



# CATIA V5 for Mechanical Engineers

**Duration: 55 Days**

## **Syllabus Content**

### **Module 1: CATIA v5 Fundamental**

#### **a) Introduction To CATIA v5**

- Understand CATIA Software
- What is CATIA V5?
- Starting CATIA using the Start Menu
- Opening an existing Document
- Understand the CATIA Interface
- The Workbench Concept
- Menus and Toolbars
- The Specification Tree
- Manipulating the Specification Tree
- Selecting Objects with the Mouse
- The Object/Action and Action/Objects Approaches
- Using Dialog Boxes
- Moving Objects with the Mouse
- What is the Compass?
- Graphic Properties
- Changing the Graphic Properties

#### **b) Profile Creation**

- Create a new Part
- Select and Appropriate Sketch Support
- Create Sketched Geometry
- Constrain the Sketch
- Sketch Analysis
- Manipulating Profile

### **Module 1: Part Design 3D**

- Create Pad,Pocket Feature
- Create Fillets and Chamfer
- Create Holes
- Create Multi-profile Sketch Features
- Create Feature Profiles and Axis System
- Create Shaft and Groove Features

- Create Rib And Slot Features
- Stiffner And Solid Combine
- Multi Section Solid

**c) Dress-up Features**

- Shell the Model
- Apply a Draft
- Create Threads and Taps
- Fillets And Chamfer
- Edit Features
- Mirror Features
- Pattern Features

**d) Reusing Data**

- Duplicate Features
- Transform a Body
- Copy and Paste the Data
- Insert Data from a Catalog

**e) Finalizing Design Intent**

- Apply Material Properties
- Analyze the Model
- Create Formulas and Parameters
- Create Advanced Drafts
- Annotate the Part

**f) Sharing Information**

- Create a Design Table
- Create a Catalog
- Geometrical Sets

**PRACTICE MODULE FOR PART DESIGN**

**Module 2: Assembly Design**

- Create a new CATProduct
- Assemble in the Base Component
- Manipulate the position of the Component
- Assemble and Fully Constrain Components
- Save the Assembly

**PRACTICE MODULE FOR ASSEMBLY**

### **Module 3: Drafting**

- Introduction to Generative Drafting
- Start a New Drawing
- Setting the Drawing Sheet Format and Drafting Standards
- Starting a Drawing with a Blank Sheet
- Sheet Properties
- 2D catalogs
- Create Views
- Types of Views
- About Projection Plane
- CGR Views
- Drafting Toolbars and Objects
- Title Blocks
- Create Dimensions and Annotations
- Section Views and Section Cuts
- Adding a Detail View, Clipping View, Broken View, Breakout View
- View Modifications
- Save the Drawing
- Print the Drawing
- Drafting of An Assembly
- BOM and Ballooning

### **PRACTICE MODULE FOR DRAFTING**

### **Module 4: Wireframe & Surface Design**

- a) Introduction to Surface Design**
  - Introduction to Surface Design
  - Why create 3D Wireframe Geometry?
  - Creative Points, Lines, Planes and Curves in 3D
  - Creating Wireframe Geometry
  - Surface Design Workbench Terminology
- b) Creating Surfaces**
  - Why create Surface Geometry?
  - Creating Basic Surfaces
  - Creating a Swept Surface
  - Creating a Surface Offset from a Reference
  - Creating a Surface from Boundaries
  - Creating a Multi-Section Surface
- c) Performing Operations on the Geometry**
  - Splitting/Trimming
  - Creating Fillets
  - Transforming Elements
  - Extrapolating Elements
  - Disassembling Elements

- Joining Elements
- Additional Methods for Operations

**d) Completing the Geometry in Part Design**

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

**PRACTICE MODULE FOR SURFACING**

**Module 5: Sheet metal Products**

**a) Introduction**

- Entering the Workbench
- Defining the Sheet Metal Parameters
- Primary Wall, Secondary Wall
- Creating a Cutout
- Editing the Sheet and Tool Parameters
- Modifying the Bend Extremities
- Computing the Bend Allowance
- Creating Bends on Walls
- Extruding
- Creating Rolled Walls
- Creating a Flange
- Creating a Hem
- Creating a Tear Drop
- Creating a User Flange
- Creating a Flanged Hole, Bead ,Circular Stamp , Surface Stamp, Bridge , Flanged Cutout, Stiffening Rib , Curve Stamp , Louver
- Creating User-Defined Stamping Features
- Mirroring
- Creating Corners, Creating Chamfers
- Patterning
- Looking For Sheet Metal Features
- Saving As DXF

**PRACTICE MODULE FOR SHEETMETAL**

**Kinematics' & Simulation on CATIA**

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