



CATIA V5 Mechanical Designer

Course Curriculum (Duration: 80 Hrs.)

Chapter 1: CATIA V5 Fundamentals

1.1] Introduction to CATIA

- a. Design Intent
- b. Understand CATIA Software
- c. PLM-Product Life Cycle Management
- d. CATIA within the PLM Solution
- e. CATIA Coverage
- f. What is CATIA V5?
- g. Open CATIA
- h. Starting CATIA using the Start Menu
- i. Opening an existing Document
- j. Understand the CATIA Interface
- k. The Windows Philosophy
- l. Introduction to V5 Documents
- m. The Workbench Concept
- n. CATIA User Interface
- o. Workbenches
- p. Menus and Toolbars
- q. The Specification Tree
- r. Manipulating the Specification Tree
- s. Selecting Objects with the Mouse
- t. The Object/Action and Action/Objects Approaches
- u. Using Dialog Boxes
- v. Moving Objects with the Mouse
- w. What is the Compass?
- x. Graphic Properties
- y. Changing the Graphic Properties
- z. Rendering Styles
- aa. Hide/Show
- bb. Help Documentation
- cc. Message Bar

1.2] 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

1.3] Basic Features

- a. Determine a suitable Base Feature
- b. Create Pad and Pocket Features



- c. Create Holes
- d. Create Fillets and Chamfers

1.4] 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

1.5] Dress-up Features

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

1.6] Reusing Data

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

1.7] Finalizing Design Intent

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

1.8] 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

1.9] Designing in Context

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

1.10] Drafting:

- a. Introduction to Generative Drafting
- b. General Process
- c. Accessing the Workbench
- d. The Drawing Environment
- e. Drafting Toolbars and Objects
- f. Start a New Drawing
- g. Setting the Drawing Sheet Format and Drafting Standards
- h. Starting a Drawing with a Blank Sheet
- i. Sheet Properties
- j. Title Blocks
- k. 2D catalogs
- l. Inserting Catalog Items
- m. Create Views
- n. Types of Views
- o. About Projection Plane
- p. CGR Views



- q. Create Dimensions and Annotations
- r. Section Views and Section Cuts
- s. Adding a Detail View, Clipping View, Broken View, Breakout View
- t. View Modifications
- u. Save the Drawing
- v. Print the Drawing

1.11] Exercises and Workshops

Chapter 2: CATIA V5 Mechanical Design Expert

2.1] Introduction

- a. Introduction
- b. Review the User Interface
- c. CATIA Workbenches
- d. Part Design Workbench
- e. Assembly Design Workbench
- f. Generative Shape Design Workbench
- g. Importance of Parent/Child Relationships
- h. Investing the Model
- i. What is Define in Work Object?
- j. Defining in Work Objects
- k. Organizing a Solid Model
- l. Model Organization
- m. Geometrical Sets
- n. Ordered Geometrical Sets
- o. Hybrid Design

2.2] Design Complex Parts

- a. Design Intent
- b. Create Advanced Sketch-Based Features
- c. Multi-Sections Solid
- d. Create Advanced Drafts
- e. Advance Dress-Up Features
- f. Use the Multi-Body Method
- g. Create Multi-Model Links

2.3] Analyze and Annotate Parts

- a. Analyze the Part
- b. Create 3D Constraints
- c. Annotate the Part

2.4] Sharing Information

- a. Create a Power Copy
- b. Create Parameters and Formulas
- c. Create a Design Table
- d. Create a Catalog

2.5] Assembly Design

- a. Manage the product structure
- b. Create flexible sub-assemblies
- c. Analyze the assembly
- d. Create scenes
- e. Create annotations
- f. Generate reports



2.6] Contextual Design

- a. Clarify the display
- b. Create Contextual Parts
- c. Create Assembly-Level Features
- d. Manipulate the Contextual Components
- e. Save the Contextual Models

2.7] Complex Assembly Design

- a. Create a Skeleton Model
- b. Create the Published Elements
- c. Use the Published Elements

2.8] Exercises and Workshops

Chapter 3: CATIA Surface Design (Generative Shape Design)

3.1] Introduction to Surface Design

- a. Introduction to Surface Design
- b. The Generative Shape Design Workbench
- c. Surface Design Workbench User Interface
- d. Surface Design Workbench Terminology
- e. Surface Design Workbench General Process

3.2] Creating Wireframe Geometry

- a. Why create 3D Wireframe Geometry?
- b. Creative Points, Lines, Planes and Curves in 3D
- c. Creating Wireframe Geometry
- d. To Sum Up

3.3] Shape Design Common Tools

- a. Why use common Tools?
- b. Stacking Commands
- c. Managing Geometrical Sets
- d. To Sum Up

3.4] Creating Surfaces

- a. Why create Surface Geometry?
- b. Creating Basic Surfaces
- c. Creating a Swept Surface
- d. Creating a Surface Offset from a Reference
- e. Creating a Surface from Boundaries
- f. Creating a Multi-Section Surface

3.5] Performing Operations on the Geometry

- a. Why are operations on Geometry needed?
- b. Joining Elements
- c. Splitting/Trimming
- d. Creating Fillets
- e. Transforming Elements
- f. Extrapolating Elements
- g. Disassembling Elements
- h. Additional Methods for Operations

3.6] Completing the Geometry in Part Design

- a. Why complete geometry in part design?
- b. Creating a Solid from surfaces?
- c. Completing Geometry recommendations



3.7] Modifying the Geometry

- a. What about modifying the Geometry?
- b. Editing Surface and Wireframe Definition
- c. Modifying Geometry Recommendations

3.8] Using Tools

- a. What about using Tools?
- b. Creating Datum Features
- c. Checking Connections between Elements
- d. Updating a part

3.9] Exercise and Workshops

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