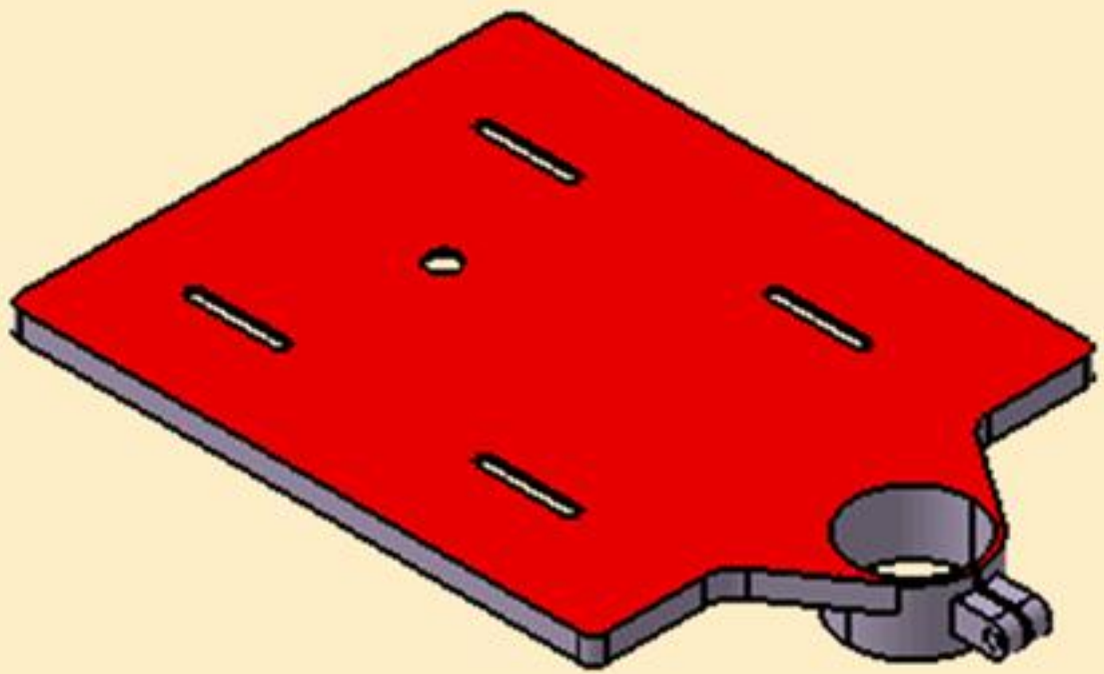
1. Shading (SHD)
2. Shading with Edges
3. Shading with Edges without smooth Edges
4. Shading with Edges with Hidden edges
5. Shading with Material
6. Wireframe (NHR)



Exercise Overview

You will practice what you have learned by working through an exercise.

