

About the CD

Bound inside the front cover is a CD containing all solid models necessary to perform the exercises. For example, the lesson on creating assemblies does not require the user to build all the parts first. The user will be able to download the models onto their own PC or workstation and then perform the exercise with them.

CATIA Version 5 Product Overview

CATIA V5 is adaptable for all customers in the engineering and manufacturing industries, especially small to mid-size companies. CATIA Version 5 has a comprehensive set of products for the integration of your processes. It is delivered on two fully scalable platforms called P1 and P2. With Platform P1, CATIA is targeted to small and medium sized process oriented companies who want to grow towards large scale digital product definition. Platform P2 is targeted at customers engaged in advanced design engineering who require Product, Process and Resource modeling.

CATIA is the flagship CAD/CAM/CAE/PDM system from Dassault Systemes and has already been chosen by more than 13,000 customers, which represent 127,000 seats.

CATIA lets you create and simulate your entire product life cycle, from preliminary concept through design, analysis & simulation to manufacture and maintenance. Thanks to this overall process integration and product simulation, you reduce requirements for physical mock-ups, shorten production lead times, cut costs and improve quality of the finished products.

CATIA Version 5 was introduced in March, 1999 with a new generation scalable architecture and is the ideal tool to ease and improve communication and collaboration at all levels of the company.

CATIA Version 5 offers solutions tailored to the needs of small and medium sized enterprises as well as large industrial corporations in all industries, specially in Fabrication & Assembly, Consumer Goods, and

Electrical & Electronics goods. CATIA Version 5 product portfolio covers:

- **Mechanical Design Solution:** provides products for specification driven modeling for Solid, Hybrid and Sheetmetal Part design, Assembly design and integrated Drafting.
- **Analysis Solution:** offers products for designer-oriented part & assembly stress and vibration analysis for early pre-validation.
- **Shape Design & Styling Solution:** provides products to create, control and modify mechanical & freeform surfaces.
- **Equipment and Systems Engineering Solution:** integrates and exchanges electrical product design information during the product definition.
- **Product Synthesis Solution:** provides advanced digital mockup review and simulation functions.
- **Infrastructure Solution:** includes interfaces with most common standard formats.

Plus:

- A Knowledge Advisor that assists you in making correct decisions and reaching optimum, error-free designs in less time.
- The V4 integration allows you to support combined CATIA V4 and V5 environments.

P2 Platform

CATIA P2 provides an environment for process-centric customers by supporting their end-to-end product lifecycle from concept to production. P2 Platform provides a set of solutions based on knowledge engineering and hybrid modeling technology.

The P2 Platform enables knowledge driven design that captures and reuses a company's intellectual capital to support engineering intuition, creativity

and innovation while allowing engineers to rapidly iterate error-free design alternatives.

P2 Platform provides for an innovative "3D Windows" user interface that delivers increased productivity.

The P2 platform is available on Windows NT and UNIX and includes the following product portfolio:

1. Defines and manages large and hierarchical assembly structures from either a top-down or a bottom-up approach:
 - With simple mouse movements you can drag and snap parts into assembly position
 - Mechanical constraints can be established and adjusted to maintain the position of the parts and established contacts
 - Analysis functions can be used to detect collisions and defined clearance limitations
 - Parts can be reused in the same assembly or in different assemblies without data duplication
 - Bill of Materials report generation ensures full accounting of all components regardless of the complexity of the assembly
 - The export/import of data in a STEP formatted files

2. CATIA Wireframe and Surface - This product addresses solid-based hybrid modeling:
 - Create wireframe construction elements during preliminary design
 - Create wireframe features and basic surface features in the 3D mechanical part
 - Ability to capture and re-use design methodologies and specifications

3. CATIA Generative Drafting - Automatically creates associative drawings from 3D mechanical designs and assemblies:
 - The generation of multiple-view drawings
 - 3D dimensions can be automatically generated in the drawings

- Add post-generation annotations and marks
-
- Associativity of the drawings to the 3D master representation
- The export of data in a DXF & DWG formatted files

The Catia Interface

The Catia user interface has the same look and feel of a Windows interface. Some of the more important aspects of the interface are identified below.



Figure 1.1
Pull-down Menu

Pull-down Menus

The pull-down menus provide access to all the Catia commands.

When the menu item has an arrow like Figure 1.1 next to “Image” and “Macro” it means that there is a sub-menu associated with it and as soon as the cursor is placed on the arrow, the sub-menu will appear.

When a menu item is followed with a series of dots like Figure 1.1 next to “Customize” or “Options” it means that a larger dialog box or table with additional choices will appear.

