



CATIA V5 Training
Foils

Student Notes:

CATIA V5
Fundamentals

Version 5 Release 19
August 2008

EDU_CAT_EN_V5F_FF_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



About the Student Guide

Using the Student Guide

This student guide is intended to be used under the guidance of a certified CATIA instructor. It contains examples and case studies that are designed to be demonstrated by the instructor.

Exercises/Case Studies

This course illustrates the process-based approach in two ways: exercises and case studies.

Exercises give you the opportunity to apply and practice the material covered during the lecture/demonstration. These exercises have been created to represent typical design and modeling situations. Extra exercises have been included in this guide to accommodate students who may want to practice more modeling.

Case Studies provide a context in which you would use particular tools and methods, and illustrate the process flow you would typically follow to complete a project.

Conventions Used in the Student Guide

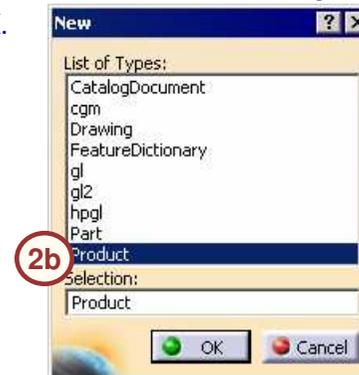
The following typographic conventions are used in the student guide:

- **Bold blue text** within a sentence denotes options selected from the CATIA menu bar.
- **Bold brown text** denotes the name of a tool, icon, button, or window option.
- *Italic text* within a sentence is used to apply emphasis on key words.
- Numerical lists are used in sequential lists, such as the steps of a procedure.
- Lower-case alphabetical sub-lists are used in sequential sub-lists, such as the sub-steps in an exercise procedure.
- **2b** identifies areas in a picture that are associated with steps in a sequential list, such as in an exercise.
- Upper-case alphabetical lists are used in non-sequential lists, such as a list of options or definitions.
- Text enclosed in < > brackets represents the name of the keyboard key that must be pressed.
- Text enclosed in [] brackets corresponds to the text that must be entered into a text field of a CATIA dialog box or prompt.
- Information whose availability depends on a particular configuration is marked using the symbol: **Conf. Dep.**

Example page:

Use the following steps to create a new document in CATIA:

1. Click **Start > Mechanical Design > Part Design**.
2. Create new part.
 - a. Click **File > New**.
 - b. Select **Product** from the **New** dialog box.
 - c. Click OK.



- d. Press <CTRL> + <S> to save the document.
- e. Type [my first document] as the document name.

You can create the following profile types:

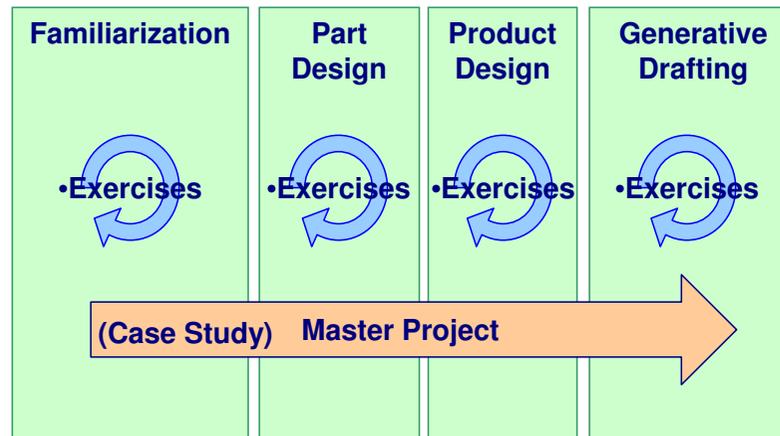
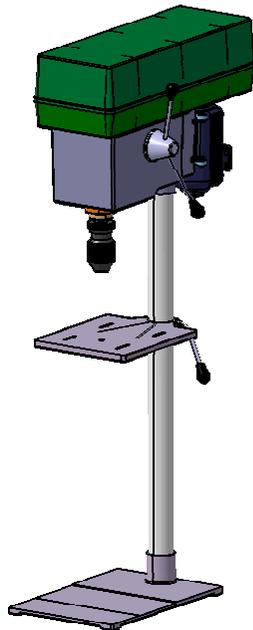
- A. User Defined Profiles
- B. Pre-Defined Profiles
- C. Circles



Case Study: Introduction to CATIA

Each lesson in this course contains a case study, which helps explain 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.

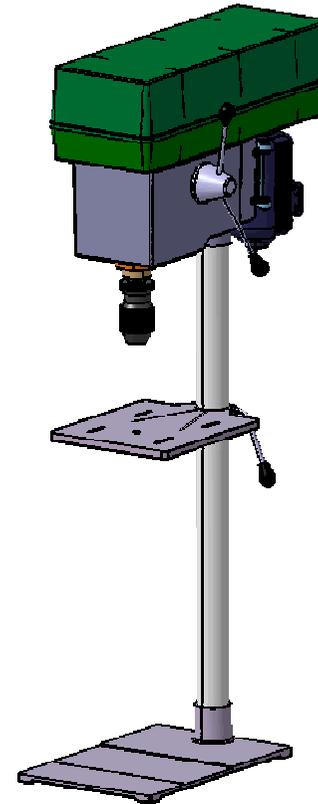
All models used in the case studies come from the Drill Press assembly, which is also your master project. In this lesson, you learn how to open and navigate models in CATIA.



Design Intent

Each case study contains a set of model requirements, known as the design intent. Design intent is discussed later in this lesson. 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.



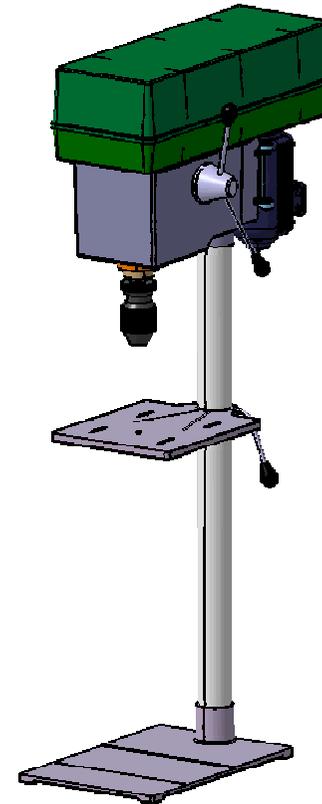
Student Notes:

Stages in the Process

Each lesson consists of steps. Each step contains the information you need to complete the case study and maintain its design intent.

For Lesson 1, 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.



Student Notes:

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

PLM is a set of standards used by companies to successfully maintain innovation in engineering and reduce manufacturing costs.

PLM can be defined as follows:

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®

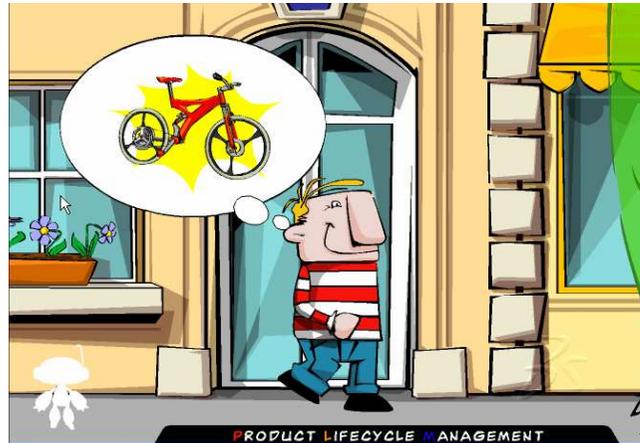
Source: CIMdata Inc.

Student Notes:

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...

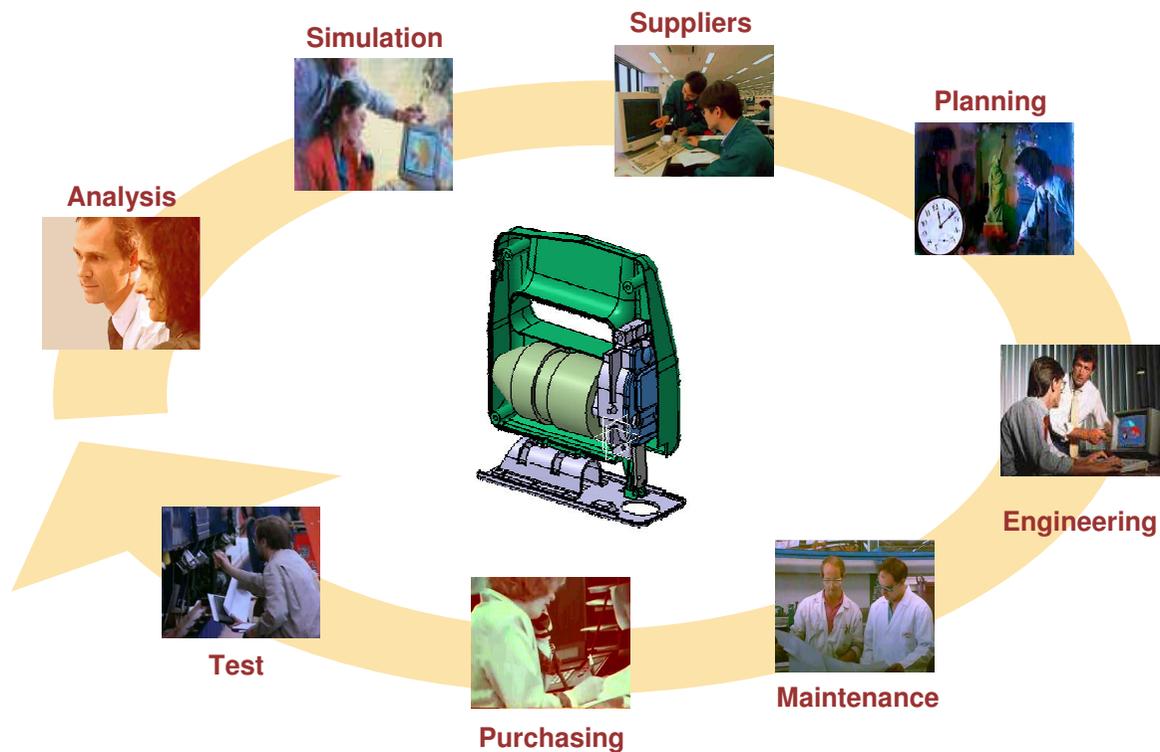
Searching for a new bike!



Student Notes:

CATIA Within the PLM Solution

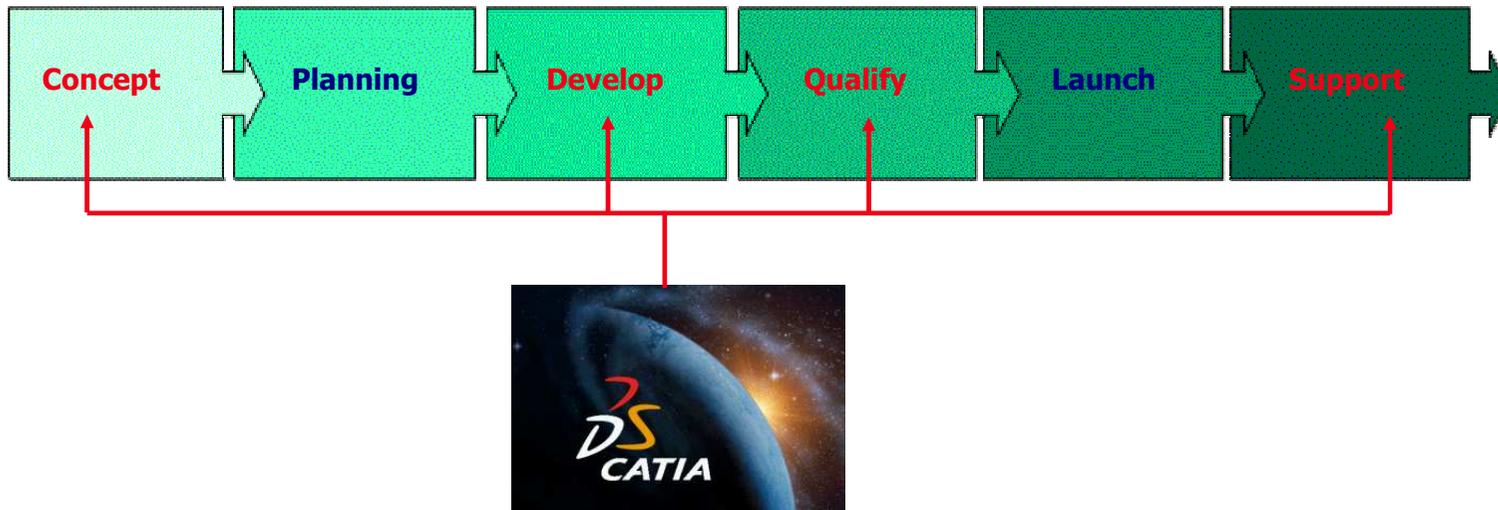
For the following functions CATIA plays an integral part of the PLM solution:



Student Notes:

CATIA's Scope From Concept To Realization

- 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.



What is CATIA V5?

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*.

To clarify this definition, the terms that are in *italics* above will be explained in the coming pages:



Key Terms (1/4)

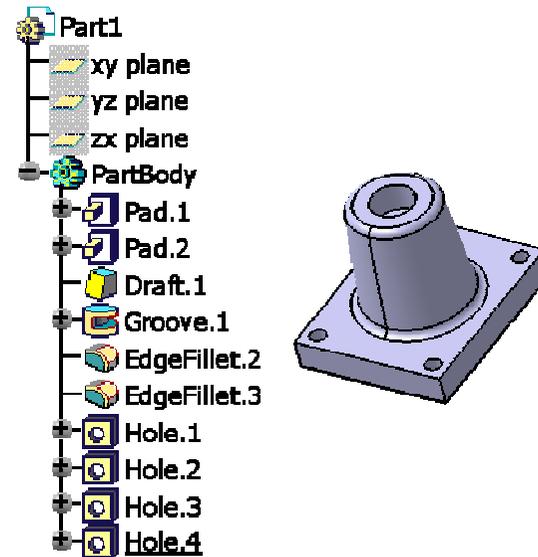
Feature-based

Like an assembly is made up of a number of individual parts, a CATIA document is made up of individual elements. These elements are called *features*.

While creating a document, you can add features such as pads, pockets, holes, ribs, fillets, chamfers, and drafts. As the features are created, they are applied directly to the work piece.

Features can be classified as sketch-based or dress-up:

- Sketch-based features are created using a 2D sketch. Generally, the sketch is transformed into a 3D solid by extruding, rotating, sweeping, or lofting.
- Dress-up features are created directly on the solid model. Fillets and chamfers are examples of this type of feature.

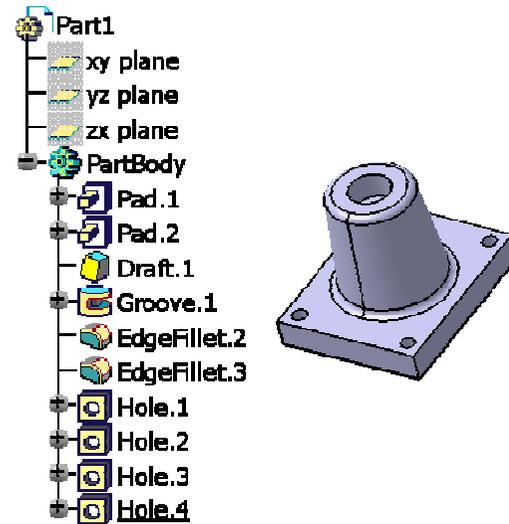


Student Notes:

Key Terms (2/4)

Feature-based (continued)

CATIA graphically displays the feature-based structure and other non-graphical data of your model in the form of a *specification tree*. The specification tree shows the sequence in which the features were created, and enables you to easily access all the underlying associated information and elements. You will learn more about the specification tree throughout this course.



Key Terms (3/4)

Parametric

The dimensions and relations used to create a feature are stored in the model. This enables you to capture the design intent, and to easily make changes to the model using these parameters.

- *Driving dimensions* are used while creating a feature. These include the dimensions that are associated with the sketched geometry, as well as dimensions that are associated with the feature itself. Let us consider the example of a cylindrical pad. The diameter of the pad is controlled by the diameter of the sketched circle, and the height of the pad is controlled by the depth to which the circle is extruded.
- *Relations* include information such as parallelism, tangency, and concentricity. This type of information is typically communicated in the drawings using feature control symbols. By capturing this information in the sketch, CATIA enables you to fully capture your design intent in the initial stage itself.

Solid Modeling

A solid model is the most complete type of geometric model used in CAD systems. It contains all the wireframe and surface geometry necessary to fully describe the edges and faces of the model. In addition to geometric information, solid models also convey their *topology*, which relates the geometry together. For example, topology might include identifying which faces (surfaces) meet at which edges (curves). This intelligence makes it easier to add new features. For example, if a model requires a fillet, you simply select an edge and specify a radius to create it.

Key Terms (4/4)

Fully Associative

A CATIA model is fully associative with the drawings and parts or assemblies that reference it. Changes in the model are automatically reflected in the associated drawings, parts, and/or assemblies. Likewise, changes in the context of the drawing or assembly are reflected back in the model.

Constraints

Geometric constraints (such as parallel, perpendicular, horizontal, vertical, concentric, and coincident) establish relationships between the features of a model by fixing their positions with respect to one another. In addition, equations can be used to establish mathematical relationships between parameters. By using constraints and equations, you can guarantee that design concepts such as *through holes* and *equal radii* are captured and maintained.

Design Intent (1/3)

Design Intent is the plan of how to construct the solid model of a part in order to properly convey its visual and functional aspects. In order to efficiently use a parametric modeler like CATIA, you must consider the design intent before and while modeling the part. The techniques used to create the model will affect how the model behaves when it is changed during its life cycle.

