



## CATIA V5 Fundamentals

### Course Curriculum (Duration: 60 Hrs.)

#### 1.1] Welcome to CATIA V5

- a. CATIA V5, the New Generation
- b. The Windows Philosophy
- c. PLM (Product Life Cycle Management)
- d. PLM in Practice
- e. CATIA Within PLM
- f. What is CATIA V5?
- g. Design Intent
- h. How features affect Design Intent?

#### 1.2] The CATIA V5 Environment

- a. CATIA User Interface
- b. The Workbench Concept
- c. Workbench Concept: Recommendations
- d. Introduction to V5 Documents

#### 1.3] Launching CATIA and Manipulating Objects

- a. Starting CATIA V5
- b. Menus and Toolbars
- c. Manipulating Objects
- d. Manipulating Objects: Recommendations
- e. Accessing the Toolbars and Dialog Boxes
- f. What to do when a Tool cannot be found?
- g. CATIA User Interface: Recommendations
- h. Saving and Closing Documents
- i. Starting and Working with CATIA: Recap Exercises

#### 1.4] The Specification Tree

- a. What about the Specification Tree
- b. Manipulating the Tree
- c. To Sum Up

#### 1.5] Objects Visualization

- a. Selecting objects with the Mouse
- b. Hiding and Showing Objects
- c. The Object/Action and Action/Object Approaches
- d. Using Dialog Boxes
- e. Moving Objects with Mouse
- f. What is the Compass?
- g. Graphic Properties
- h. Changing the Graphic Properties
- i. Rendering Styles
- j. Objects Visualization: Recap Exercises
- k. CATIA User Companion
- l. Help Documentation
- m. Message Bar

#### 1.6] Profile Creation

- a. Create a new Part
- b. Select and Appropriate Sketch Support
- c. Create Sketched Geometry



- d. Constrain the Sketch
- e. Create Pad Feature
- f. Save and Close the Document

#### **1.7] Basic Features**

- a. Determine a suitable Base Feature
- b. Create Pad and Pocket Features
- c. Create Holes
- d. Create Fillets and Chamfers

#### **1.8] Additional Part Features**

- a. Create Feature Profiles and Axis System
- b. Create Multi-profile Sketch Features
- c. Create Wireframe Geometry
- d. Create Shaft and Groove Features
- e. Shell the Model

#### **1.9] Dress-up Features**

- a. Apply a Draft
- b. Create a Stiffener
- c. Create Threads and Taps
- d. Edit Features

#### **1.10] Reusing Data**

- a. Duplicate Features
- b. Transform a Body
- c. Copy and Paste the Data
- d. Insert Data from a Catalog

#### **1.11] Finalizing Design Intent**

- a. Apply Material Properties
- b. Analyze the Model
- c. Create Formulas and Parameters

#### **1.12] Assembly Design**

- a. Create a new CATProduct
- b. Assemble in the Base Component
- c. Manipulate the position of the Component
- d. Assemble and Fully Constrain Components
- e. Save the Assembly

#### **1.13] Designing in Context**

- a. Open an existing assembly
- b. Insert a new model
- c. Create a Sketch in Context
- d. Create Assembly-Level Features

#### **1.14] Drafting:**

- a. The Drawing Environment
- b. Drafting Toolbars and Objects
- c. Start a New Drawing
- d. Setting the Drawing Sheet Format and Drafting Standards
- e. Starting a Drawing with a Blank Sheet
- f. Sheet Properties
- g. Drawing Title Blocks
- h. 2D Catalogs
- i. Inserting Catalog Items
- j. Create Views
- k. Types of Views
- l. Using the Compass
- m. Projection Plane / Views
- n. Using Wizard



- o. CGR Views
- p. Dimensions and Annotations
- q. Section Views and Section Cuts
- r. Detail, Clipping, Broken, Breakout and Auxiliary View
- s. Modifications
- t. Check links to 3D Parts
- u. Printing

**1.15] Exercises and Workshops**

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