Keyboard Shortcuts

Some menu items indicate a keyboard shortcut as seen in Figure 1.2. For example the “Undo” option has a shortcut of

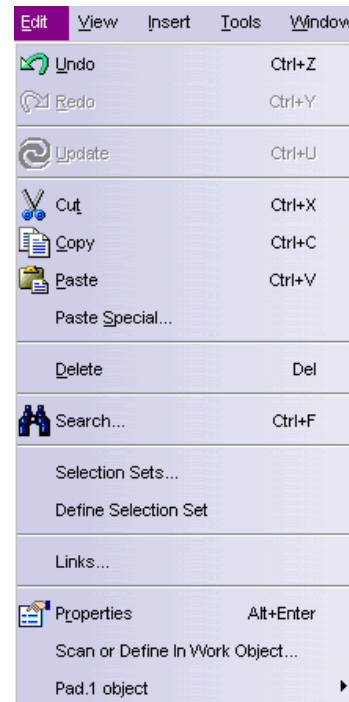


Figure 1.2
Pull-down Menu

“Ctrl+Z”. This mean that if you hold the Ctrl down while hitting “Z”, the last operation you performed will be removed or undone.

Catia conforms to standard Windows conventions for such shortcuts such as:

- Ctrl+N for File, New
- Ctrl+O for File, Open
- Ctrl+S for File, Save

You can also customize Catia by creating your own shortcuts.

Workbenches

Catia Workbenches provide shortcuts enabling you to quickly access the most frequently used commands. Workbenches are fully customizable and are organized according to functionality. Individual options on them will be discussed in detail throughout the course. The workbenches we will be using in this course are shown in Figures 1.3-1.6.

Standard Toolbar



Figure 1.3
Standard Toolbar

Sketcher Toolbar



Figure 1.4
Sketcher Toolbar

Part Toolbar



Figure 1.5
Part Toolbar

Wireframe & Surface Toolbar



Figure 1.6
Wireframe Toolbar

Assembly Toolbar



Figure 1.7
Assembly Toolbar

Workbench Arrangement

Arranging the Workbenches can be done in many ways. One such arrangement is shown in Figure 1.8. They can be docked around the borders of the Catia window or they can be dragged onto the graphics area. The workbenches are actually comprised of several toolbars. Each toolbar can be handled separately. To move a toolbar, drag it with the left mouse button by selecting the horizontal bar if the toolbar is vertical or the vertical bar if the toolbar is horizontal. See Figure 1.9.

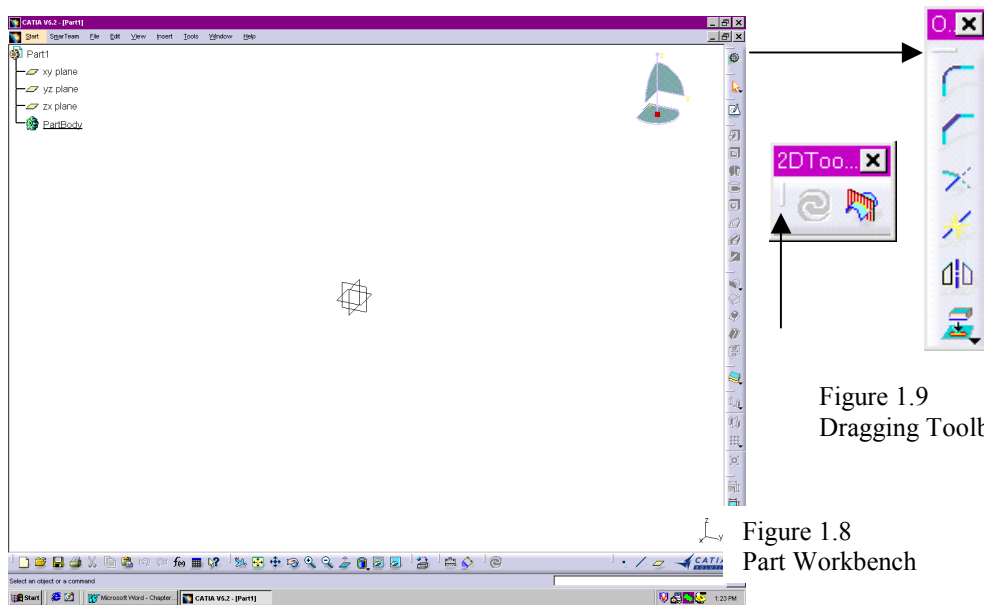


Figure 1.9
Dragging Toolbars

Figure 1.8
Part Workbench

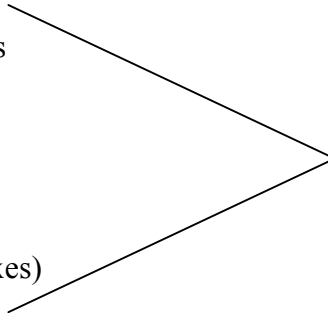
Mouse Buttons

The use of the mouse differs according to the type of action you need to perform.