The manner in which a solid model is built can affect many aspects, including its flexibility to change, its stability during the change process, and the resources it requires to compute a new result. Therefore, it is important to take the design intent into account for creating an efficient solid model of the part.

The following factors contribute to how the design intent is captured:

- **Automatic (Implicit) Relations:** Based on how the geometry is sketched, automatic relations provide common geometric relationships between objects, such as tangency, parallel, perpendicular, horizontal, and vertical.
- **Equations:** Equations relate the dimensions mathematically; they provide an external way to force changes.
- **Additional Relations:** Relations that are defined while creating the model provide another way to connect related geometry. Some common relations are concentric, coincident, and offset.
- **Dimensioning:** The manner in which a sketch is dimensioned impacts the design intent. You must add dimensions in a way that reflects how you would like to change them to control the elements.

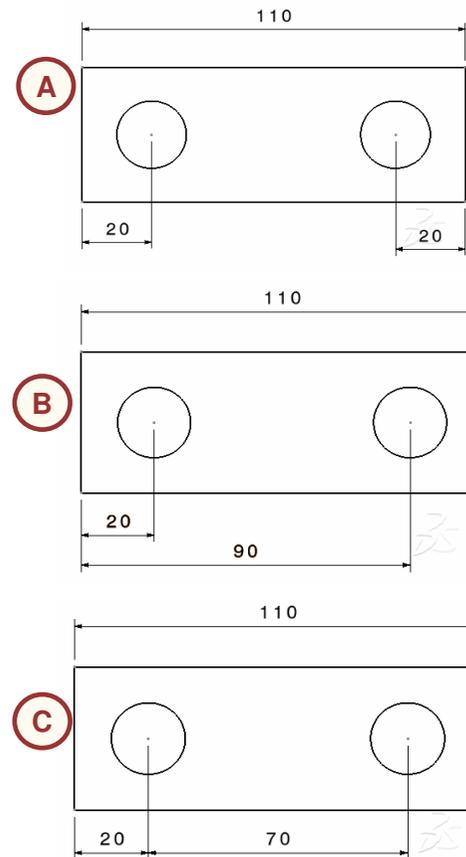
Design Intent (2/3)

The following examples show different design intents for the same sketch:

- A. This sketch is dimensioned to keep the holes at a distance of 20mm from the left and right edges, regardless of the overall plate width (110mm).

- B. Baseline dimensions keep the holes positioned relative to the left edge of the plate. The positions of the holes will not be affected if the width of the plate is changed.

- C. Dimensioning from the left edge, and then from the center of the first hole to the center of the second hole maintains their distance from the left edge and between themselves, regardless of the overall plate width.

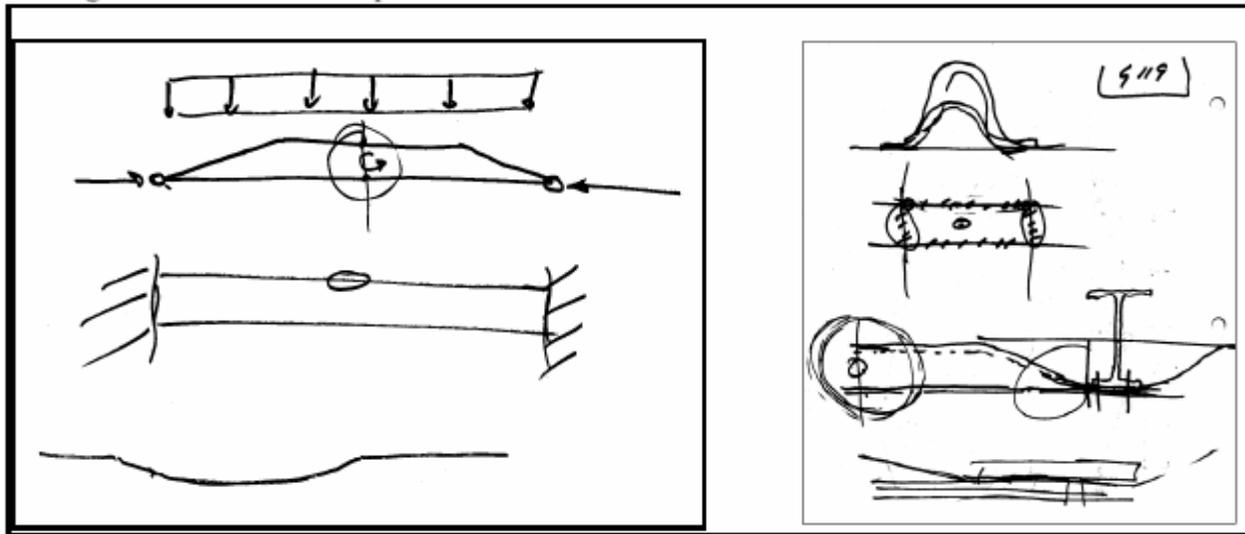


Student Notes:

Design Intent (3/3)

The example below shows a simple hand-drawn sketch of a modeling plan that details the complete design intent of the part. This type of pre-planning, even before starting to model the part using CATIA, is an excellent strategy to ensure the desired outcome is clear and precise.

Through pre-planning, you can become efficient at creating a robust model design that provides both flexibility and stability during modifications.

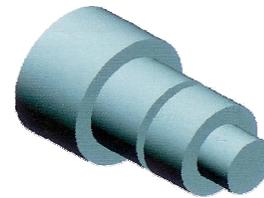


How Features Affect Design Intent (1/2)

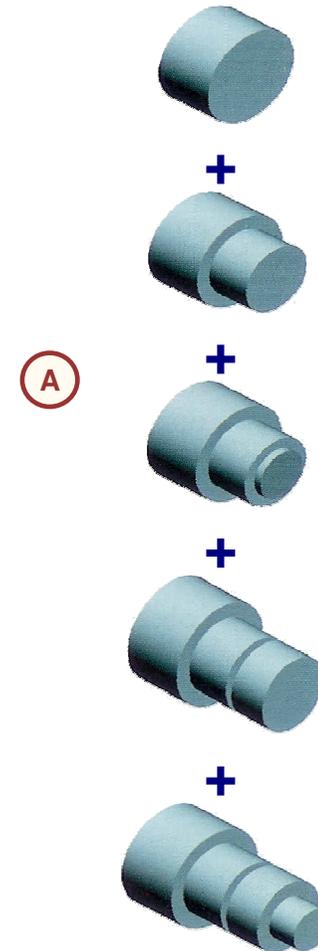
In addition to how a sketch is dimensioned, the choice of features and the modeling methodology affect the design intent. Consider the stepped shaft (Final Result) shown below. You can build this part in several ways:

A. "Layered" Approach

- The layered approach builds the part one piece at a time, adding a layer or feature onto the previous one until the desired solution is obtained.
- Changing the thickness or the shape of one layer has a ripple effect; it changes the position or location of all the other layers that were created after it.



Final Result



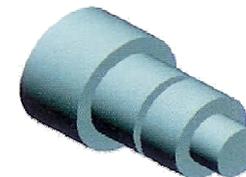
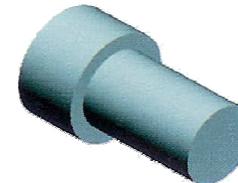
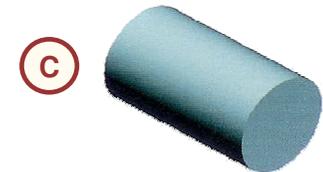
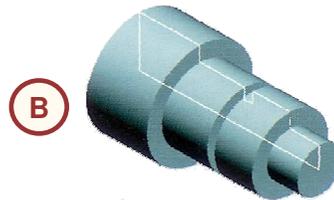
Student Notes:

How Features Affect Design Intent (2/2)

You can build this part in several ways (continued):

B. "Potter's Wheel" Approach

- The potter's wheel approach builds the part as a single, revolved feature. A single sketch, representing the cross-section, includes all the information and dimensions necessary to make the part as one feature. While this approach may seem the most efficient, having all the design information contained within a single feature limits flexibility and can make changes difficult.



C. "Manufacturing" Approach

- The manufacturing approach to modeling mimics the way the part would be manufactured. For example, if this stepped shaft was turned on a lathe, you would start with a piece of bar stock and remove material using a series of cuts.

Exercise: Design Intent

Recap Exercise



In this exercise, you will identify and describe the key terms of CATIA and identify the design intent for various parts.

By the end of this exercise you will be able to:

- Define the key terms of CATIA
- Identify and describe the design intent for various parts

Do it Yourself

Identify the correct definition for the following terms:

- **Solid modeling**
- **Feature-Based**
- **Parametric**
- **Constraints**
- **Design Intent**
- **PLM**

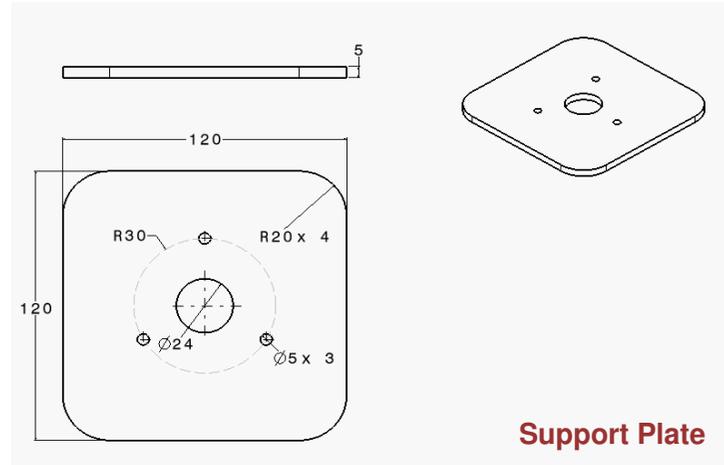
- A. Geometric elements, such as pads, pockets, holes, ribs, fillets, chamfers, etc. that are applied to a work piece to form a part.
- B. A geometric model that contains all the topology, wireframe and surface geometry necessary to fully describe the edges and faces of a part.
- C. The dimensions and relations used to create a feature are captured and stored in the model.
- D. The plan that explains how to construct or develop the solid model of a part, in order to properly convey the visual and functional aspects of the part.
- E. 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.
- F. Geometric relationships, such as parallel, perpendicular, horizontal, vertical, concentric, and coincident, or equations to establish mathematical relationships among parameters.

Student Notes:

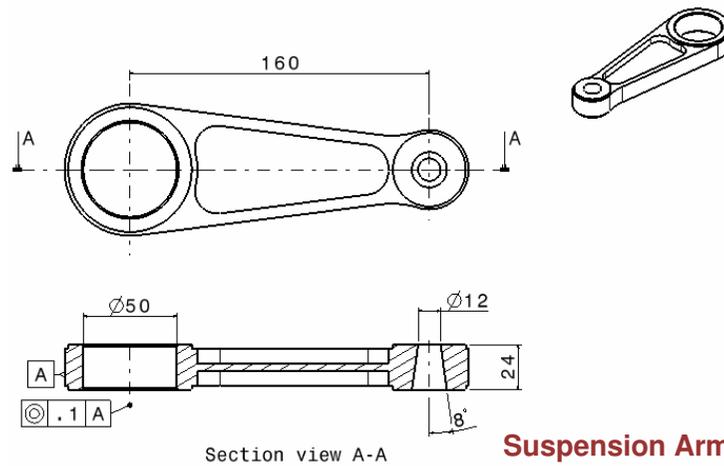
Exercise Illustration

Using the drawings, identify all items that should be reflected in the design intent:

Support Plate



Suspension Arm



Open CATIA

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



Copyright DASSAULT SYSTEMES

A large black clipboard with a silver clip at the top. Inside the clipboard, the text reads: "Use the following steps:" followed by a list of three items. The second item, "2. Open CATIA.", is highlighted in red. A faint "DASSAULT SYSTEMES" logo is visible in the background of the clipboard.

Use the following steps:

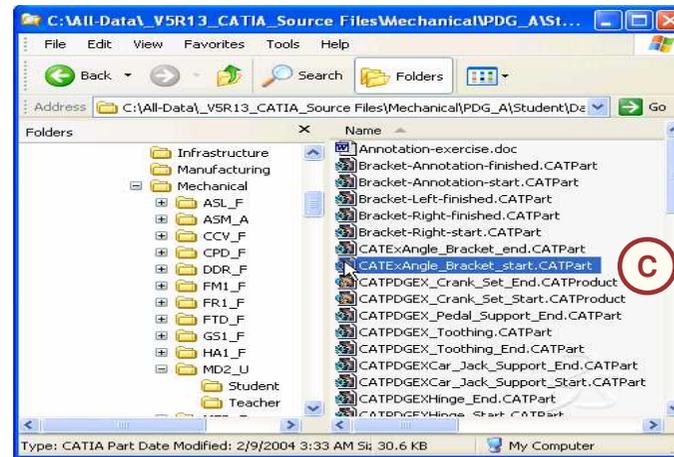
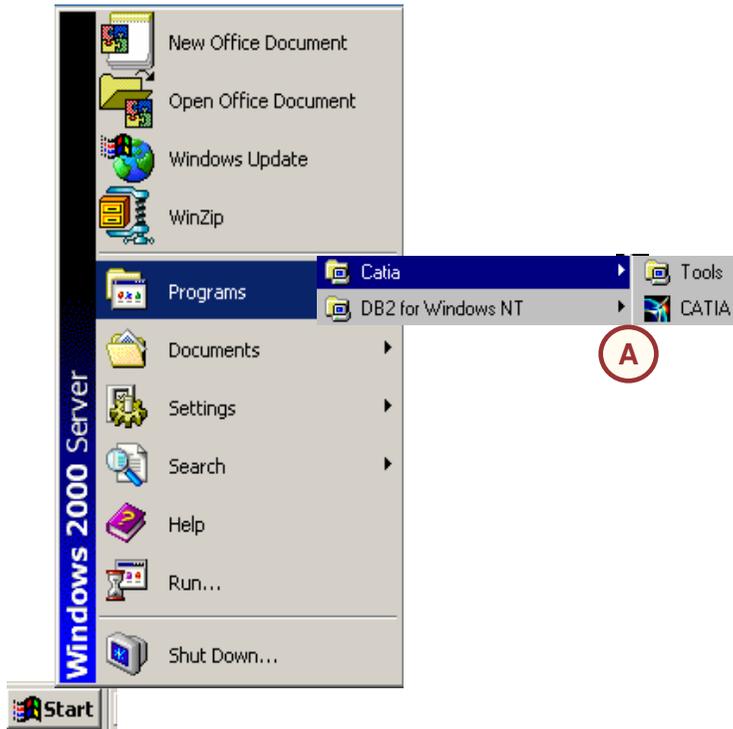
- ✓ 1. Understand the CATIA software.
- 2. Open CATIA.**
3. Understand the CATIA interface.

Student Notes:

Starting CATIA Using the Start Menu

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

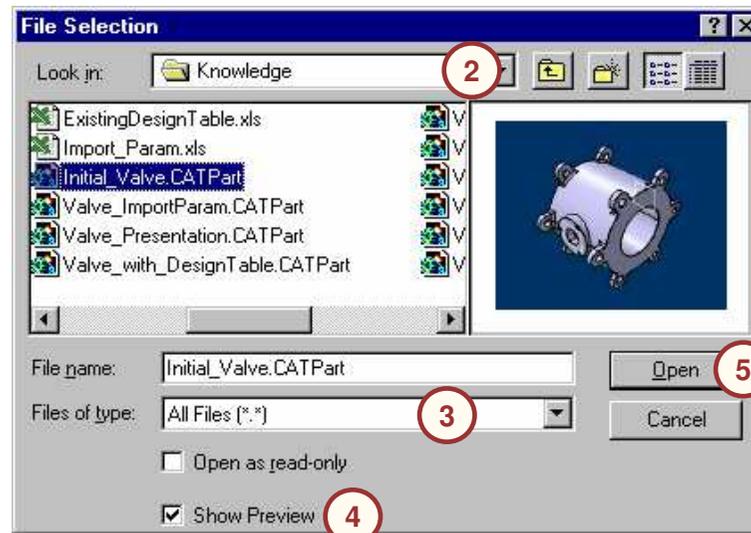
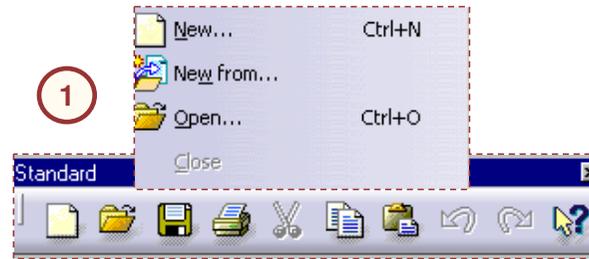
- A. Select **CATIA** from the **Start > Programs > CATIA** menu.
- B. Double-click the **CATIA** icon on your Windows desktop.
- C. Double-click on an existing CATIA document.



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**.