CATIA V5 User Interface

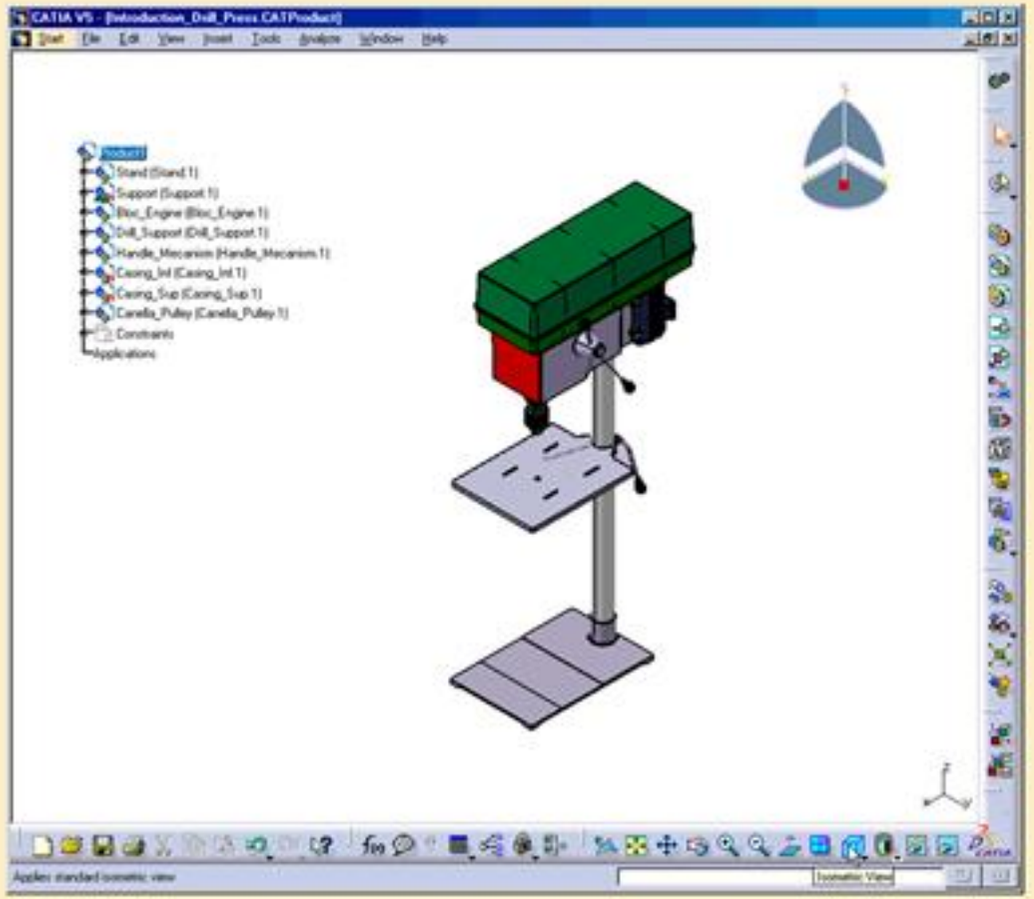


Case Study: Introduction to CATIA

In this case study you will review the master project assembly.

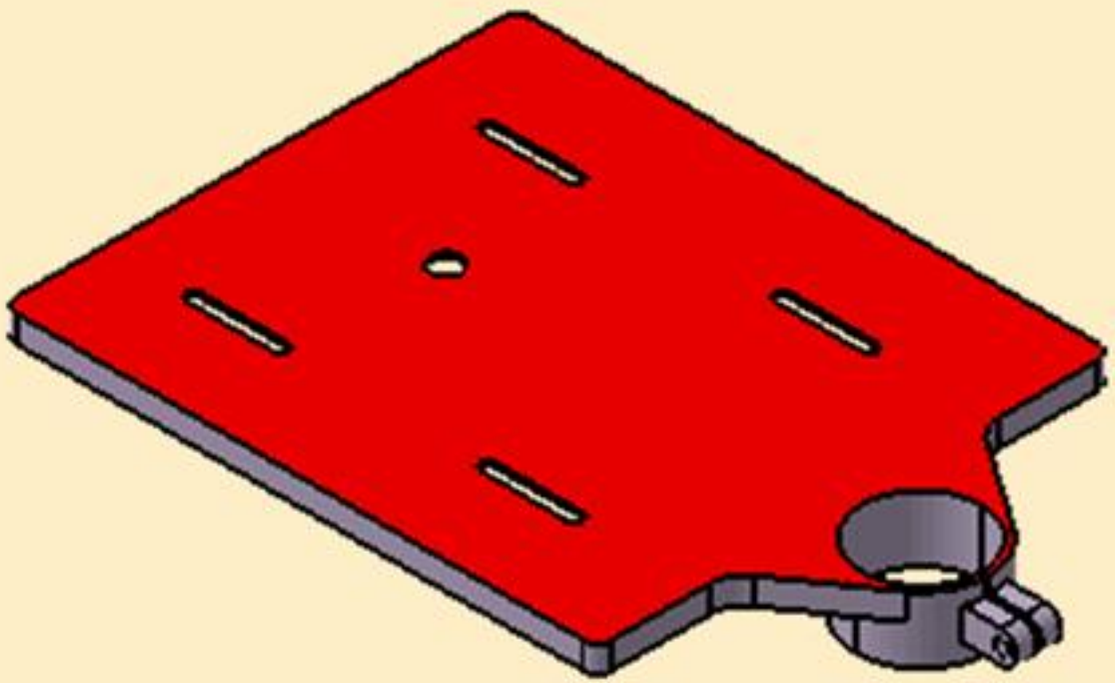
With the knowledge gained in this lesson, you should be able to:

- ✓ Change the orientation of a model.
- ✓ Change the visualization properties of a model.
- ✓ Manipulate the Specification Tree.
- ✓ Access the help utilities of CATIA



CATIA V5 User Interface: Recap

- ✓ Start CATIA
- ✓ Open an existing Document
- ✓ Identify the fundamental document types of CATIA
- ✓ Close the Document



Case Study: Introduction to CATIA Recap

- ✓ Change the orientation of a model
- ✓ Change the visualization properties of a model
- ✓ Manipulate the Specification Tree
- ✓ Access the help utilities of CATIA