Whenever you read...

...Use this mouse button

Select Menus
 Select Commands
 Select Geometry
 Click icons
 Double-click
 Shift-click
 Ctrl-click
 Check (check boxes)
 Drag icons



Mouse Button 1 (MB1)



Drag Geometry around graphics area

Mouse Button 2 (MB2)



Rotate Geometry in graphics area

Hold MB2 & Hold MB1

Zoom Geometry in graphics area

Hold MB2 & Click MB1

Contextual Menus

Mouse Button 3 (MB3)



Other options with the mouse operations are shown in Figure 1.10.

Shortcut For Manipulations					
Function	Rotate	Pan	Zoom	Reframe	
Keys to use	Shift + mouse button 2 *	mouse button 2 *	Ctrl + mouse button 2 *	Shift + R + mouse button click *	
Manipulation	Move the	Move the	Move the	Replace the	

Figure 1.10
 Mouse Manipulations

on	mouse in the rotatio n directi on	mouse in any directi ons	mouse top to zoom in and move down to zoom out	all model in your windo w
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Note: for a 2 buttons mouse (or in case of trouble with some browsers), use Alt + mouse button 1, to simulate button 2.

The V5 Menus

When the V5 program is initiated, the screen below is presented with a table that says “Welcome to CATIA Solutions V5R2”.

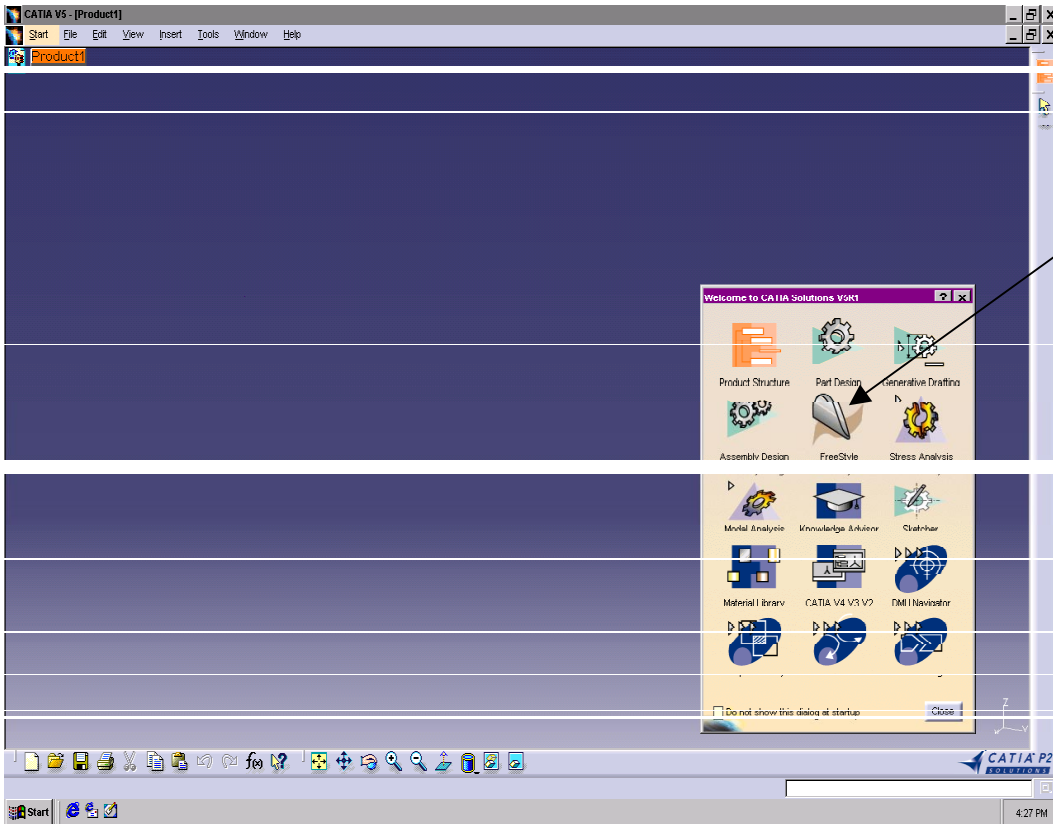


Figure 1.11
Opening Window

We will not go into all the options in the table at this time, but in general these are the various programs that run inside V5. Each one of these programs falls into one of four categories:

- | | | |
|----|----------|------------|
| 1. | ANALYSIS | 3. PART |
| 2. | DRAWING | 4. PRODUCT |

These four categories are referred to as **DOCUMENTS**. Each time one of the programs above is launched, a new document is created of the appropriate type.