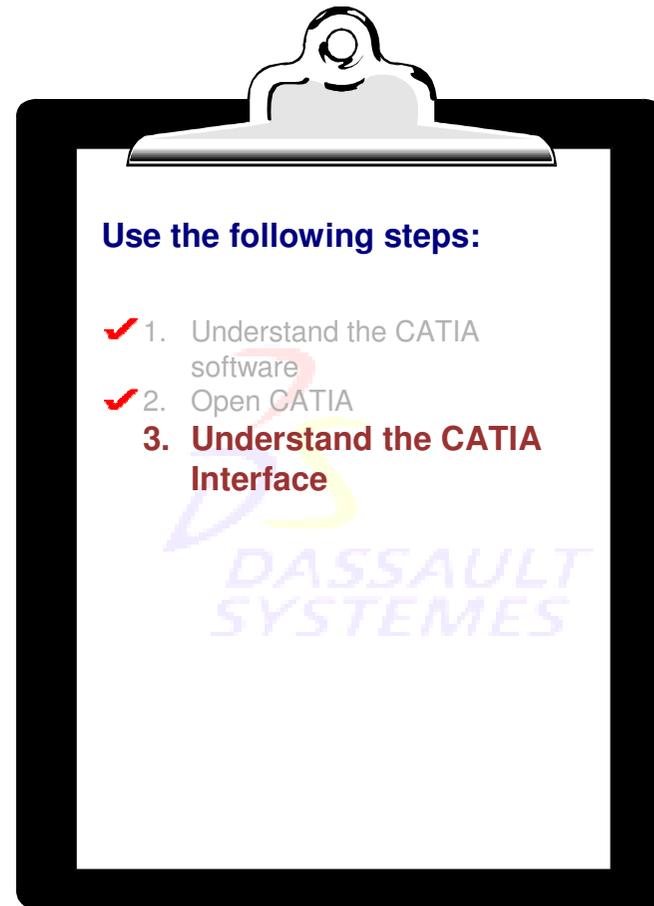
Note that if you are working in an environment where CATIA V5 is connected to a Product Data Management (PDM) system, CATIA documents will not be stored as files on a local disk, but will be managed by the PDM system. The interface will differ from that shown above.

Understand the CATIA Interface

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



Copyright DASSAULT SYSTEMES



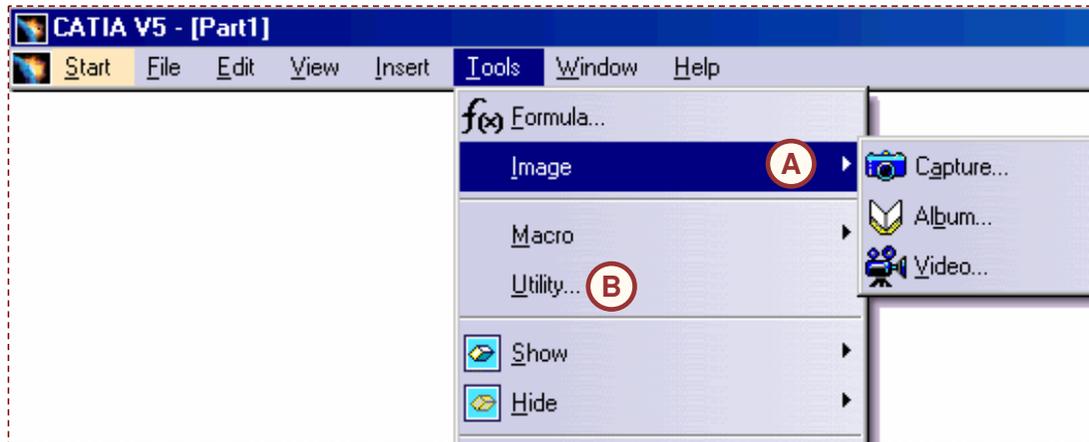
Windows Philosophy (1/4)

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.

Some menu options have additional options related to them:

- A. An arrow pointing to the right indicates a sub-menu.
- B. A command name followed by a series of dots indicates that selecting the command will open a dialog box with additional options.



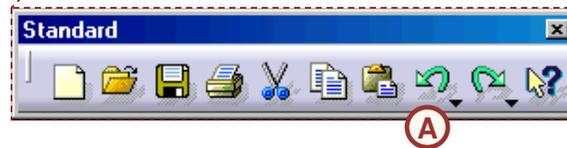
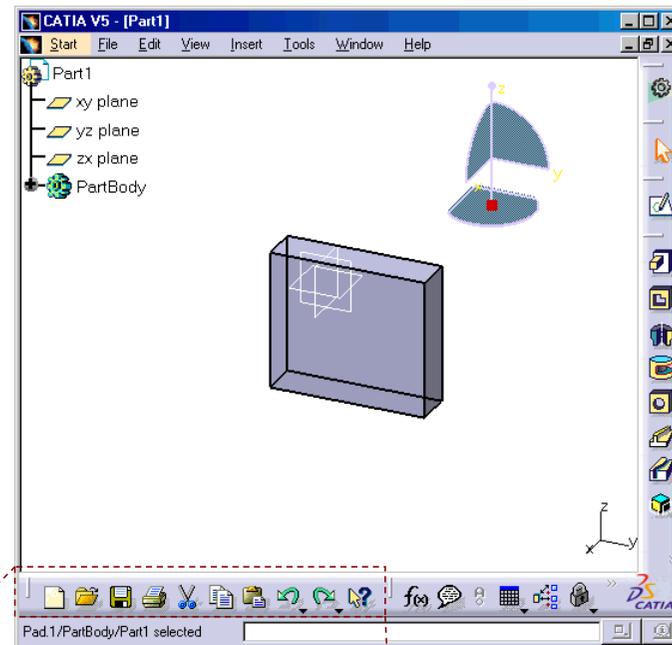
Windows Philosophy (2/4)

Toolbars contain icons for quick access to the most frequently used commands. Toolbars are organized into workbenches. They can be customized, rearranged, and relocated as per your preferences.

For example, the Standard toolbar contains commands to open, save, print, cut, undo, and access online documentation.

Some toolbar icons have additional icons related to them:

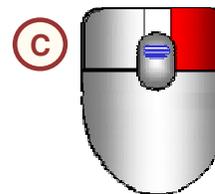
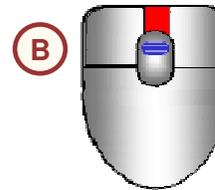
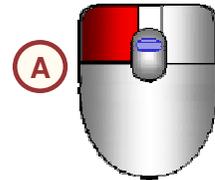
- A. An arrow (pointing downwards) besides an icon means more tools of the same type are available under it.



Windows Philosophy (3/4)

CATIA V5 uses a three-button mouse for selecting and indicating input from the user. Mentioned below are the general functionalities of the mouse buttons. A complete description of their use will be covered later.

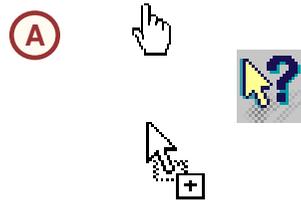
- A. The left mouse button is used to select displayed elements or items on the screen.
- B. The center mouse button (or the thumb wheel) is used to indicate or point to a direction on the screen.
- C. The right mouse button is used to display a contextual menu for the currently selected or pre-selected elements on the screen.



Windows Philosophy (4/4)

CATIA provides various levels of system feedback to users, such as:

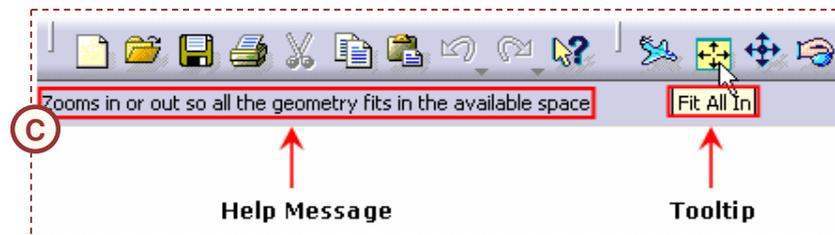
A. The mouse cursor is represented by different symbols to indicate various statuses.



B. Messages are displayed to convey information about the progress, failure, or result of a procedure.



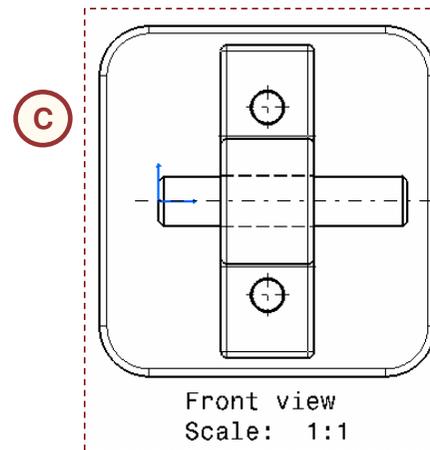
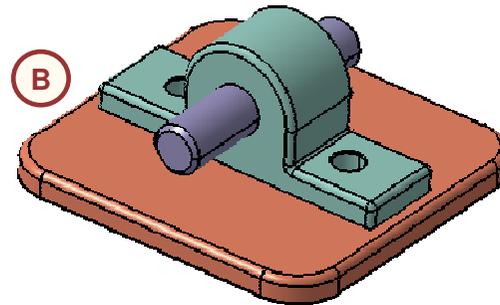
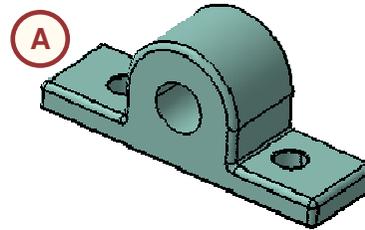
C. A ToolTip and a short help message is displayed when you place the mouse cursor over a particular tool (without selecting it).



Introduction to V5 Documents

You can create, modify, and save different types of documents in CATIA. These documents contain the geometrical information and specifications that define an object. The most common documents are:

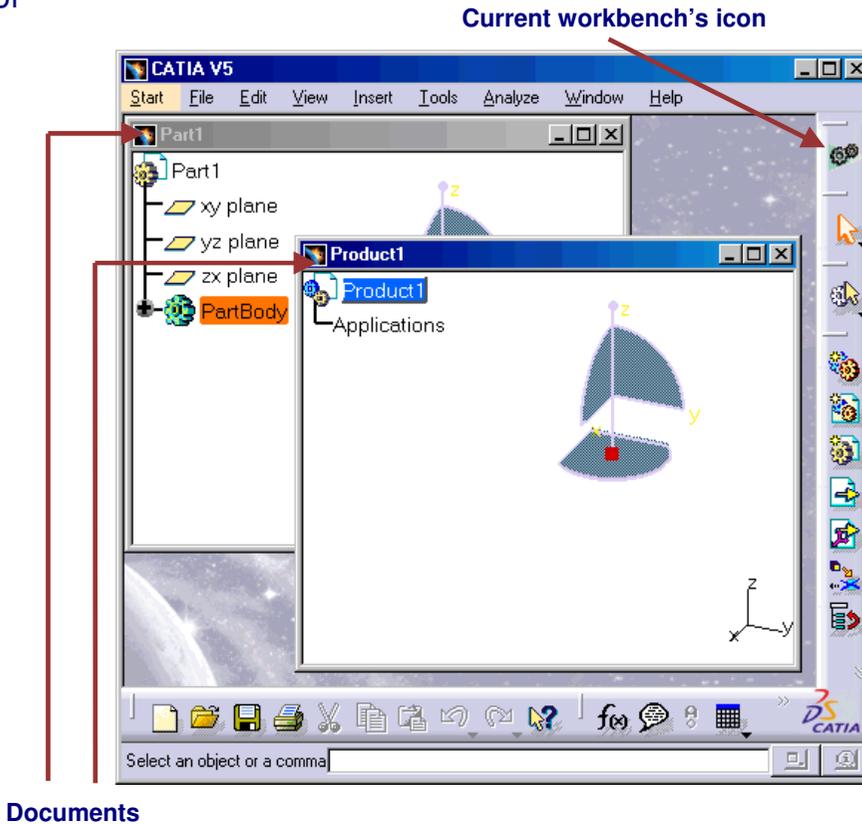
- A. A part document (.CATPart)
- B. An assembly document (.CATProduct)
- C. A drawing document (.CATDrawing)



The Workbench Concept

To work on a CATIA document, you must use one of the workbenches assigned to the document type. Each workbench contains a set of tools that is dedicated to perform a specific task. Following is a list of workbenches that are most commonly used:

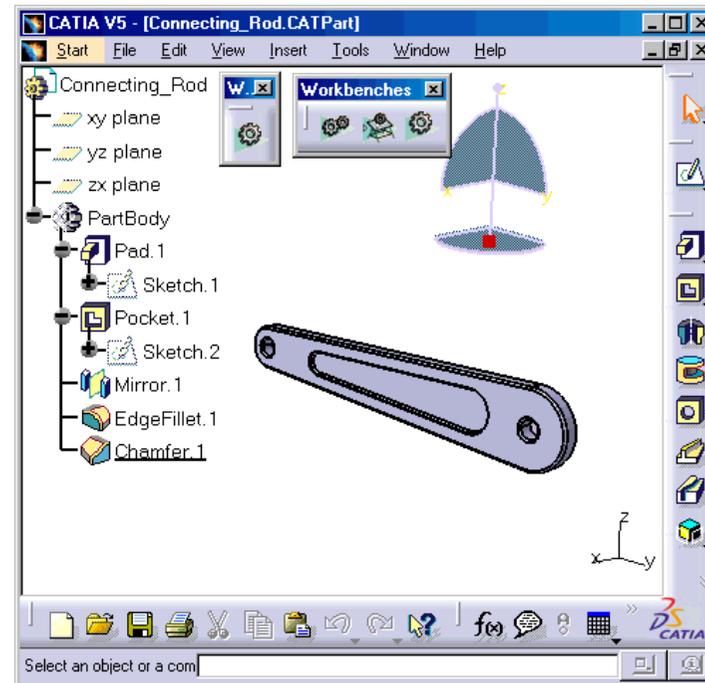
- 
•
Part Design: For designing parts using the solid modeling approach
- 
•
Sketcher: For creating 2D profiles with associated constraints, which can then be used to create other 3D geometry.
- 
•
Wireframe & Surface: For creating complex part features with 3D wireframe and surface elements.
- 
•
Assembly Design: Create constraints, features, and specifications for parts in the context of an assembly.
- 
•
Generative and Interactive Drafting: For creating drawings from parts and assembly designs.



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

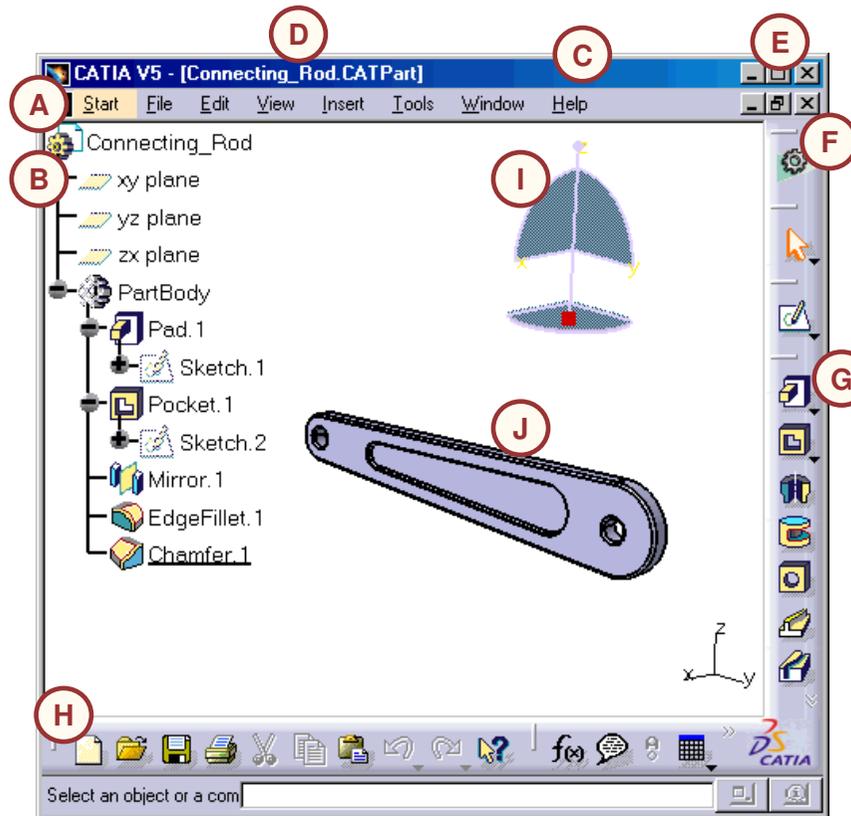


Student Notes:

CATIA User Interface (2/2)

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

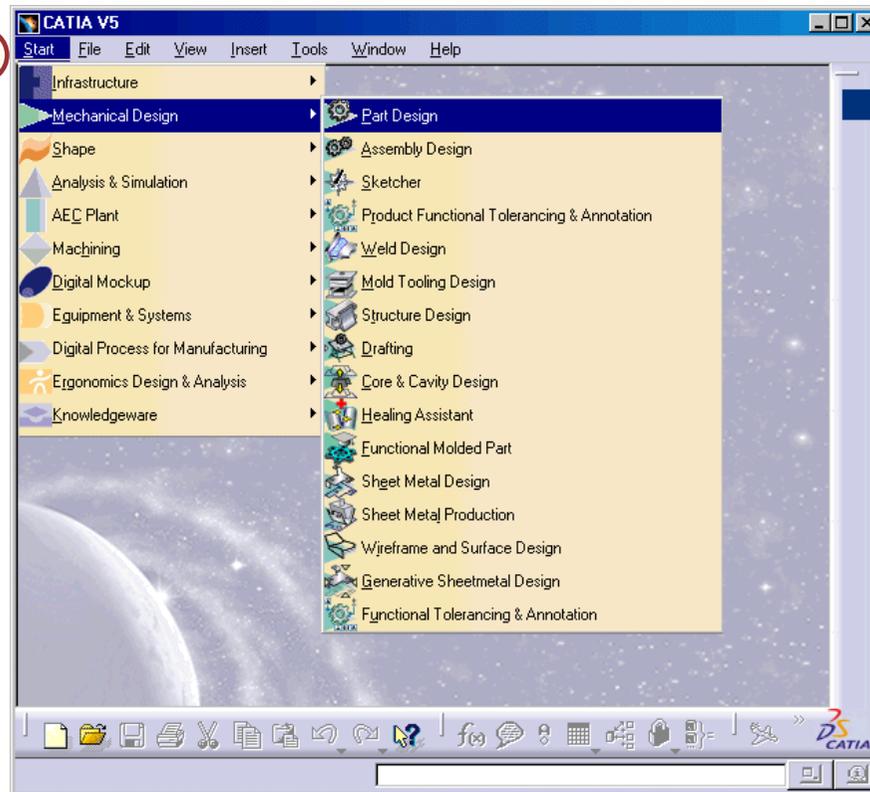
- A. Menu Commands
- B. Specification Tree
- C. Window of the active document
- D. Filename and extension of current document
- E. Icons to maximize, minimize, and close the window
- F. Icon of the active workbench
- G. Toolbars specific to the active workbench
- H. Standard toolbar
- I. Compass
- J. 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:

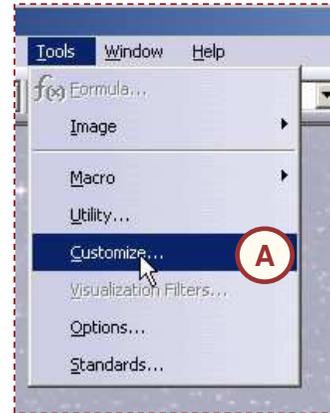
- A. Use the **Start** menu to open the required workbench. A
- B. Click **File > New** to create a new document of a particular file type. The associated workbench automatically opens.



Workbenches (2/2)

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

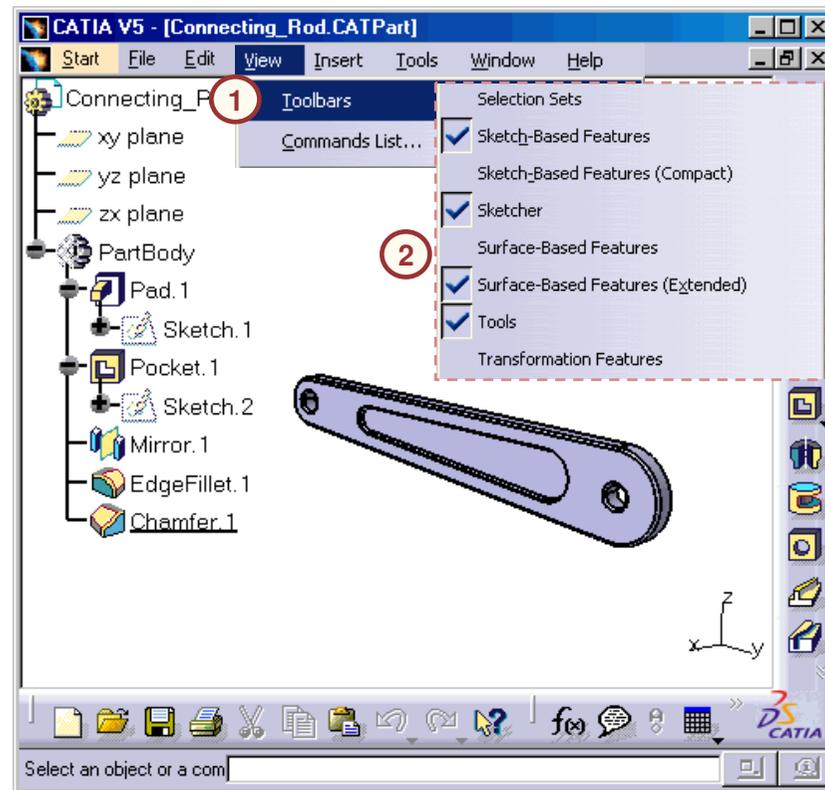
- A. Click **Tools > Customize** to access the dialog box.
- B. Using the Start Menu tab, select the workbench from the list.
- C. Select the arrow to add the workbench to the Favorites list.
- D. Click **Close** to exit.
- E. The workbench will be displayed when the **Start** menu is next selected.



Menus and Toolbars (1/3)

Toolbars provide quick access to tools that are also available in the menus. Use the following steps to activate/deactivate a toolbar:

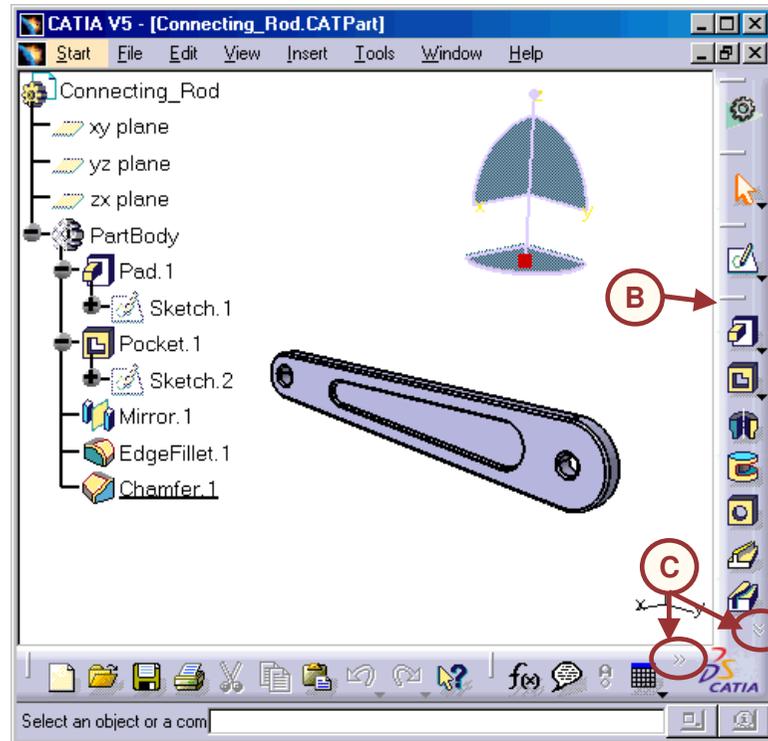
1. Click **View > Toolbars**.
2. A list of toolbars is shown with respect to the current workbench.
3. Activated toolbars have checkmarks beside them.
4. Select a particular toolbar to activate or deactivate it.



Menus and Toolbars (2/3)

You can further manipulate toolbars to close, reorganize, or display them:

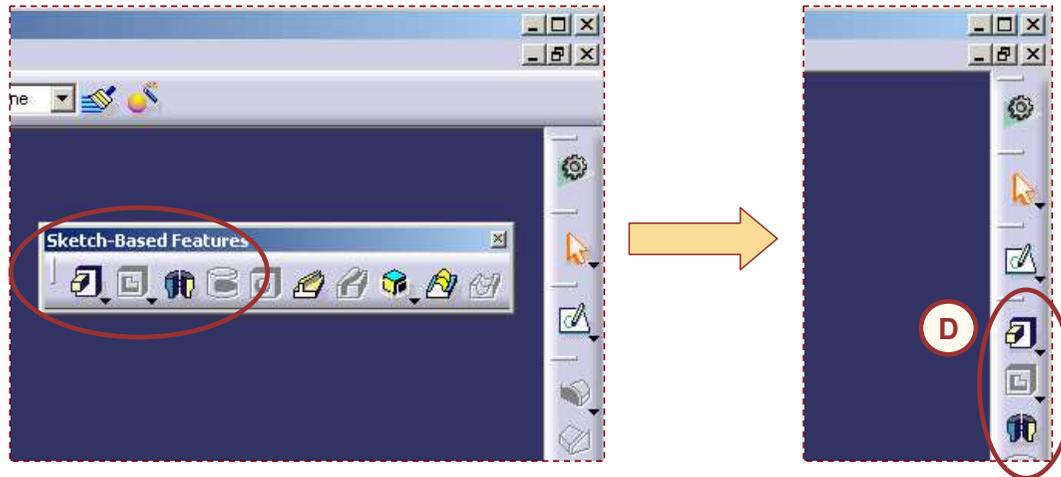
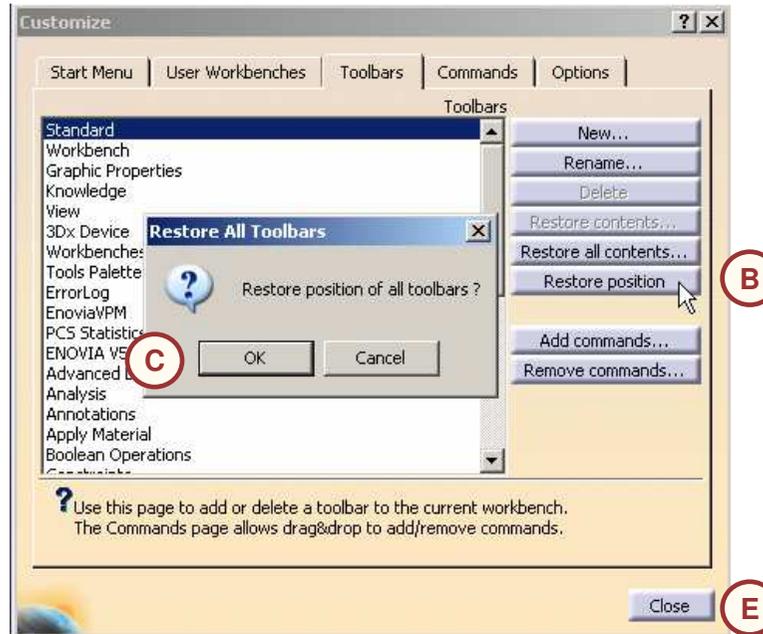
- A. Click **Close** to deactivate a floating toolbar.
- B. Reorganize the toolbars by dragging the separator of any toolbar and moving it to another location on the screen.
- C. A double-arrows (>>) symbol in the corner of the toolbar area indicates that there are more tools available, but are not visible due to the window's size. Drag the symbol to the Geometry Display Area to view these tools.



Menus and Toolbars (3/3)

You can reset the original toolbar configuration as follows:

- A. Click **Tools > Customize** to access the dialog box.
- B. Using the Toolbars tab, click the Restore Position button.
- C. Confirm by clicking **OK**.
- D. The floating toolbar will be reset to its original fixed position.
- E. Click **Close** to exit.

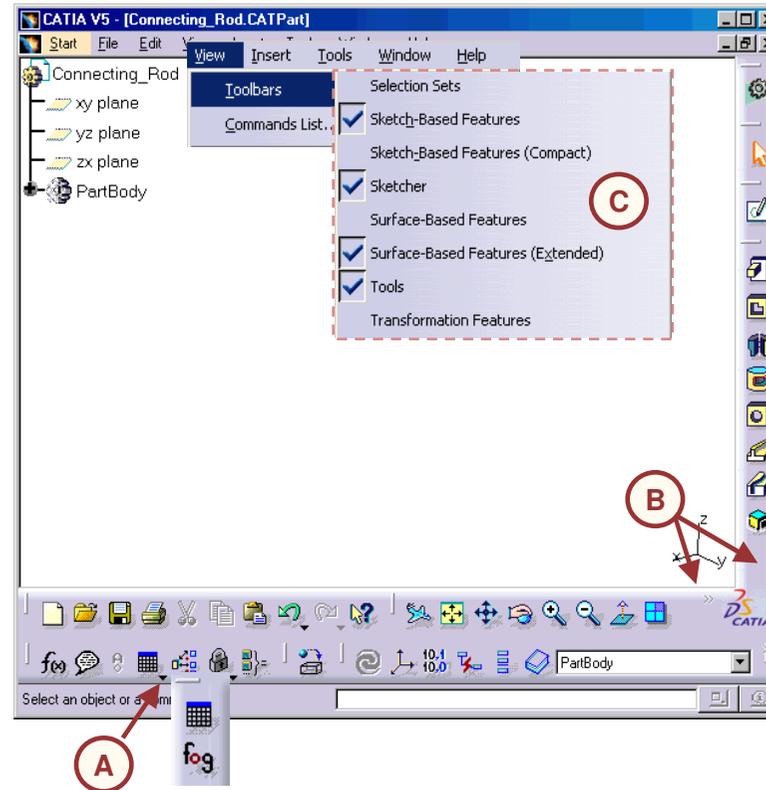


Finding Tools

Since CATIA contains a large number of tools and toolbars, it is not possible to display all of them within the limited window space. CATIA manages this using collapsible toolbars and by stacking extra toolbars at the corners of the window.

If you are unable to find a tool/toolbar, ensure that:

- A. the tool you are looking for is not located as a “fly out” optional tool. CATIA groups variants of the same tool into a single display group, which can be accessed by clicking the black arrow.
- B. the tool/toolbar is not located outside the display area. Look for the >> symbols at the corners of the window. You can drag the toolbar out to see all the tools.
- C. the toolbar is active.



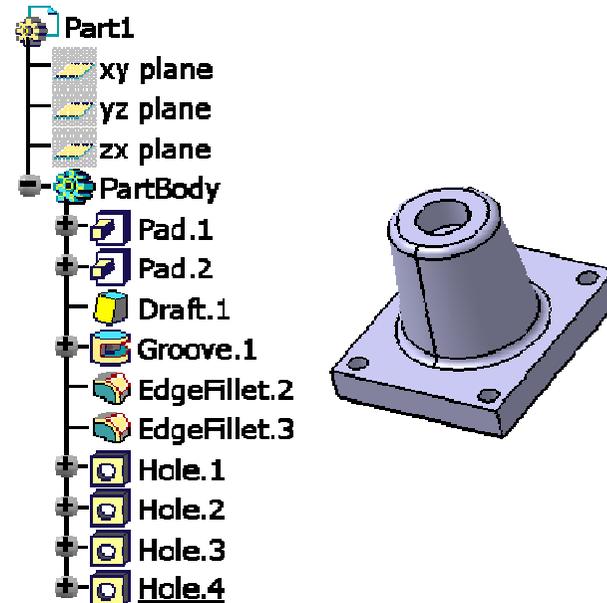
The Specification Tree

CATIA V5 provides a specification tree, which keeps the hierarchy of features, constraints, and processes, and the assembly information for a CATIA document. The specification tree provides a visual step-by-step record of the sequence followed while creating a solid model.

You can edit, reorder, or remove steps in the design process and specifications to achieve a new finished part without having to recreate the model.

The specification tree can suppress certain features and information by temporarily removing them from consideration for the model.

For example, the model on the right contains several hole features. You can simply “turn off” (suppress) the holes and have the model reflect the base area without actually deleting the holes.



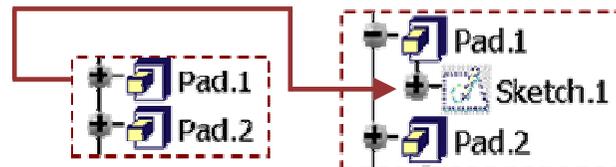
Student Notes:

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		Press the <F3> key to hide or show the tree.
Activate/Deactivate the tree	 Or 	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.

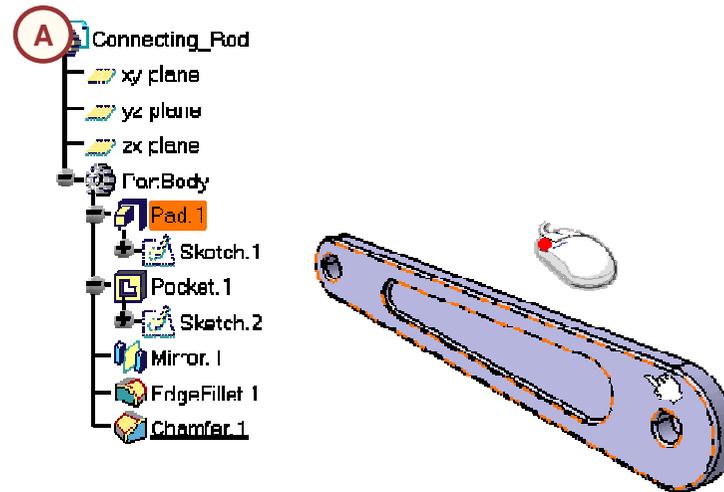


Selecting Objects with the Mouse

CATIA, like any other Windows environment application, has an interface that is primarily mouse-driven. When working in CATIA, there are two ways to select the objects:

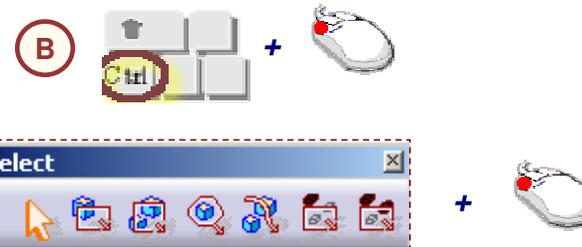
A. Simple Selection

- To select an object, press the left mouse button.
- You can select an object directly from the model or by selecting its corresponding feature in the tree. Selecting the geometry highlights the tree feature and vice versa.



B. Multi-Selection

- To multi-select, press the <Ctrl> key while using the left mouse button to select the objects.
- Multi-selection can also be done by trapping objects within a selection area.