The START pulldown menu (Figure 1.12) gives you the same options and will either start a new document or change the current document into the correct **WORKBENCH**. A Workbench is a unique set of icons that goes with a particular program.

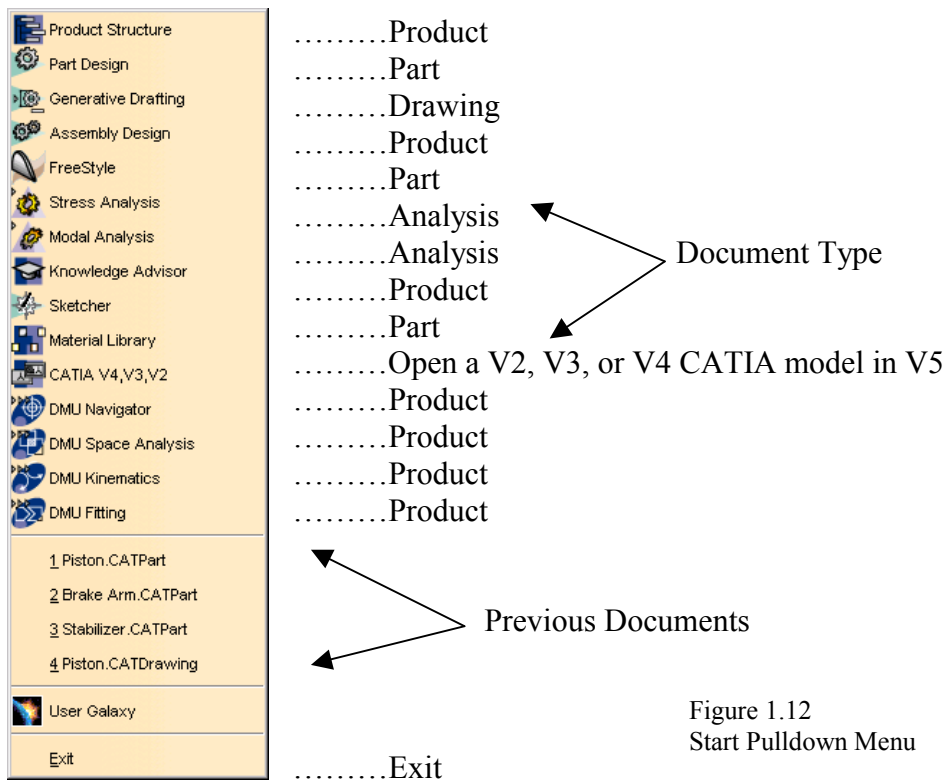
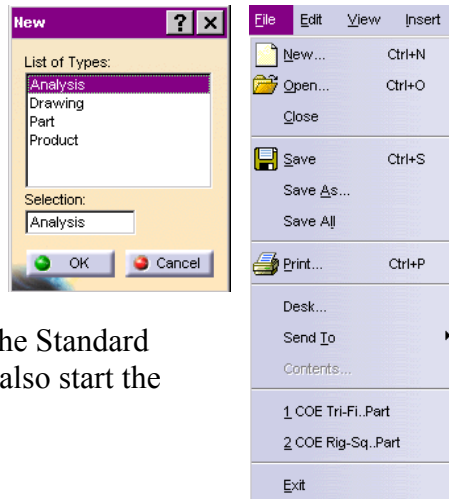


Figure 1.12
Start Pulldown Menu

Another option to start a new document is to select from the File pulldown menu the New option. Figure 1.13 shows both the File Pulldown menu and the result of selecting the New option. Select the type of document you want to create and hit OK.




Selecting the New Icon  from the Standard toolbar on the base of the screen will also start the New table shown in Figure 1.13.

Figure 1.13
File Pulldown Menu

Specification Tree

The Catia Specification Tree is a unique specification-driven and generative modeling approach. This is the mechanism for all CATIA V5 applications such as part design, assembly design and drawing generation. Figure 1.14 shows the Specification Tree for an assembly of three Products and four Parts. A Product is a document type and is an assembly of other assemblies and parts.

The Base Asm has been expanded to show that it contains two parts and one of the part has been expanded to show that it contains one PartBody.

CATIA part, drawing and assembly documents allow the user to view and edit data either in the specification tree, the geometry area, or in both at the same time.