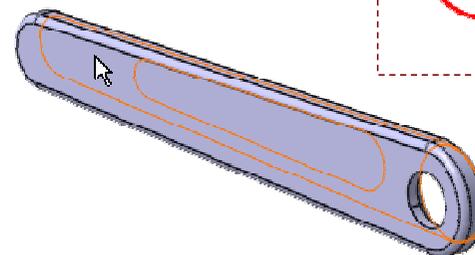
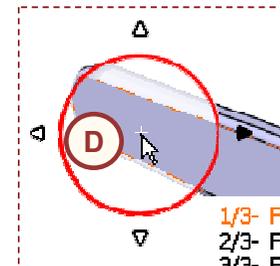
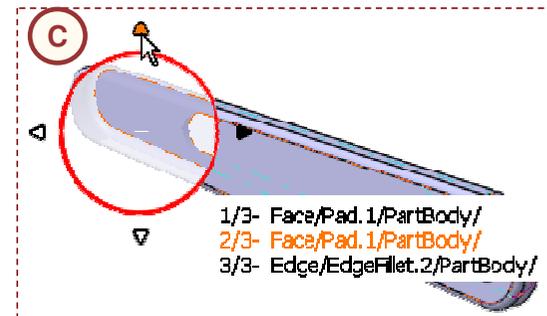
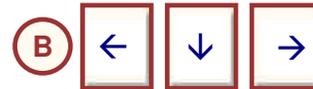
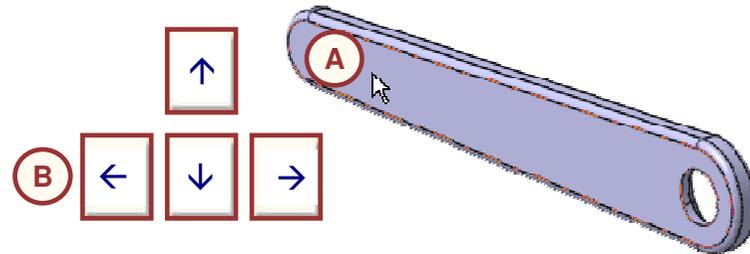


The Pre-selection Navigator

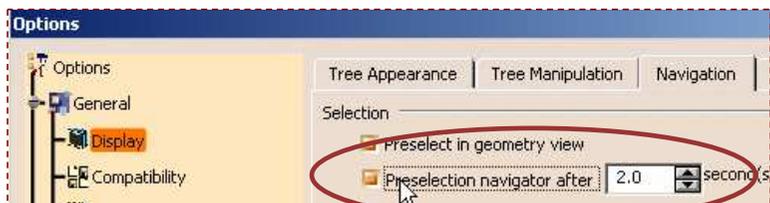
In certain cases the element that you wish to select is not visible, for example, an internal or hidden face. To select the rear face of the example part shown:

- Position the cursor at a position where the face would be located.
- Select the Right or Left arrow on the keyboard to display the Pre-selection Navigator. The nearest element will be pre-selected by default.
- Select the Up or Down arrow to scroll through the displayed list of elements until the required element is highlighted.
- Click within the circular zone to select the element.

Note: once the Pre-selection Navigator is displayed, the left/right arrows can then be used to scan the model feature hierarchy i.e. the parent features of the pre-selected element.



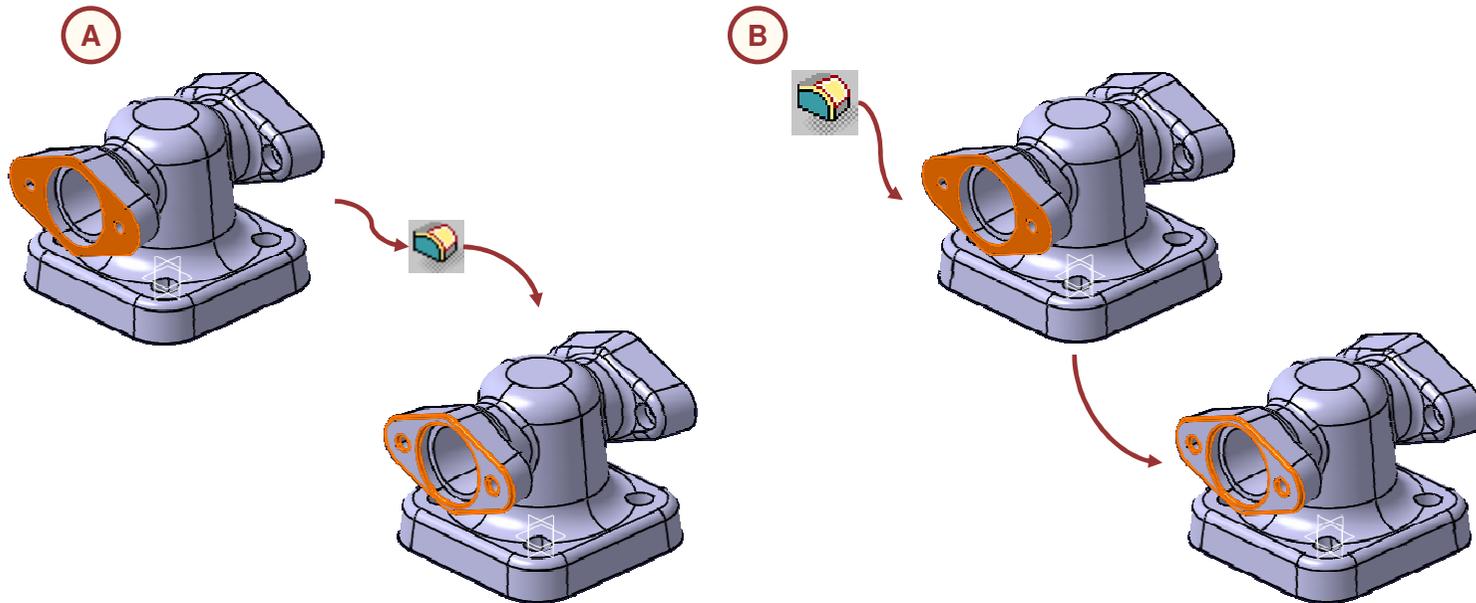
To display the Pre-selection Navigator automatically, select **Tools > Options** then the Navigation tab. Check the Pre-selection Navigator button.



The Object/Action and Action/Object Approaches

CATIA is different than other Windows application. You can control the order of selection for the elements and tools. You can either:

- A. First select the object(s) to be processed, then select the tool required to perform the operation.
- B. First select the tool required to perform the operation, then select the object(s) to be processed.

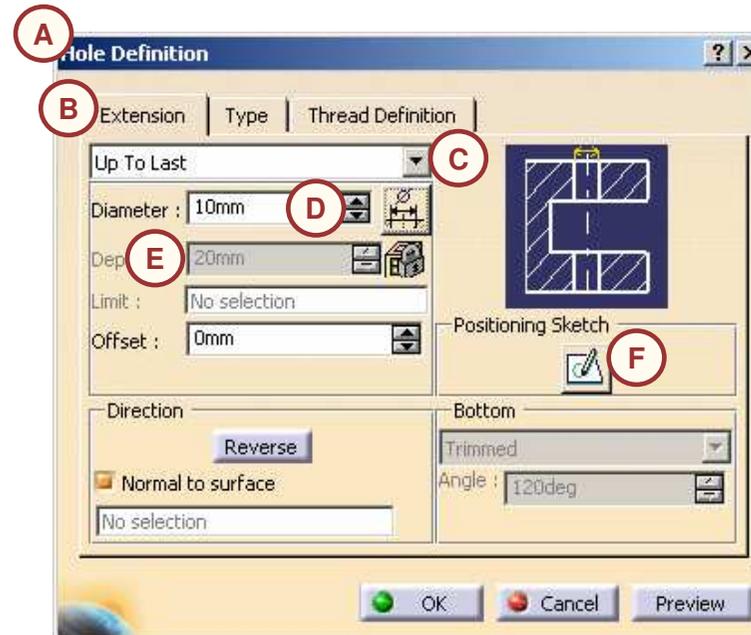


Using the CATIA Dialog Boxes (1/2)

The CATIA dialog boxes provide various parameters for the definition of features. They are standardized and easy-to-use to define the inputs for a feature or a process. For example, the Hole Definition dialog box is shown below.

Following is a list of common items you might find in a CATIA dialog box:

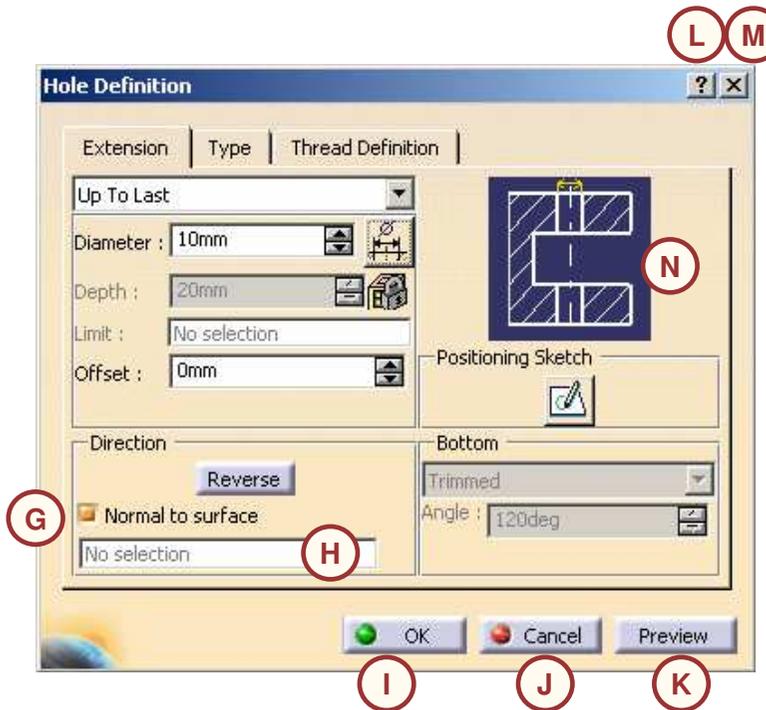
- A. Name of the dialog box
- B. Tabs that group similar types of parameters
- C. Drop-down list box for various options
- D. Spin box to increase or decrease the input value
- E. Locked fields to protect data
- F. Iconic buttons to activate other actions



Using the CATIA Dialog Boxes (2/2)

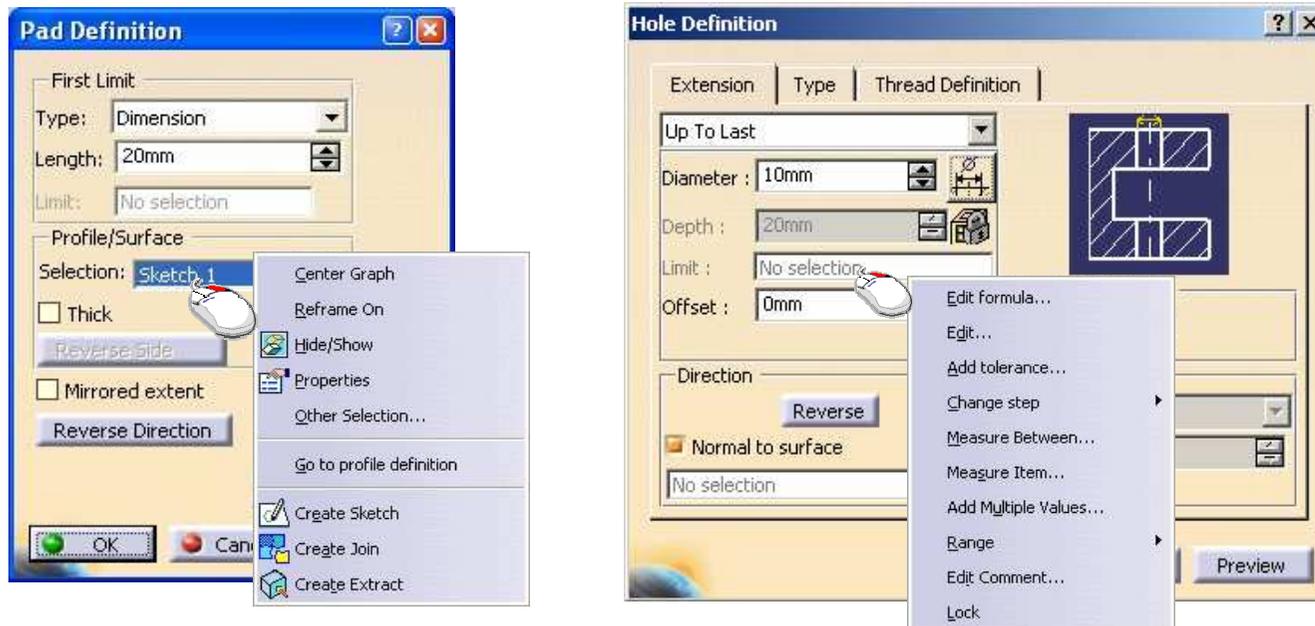
The following lists common items you might find in a window (continued):

- G. Check boxes to activate the options
- H. Inactive fields that do not apply to a particular case
- I. **OK** button to confirm the action
- J. **Cancel** button to quit the command
- K. **Preview** button to pre-visualize the result
- L. Help for the dialog box dialog box
- M. **Close** button to close the dialog box
- N. Visual Aid Assistants to show the intent of the selected options



Using Dialog Boxes and Right-click

Data can also be entered into certain fields of a dialog box by clicking the right mouse button (*right-clicking*) on them. The options that appear in the contextual menu are dependent on the type of data that the field is capable of receiving.



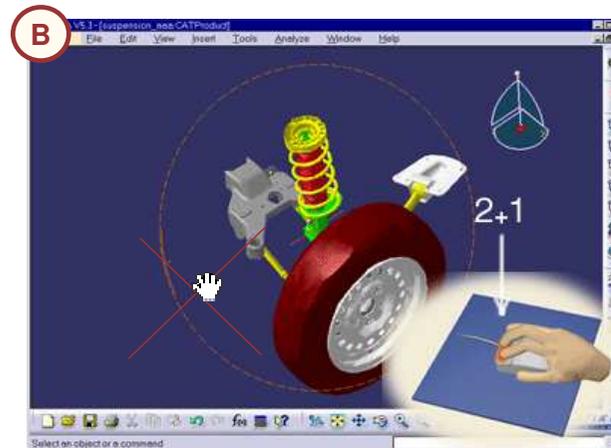
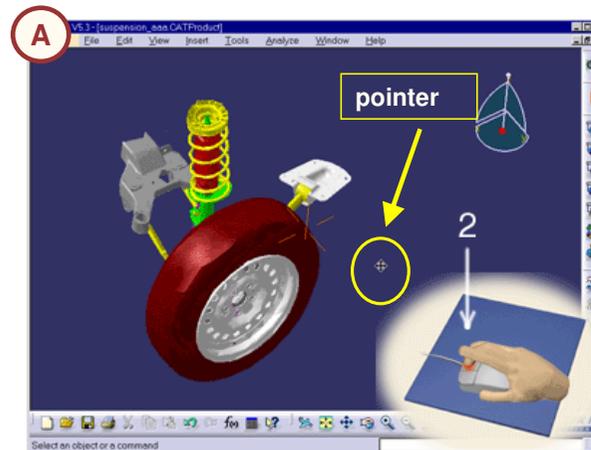
Moving Objects with the Mouse (1/3)

Since CATIA is a 3D environment, you can view the model in various ways.

The easiest way to change the view of a model is to zoom, pan, and rotate it.

- A. Panning enables you to move the model on a plane parallel to the screen. It involves dragging the mouse while clicking and keeping the middle (#2) mouse button pressed.

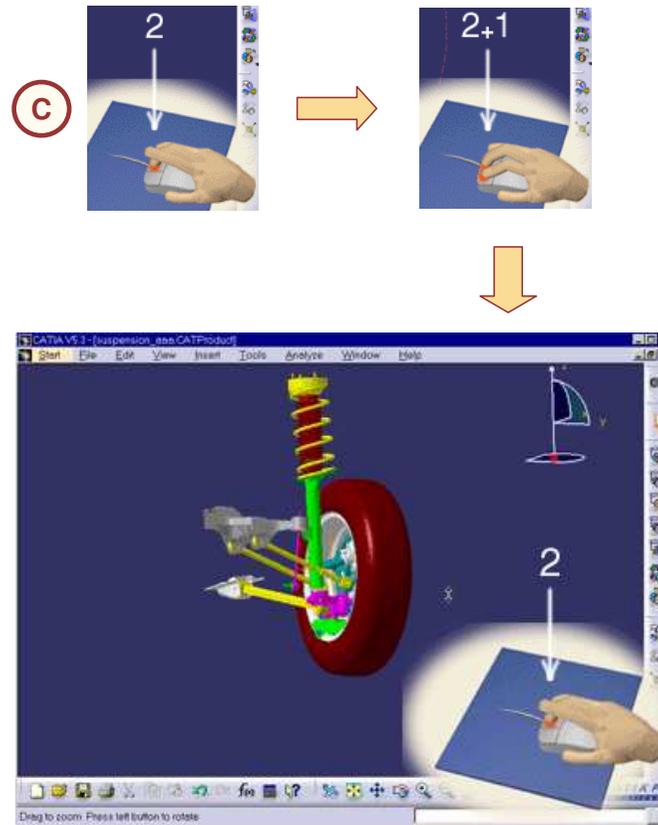
- B. Rotating enables you to turn the model around a point. It involves dragging the mouse while clicking and keeping the middle (#2) and left (#1) mouse buttons pressed.



Moving Objects with the Mouse (2/3)

The easiest way to change the view of a model is to zoom, pan, and rotate it (continued...)

- C. Zooming enables you to move closer to the model or away from it along a plane perpendicular to the screen. It involves clicking and keeping the middle (#2) mouse button pressed while clicking the left (#1) mouse button once, and then dragging the mouse up (to move closer) and down (to move away)."

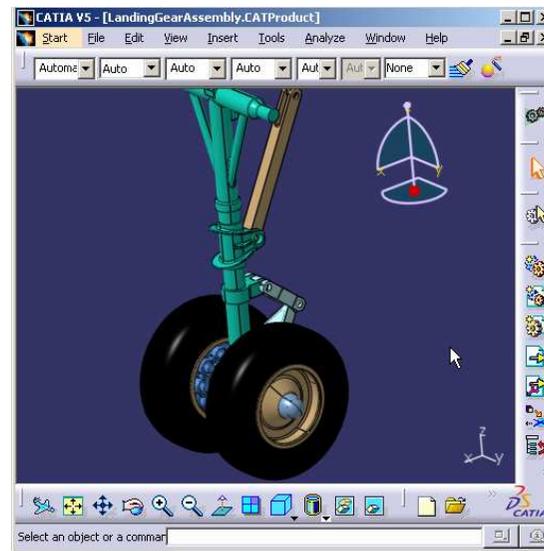
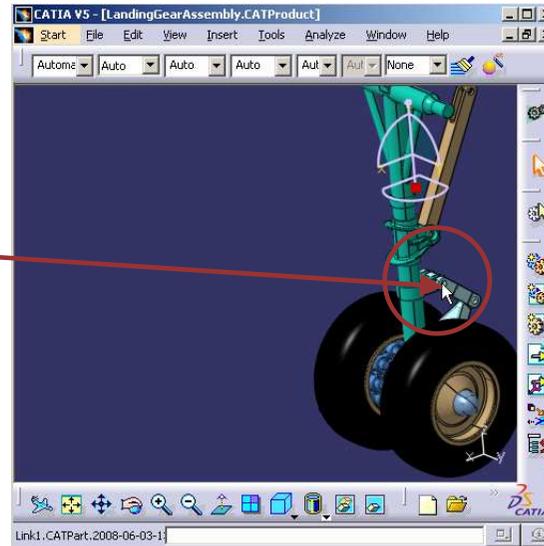
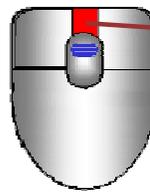


Student Notes:

Moving Objects with the Mouse (3/3)

To centralize an object on the screen simply click the middle (#2) mouse button on the geometry that you wish have at the center of the screen. The orientation and the scale of the view will remain unchanged.

To see all displayed objects, click the **Fit All In** tool.

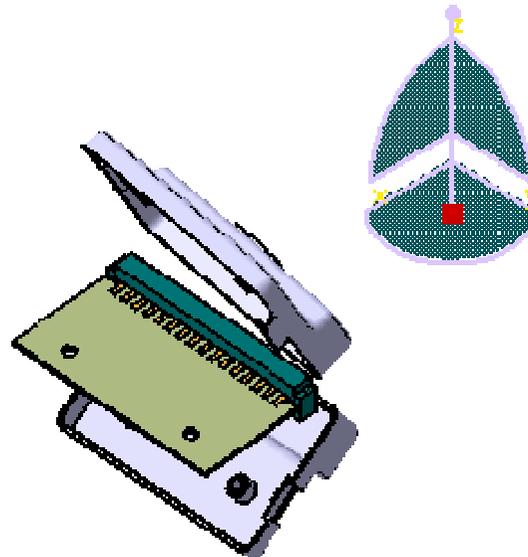


Compass (1/2)

Besides being an orientation reference tool that helps while performing view rotations, the Compass is a powerful tool that can be used to physically move and manipulate objects. This is especially useful in Assembly Design, Freestyle, and Digital Mockup workbenches.

By default, the base of the Compass (also called the Privileged Plane) is the XY plane. The default orientation of the compass is parallel to the reference XYZ axis system and is located to the top right corner of the screen.

If the 3D view is changed, the Compass updates its orientation to reflect the new viewing angle/direction.

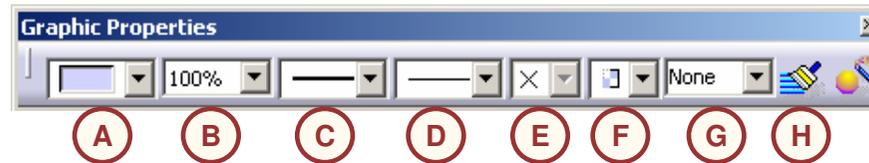
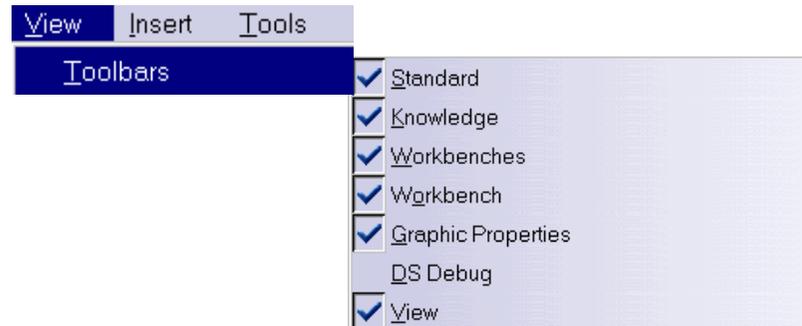


Graphic Properties

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:

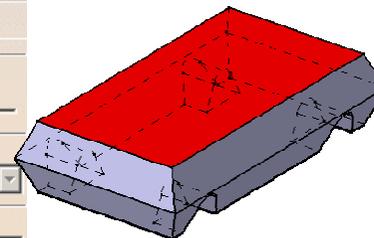
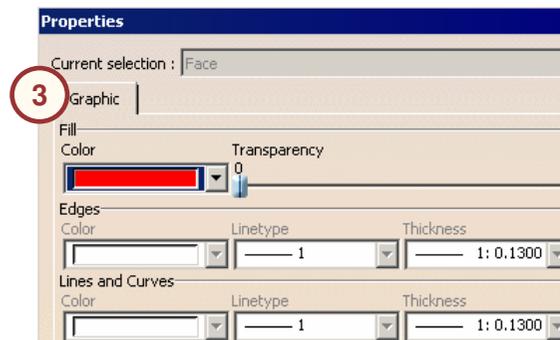
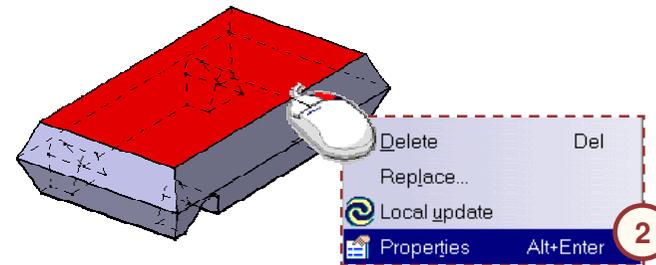
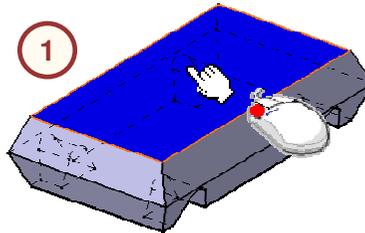
- A. Fill color
- B. Transparency
- C. Line thickness
- D. Line type
- E. Point symbol
- F. Rendering style
- G. Active Layer
- H. The Painter tool copies the graphic properties of one feature to another



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.

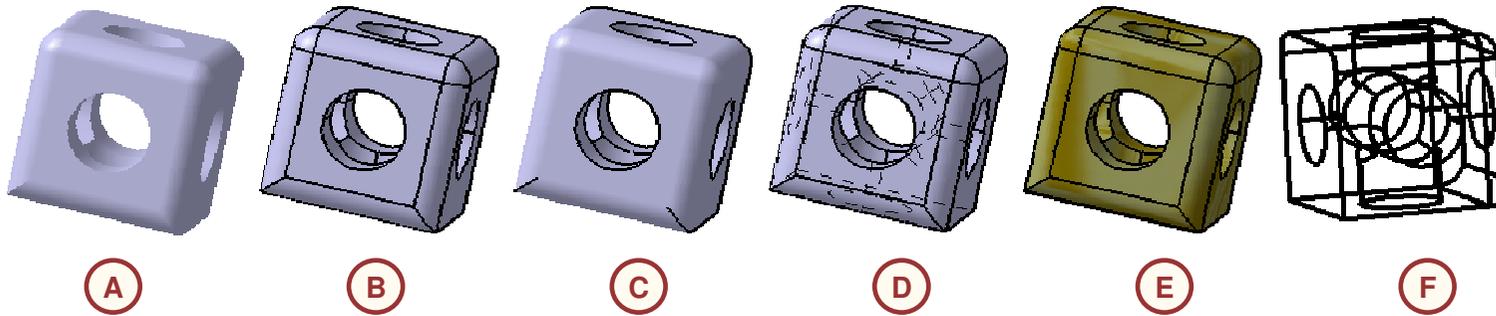
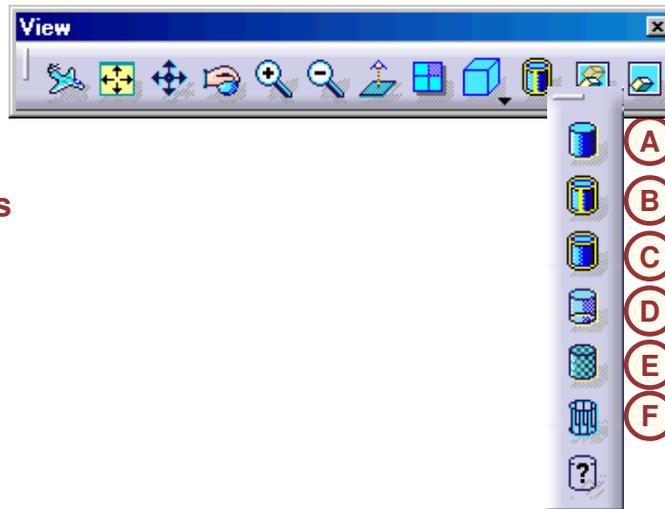


Student Notes:

Rendering Styles

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

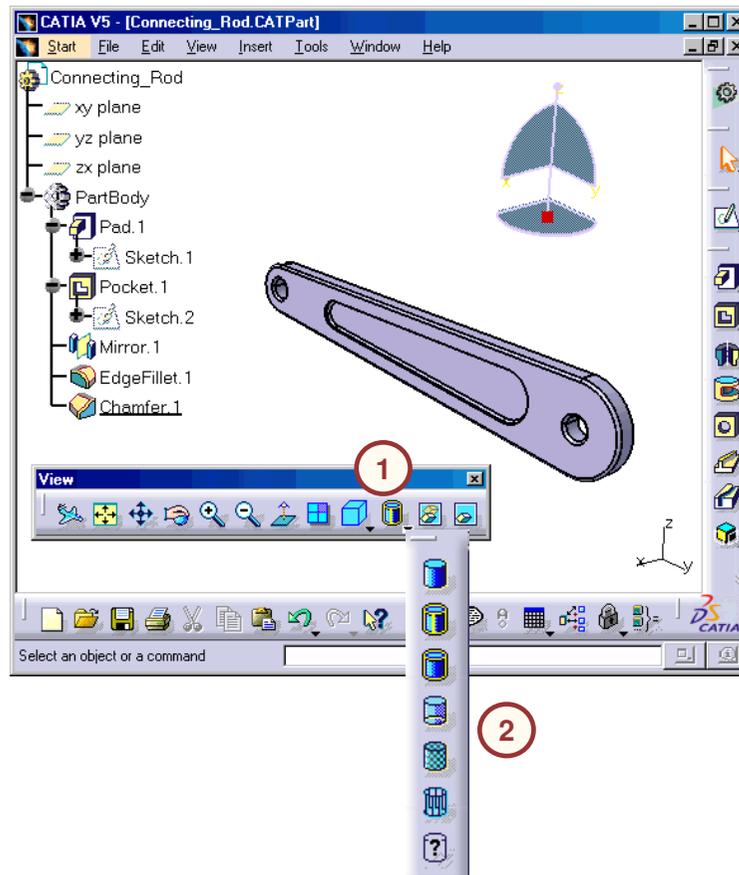


Student Notes:

Applying Rendering Styles

Use the following steps to apply a rendering style:

1. Select the current rendering style icon in the View toolbar.
2. Select the new rendering style to apply. It is applied automatically to the geometry.

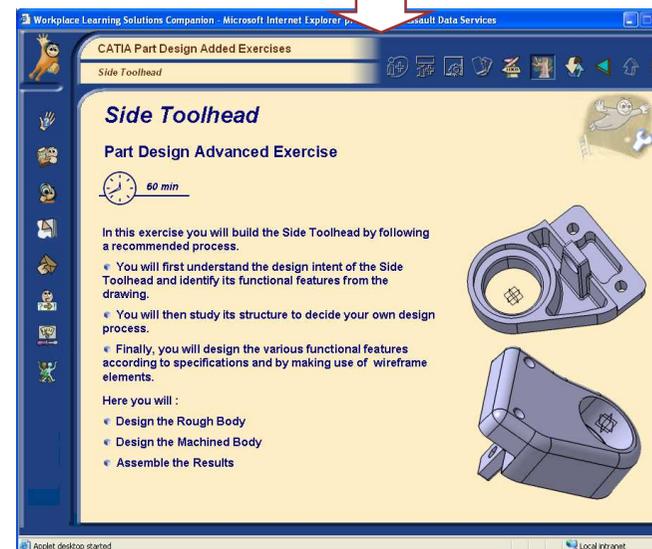
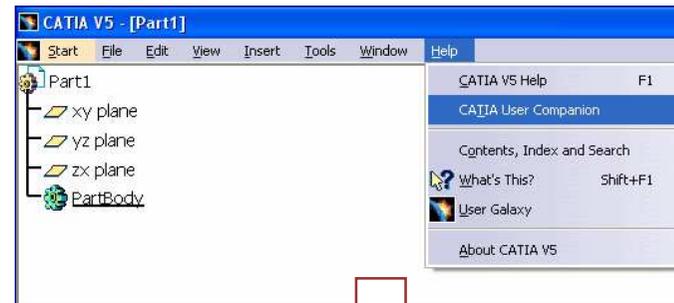


CATIA User Companion

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:

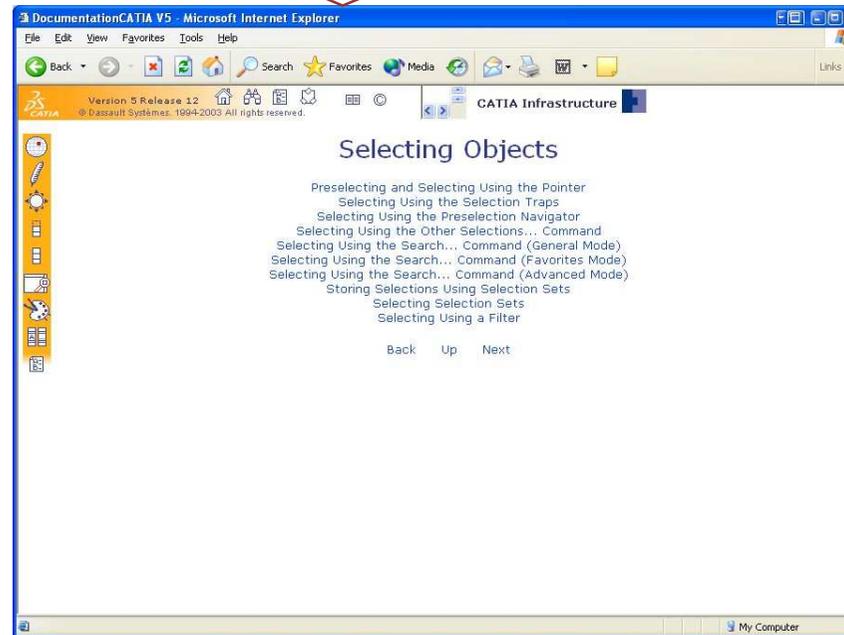
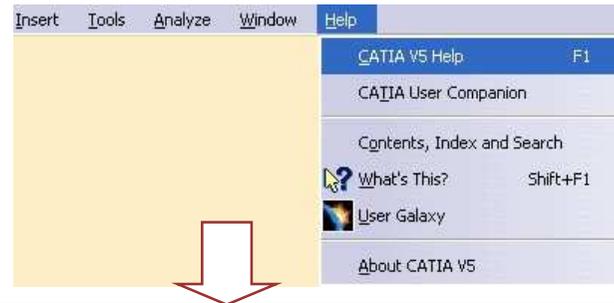
- CATIA V5 (Mechanical Design, Hybrid Design, Generative Sheet Metal Design, Analysis)
- DMU
- ENOVIA
- SMARTEAM



Help Documentation

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

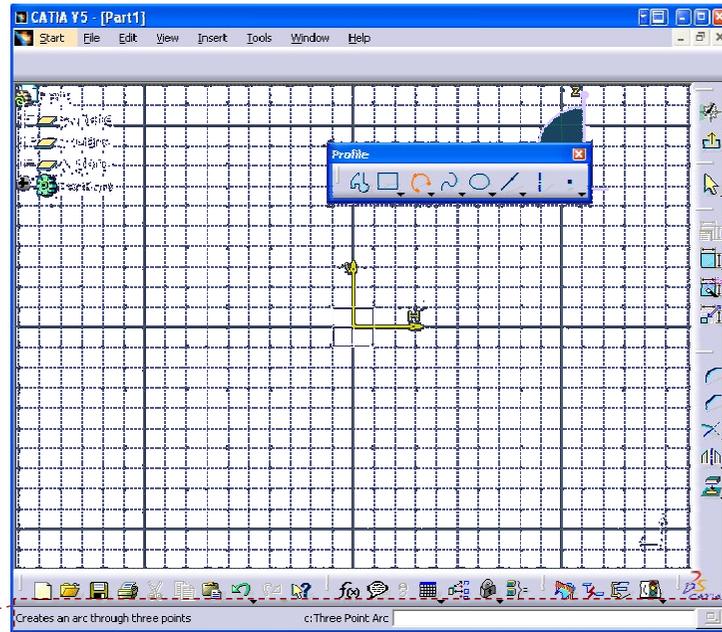
The help system is a self-contained series of HTML documents that break down the vast CATIA information into workbenches, products, and solutions.



Message Bar

Another useful tool that can help 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.