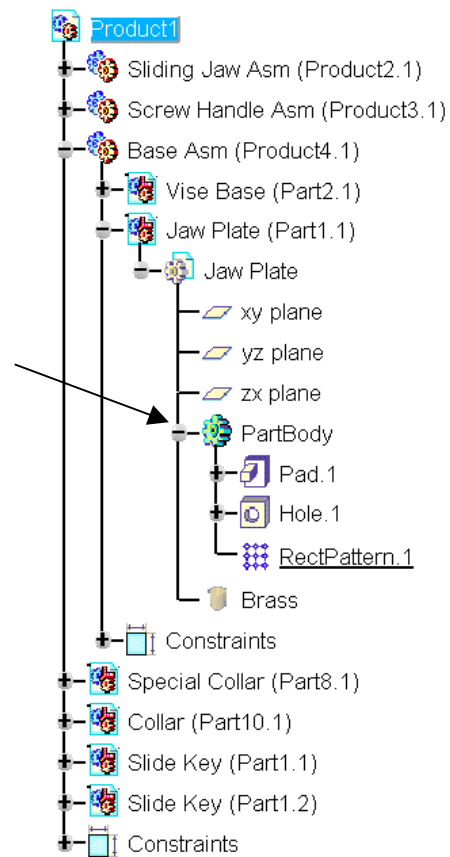


Figure 1.14
Specification Tree

In general, the Specification Tree is a detailed outline that shows every operation performed on the part, assembly, or drawing. Any modification that is made to the specification tree automatically changes the graphical representation and vice versa.

Options

Located on the Tools menu, the Options dialog box allows you to customize the Catia software to reflect such things as your company's drafting standards as well as your individual preferences and work environment.



Figure 1.15
Tools Pulldown

Feature-Based Parts

Just as an assembly is made up of a number of individual pieces, sub-assemblies or parts, a Catia part (new term for model) is also made up of individual elements. These elements are called features.

When you create a part using the Catia software, you work with intelligent features such as pads, pockets, holes, ribs, fillets, chamfers, and drafts, etc. As the features are created, they are applied directly to the current work piece now known as a PartBody.

Features can be classified as either sketched or applied.

- **Sketched Features:** Features based on a 2-D sketch. Generally a sketch is transformed into a solid by extrusion, rotation, or sweeping.
- **Applied Features:** Created directly on the solid model. Fillets, Chamfers, and Drafts are examples of this type of feature.

The Catia software graphically shows you the feature-based structure of your model in the Specification Tree. The Specification Tree not only shows you the sequence in which the features were created, it gives you

easy access to all the underlying associated information. You will learn more about the Specification Tree throughout this course.

To illustrate the concept of feature-based modeling, consider the part shown in Figure 1.16.

The part can be visualized as a Collection of several different features Some of which will add material, like the cylinder, and some which will Remove material, like the through hole in the cylinder.

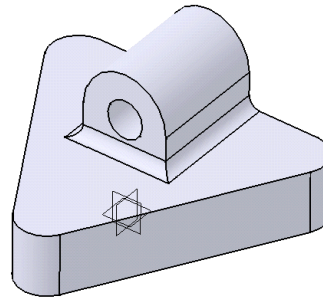


Figure 1.16
Part with Features

If we were to map the individual Features to their corresponding listing in the Feature Manager tree, it would Look like this:

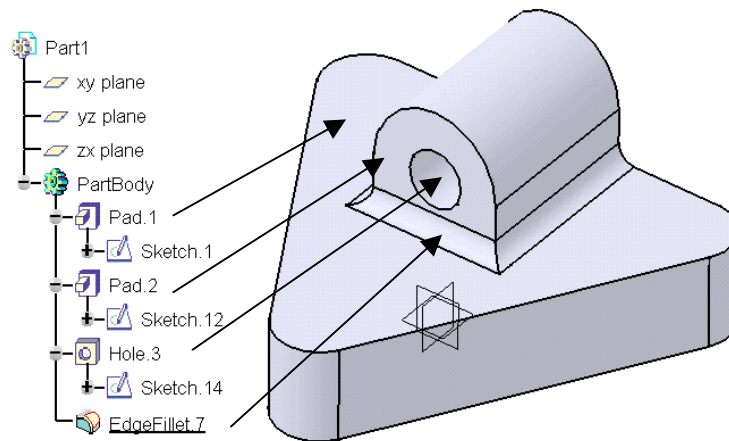


Figure 1.17
Part and Tree

Parametrics

Parametrics are the dimensions and relations used to create a feature. All of the dimensions and relations are stored in the model. This not only enables you to capture you design intent, it also allows you to quickly and easily make changes to the model.