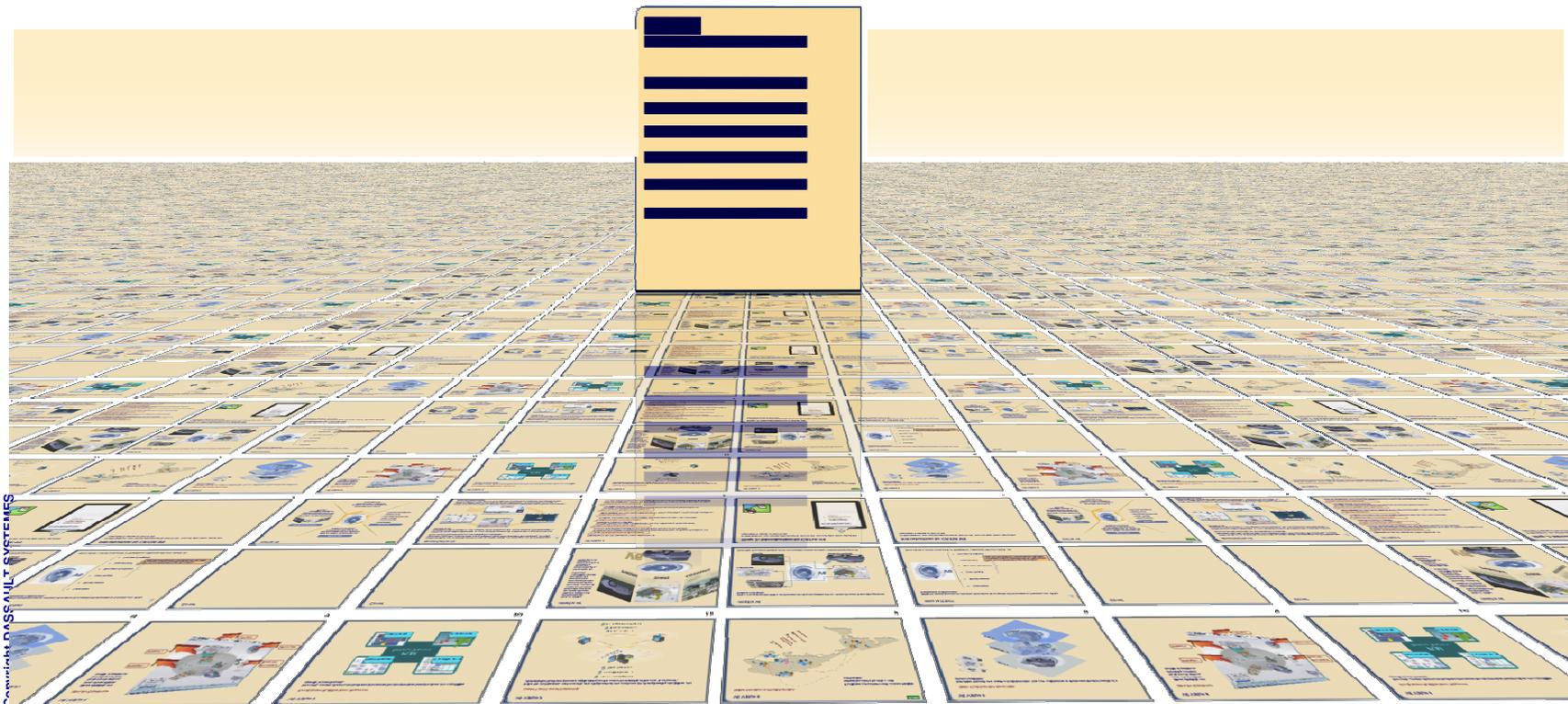
Creates an arc through three points

c:Three Point Arc

To Sum Up

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

Student Notes:



Copyright DASSAULT SYSTEMES

Student Notes:

Understand the CATIA software

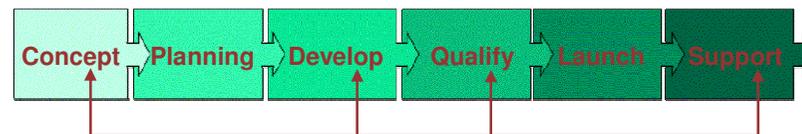
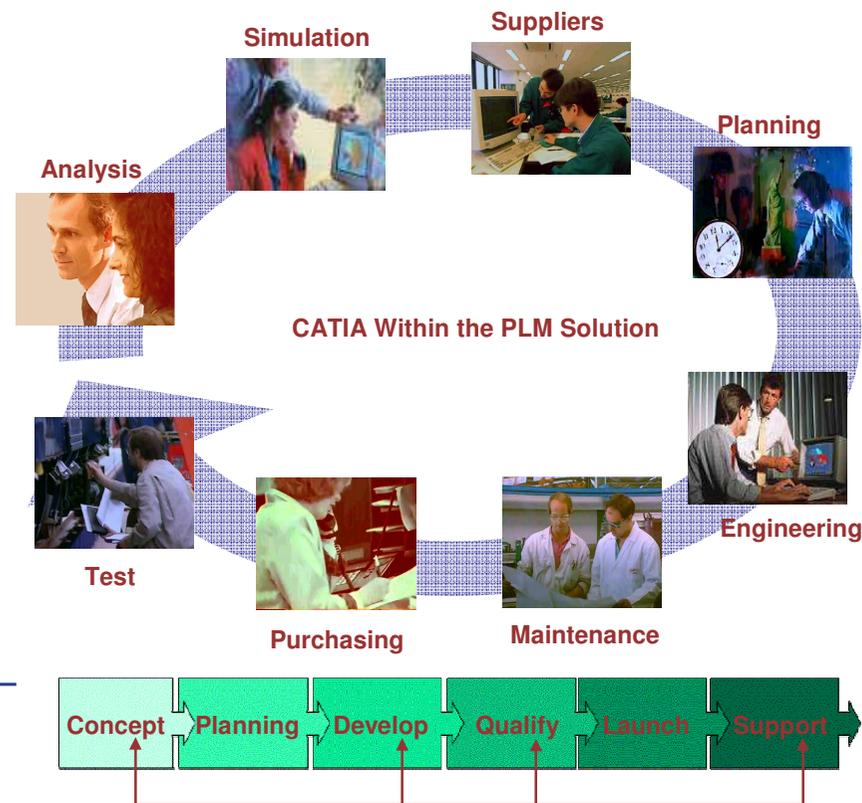
CATIA is a mechanical design software. It is a feature-based, parametric solid modeling design tool that takes advantage of the easy-to-learn Windows graphical user interface. You can create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations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

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



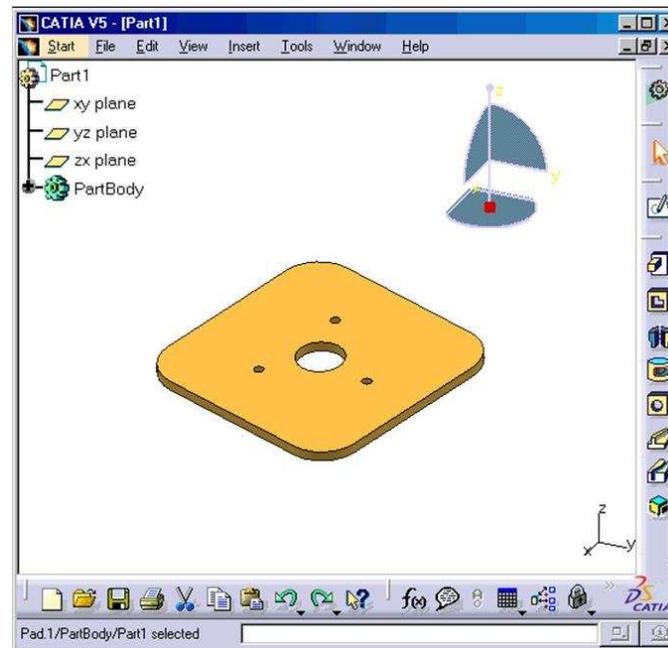
CATIA's Scope From Concept To Realization

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.



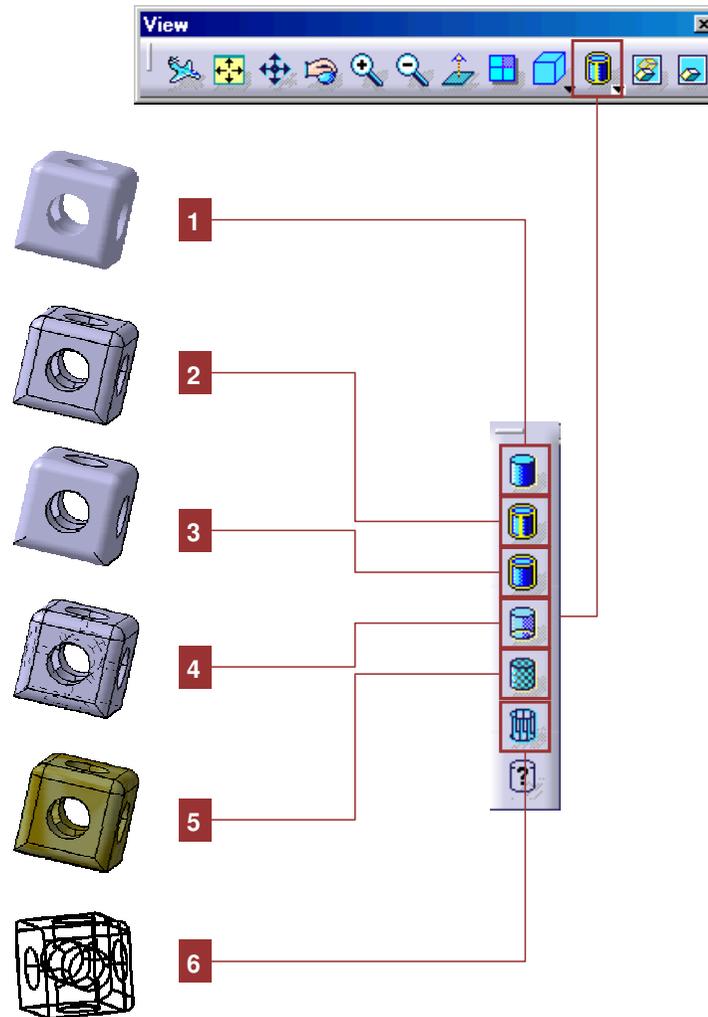
Student Notes:

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: CATIA V5 User Interface

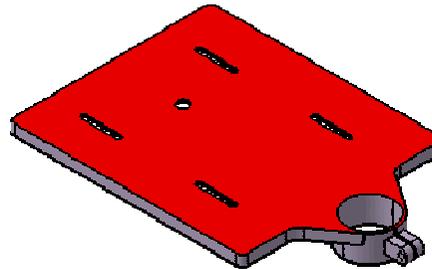
Recap Exercise



In this exercise, you will start CATIA and open existing documents within a file based environment.

By the end of this exercise you will be able to:

- Launch CATIA
- Open an existing document
- Identify the fundamental CATIA document types
- Close a file



Do it Yourself (1/5)

1. Start the CATIA application.

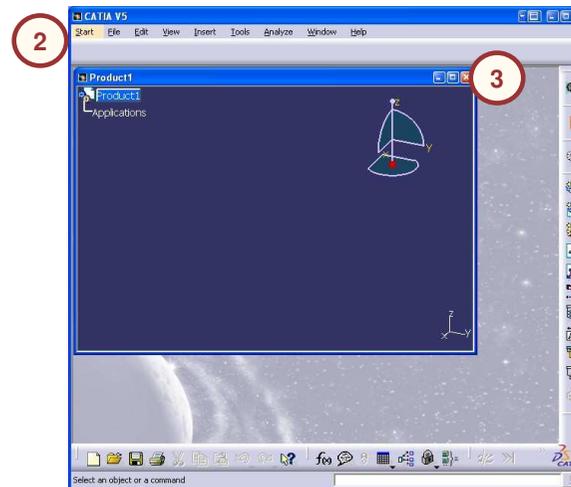
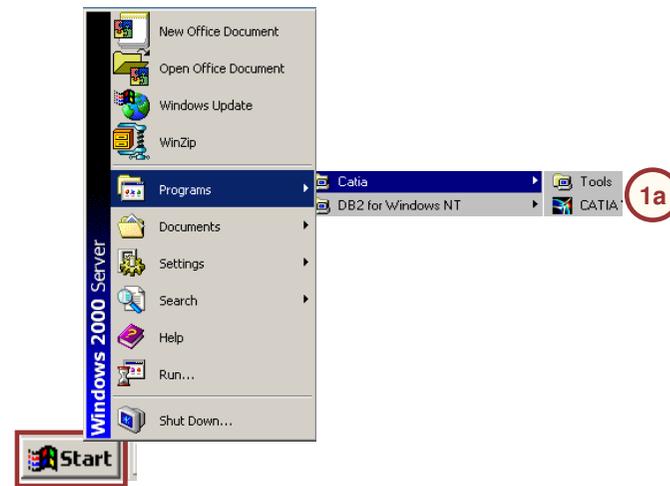
- Start the application using the Windows START menu.
 - a. Double-click on the **CATIA** icon on your desktop or select it through the **START > Program** files menu.

2. Review the default window and document.

- Review the default CATIA window.
 - a. Notice that CATIA opens a CATProduct document by default and automatically launches the Assembly Design workbench.
 - b. Examine the various tools and menus that appear. Try to locate the Design Table tool by using the tool tips.

3. Close the Product1.CATProduct window.

- Select **Close** to close Product1.



Student Notes:

Do it Yourself (2/5)

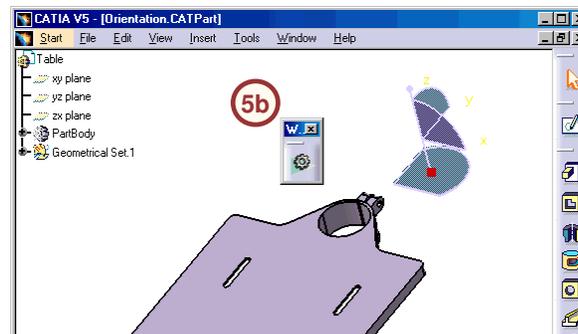
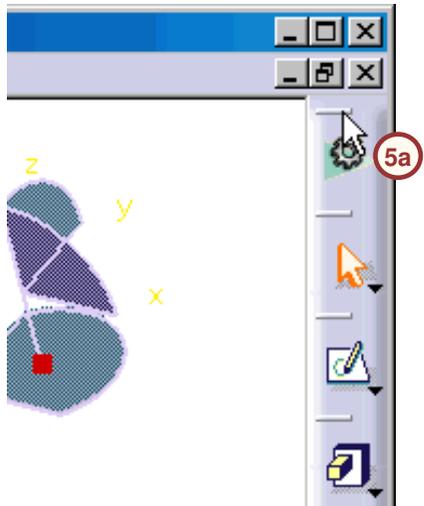
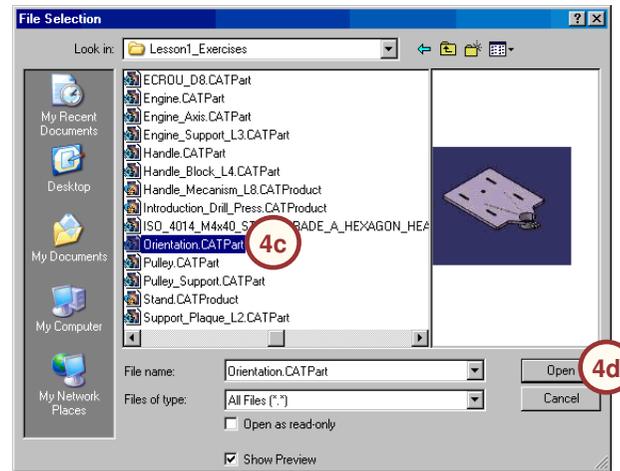
4. Open a part file.

- Open a file using the **Open** icon.
 - a. Click **File > Open**.
 - b. Browse to the Orientation.CATPart.
 - c. Highlight the file.
 - d. Select **Open**.



5. Customize the toolbars.

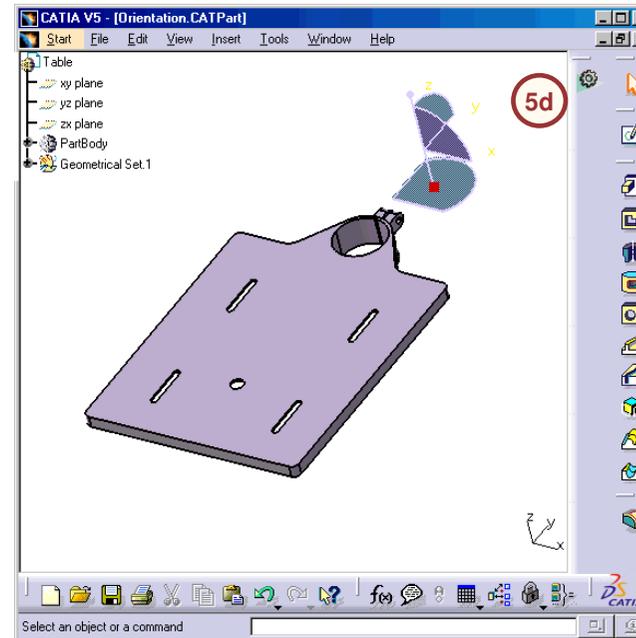
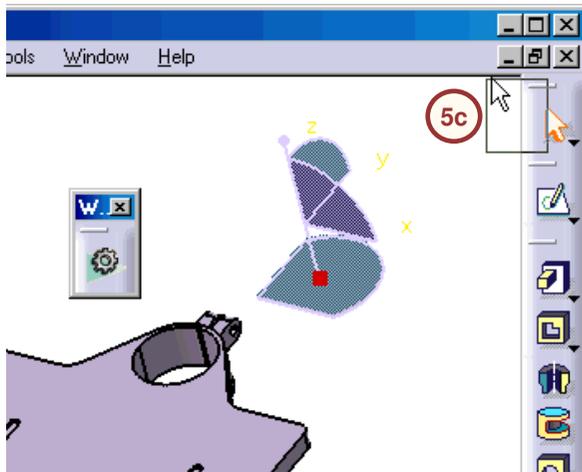
- a. Press and hold the left mouse button on the Workbench toolbar separator.
- b. Drag the toolbar to the middle of the screen. The toolbar is now “floating”.



Do it Yourself (3/5)

5. Customize the toolbars (continued):

- c. Drag the toolbar back to the right-hand side of the screen until the outline changes thickness.
- d. Release the left mouse button. A second row of toolbars is added to the right-hand side.
- e. Move other toolbars to customize your screen.

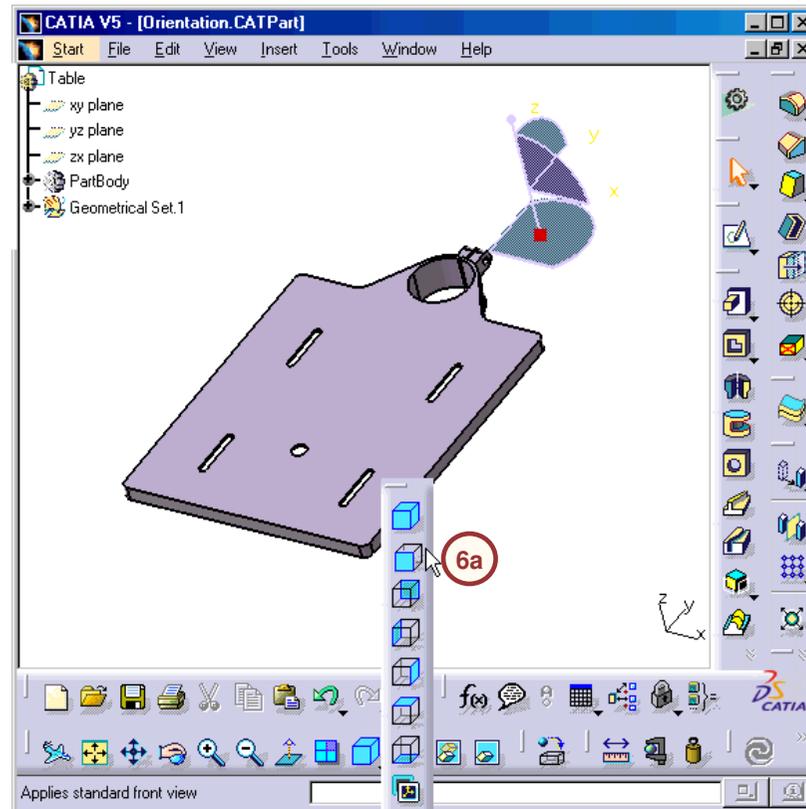


Student Notes:

Do it Yourself (4/5)

6. Change the Orientation.

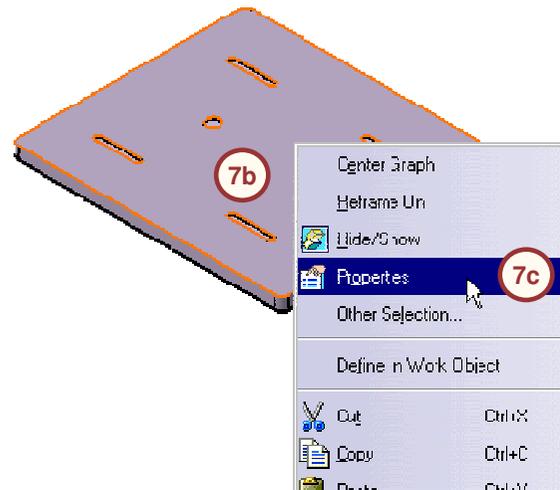
- Use the mouse and the view icons to change the orientation of the model.
 - a. Select the **Front View** icon.
 - b. Press and hold the middle mouse button to pan the model.
 - c. Press and hold the middle and the left mouse button to rotate the model.
 - d. Press and hold the middle mouse button, and click the left mouse button to zoom.
 - e. Experiment with the other view orientations, using the mouse buttons and View icons.



Do it Yourself (5/5)

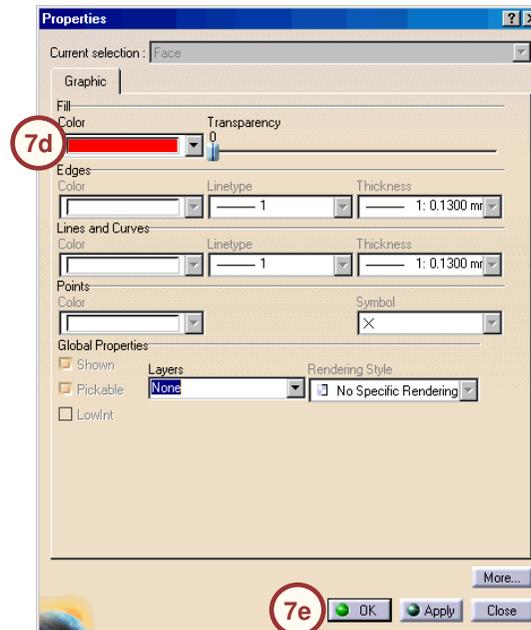
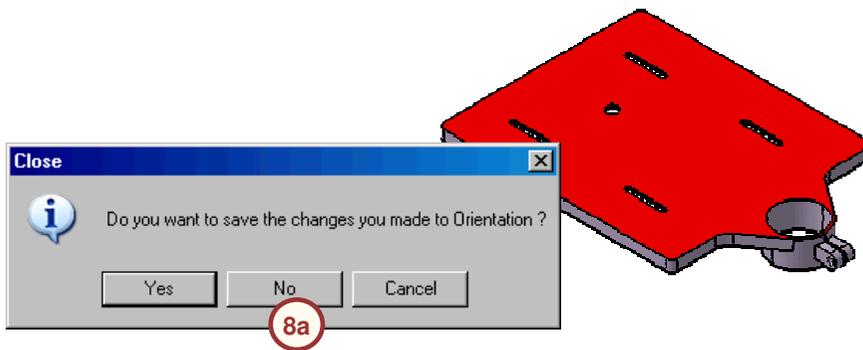
7. Change the properties.

- Make the top surface of the model red.
 - a. Select the **Isometric View** icon.
 - b. Select the top face of the table.
 - c. Right mouse click and click Properties.
 - d. Change the color of the surface to Red.
 - e. Select **OK** to apply the change and close the window.



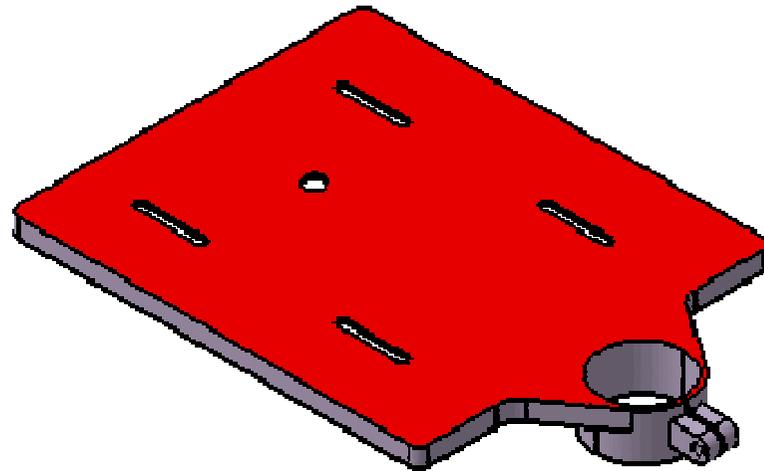
8. Close the file.

- a. Click **File > Close** to close the part file.
- b. Select **No** to the Close window.



Exercise Recap: CATIA V5 User Interface

- ✓ Launch CATIA
- ✓ Open an existing Document
- ✓ Identify the fundamental of CATIA document types
- ✓ Close a file



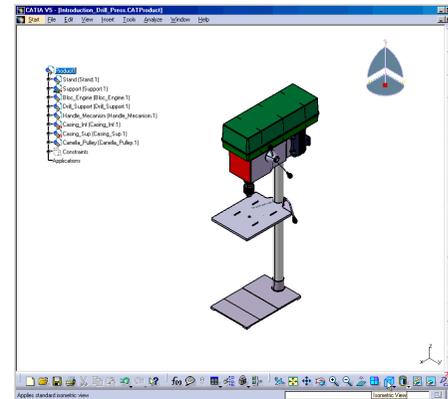
Case Study: Introduction to CATIA

Recap Exercise



In this exercise, you will review the master project assembly. With the knowledge you have learned in this lesson, you should be able to:

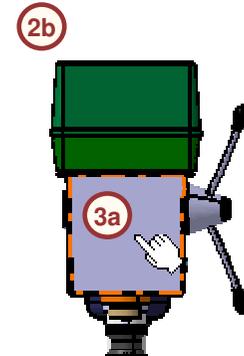
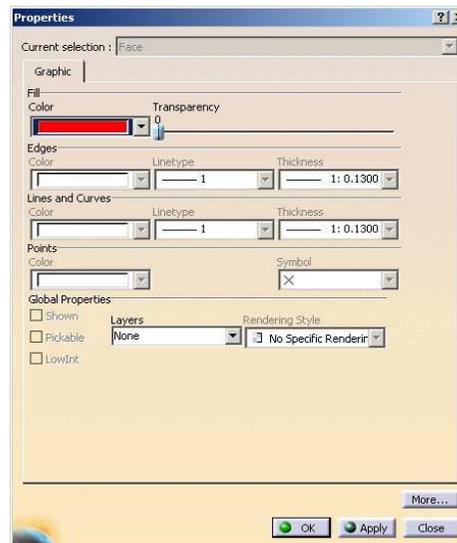
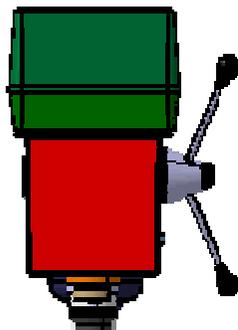
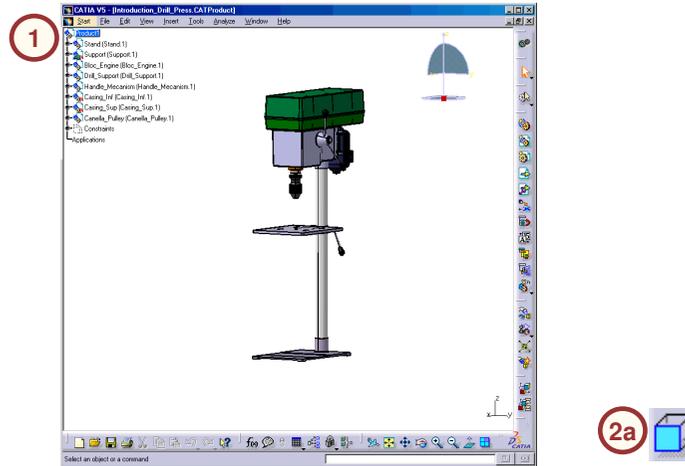
- ✓ Change the orientation of a model
- ✓ Change the visualization properties of a model
- ✓ Manipulate the specification tree
- ✓ Access the CATIA help system



Student Notes:

Case Study (1/6)

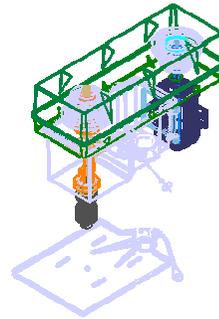
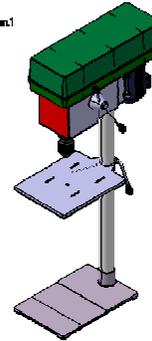
1. Open Introduction_Drill_Press.CATProduct.
2. Change the Orientation of the assembly.
 - a. Change the model orientation to Front.
 - b. Zoom in on the area as shown.
3. Change the Visualization properties.
 - a. Select the front face of the support part as shown and change the color to red.



Student Notes:

Case Study (2/6)

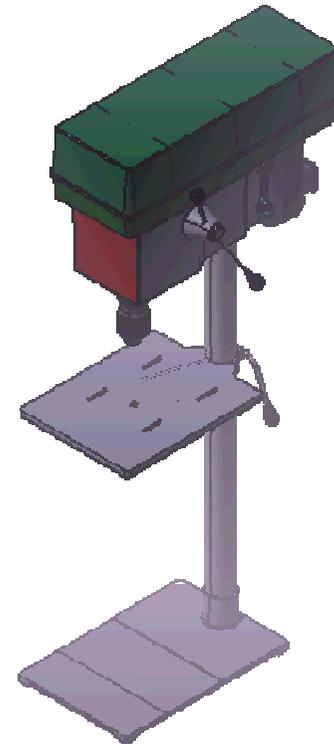
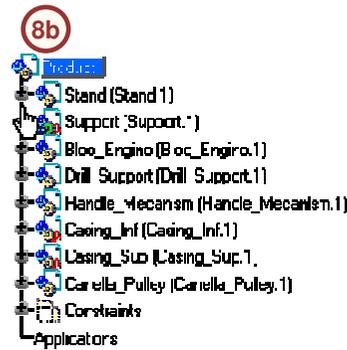
4. Zoom out on the model.
5. Change to the Isometric view.
6. Change the rendering style to Wireframe.
7. Change the rendering style to Shading with Edges.



Case Study (3/6)

8. Change the display of the specification tree.

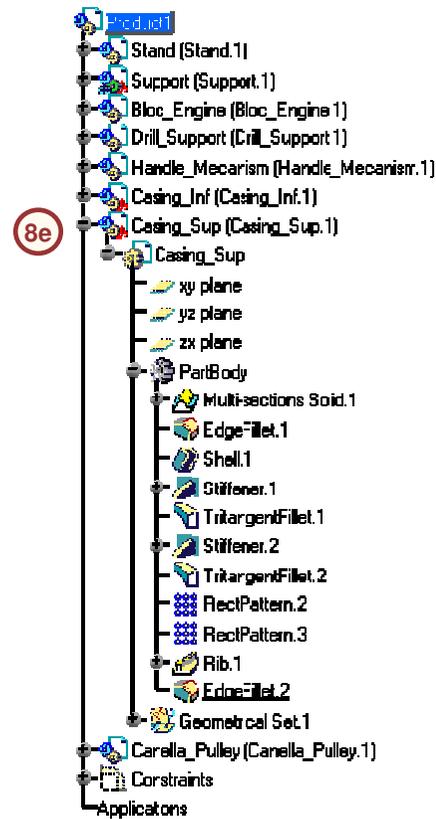
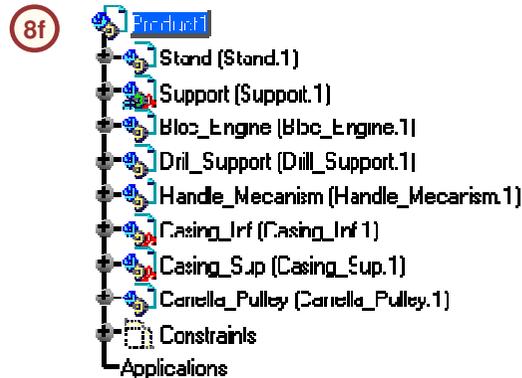
- a. Press the <F3> key to toggle the specification tree on and off.
- b. Select one of the branches of the specification tree and notice that the model darkens.
- c. Try zooming out; notice that the specification tree is being manipulated and not the model.
- d. Press <Shift> and <F3> to re-activate the model.



Case Study (4/6)

8. Change the display of the specification tree (continued)

- e. Expand the Casing_sup node of the tree. Notice that the features of the part are now displayed in the tree.
- f. Collapse all the nodes to show only the top level of the tree.

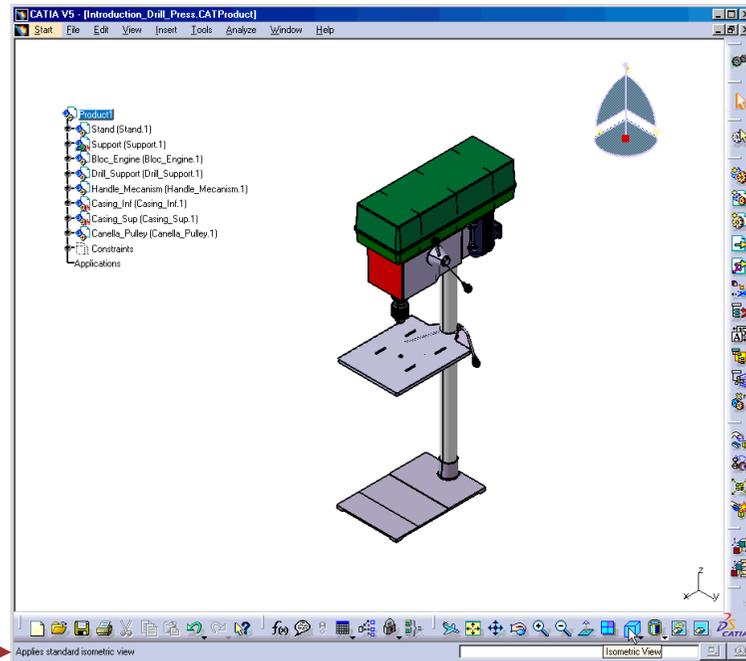


Student Notes:

Case Study (5/6)

9. Review the areas of information.

- a. Hover the mouse over the various tools and areas of the toolbars and notice the tool tip comments.
- b. Review the message bar as you pause the mouse cursor over the icons for more information.



Case Study (6/6)

9. Review the areas of information (continued).

- c. Click **Help > CATIA V5 Help**.
- d. A web browser launches, with the CATIA Help start page loaded.
- e. Spend few minutes browsing the various links within the system.



10. Close the assembly without saving changes.



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 CATIA help system

