

CATIA V5 Complete Design Mastery

Target Audience

This program is designed for working mechanical and design engineers, product development professionals, and manufacturing engineers in automotive, aerospace, and industrial sectors who are seeking to enhance their CATIA expertise. It is ideal for professionals aiming to strengthen their skills in advanced modeling, surfacing, assemblies, and GD&T while aligning with OEM-level design standards. The program also benefits engineers transitioning from other CAD platforms and those targeting career advancement in high-level product development and engineering roles.

Course Objectives

- To establish strong foundations in CATIA V5 for advanced 3D part modeling, assembly design, and engineering drafting
 - To enable professionals to develop fully parametric, production-ready CAD models aligned with industry standards
 - To strengthen expertise in complex assemblies, surface modeling, and design validation practices
 - To enhance proficiency in GD&T application and drawing standards as per industrial requirements
 - To build end-to-end product design capabilities aligned with OEM-level workflows
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Course Outcomes

Upon successful completion of this program, participants will be able to create fully parametric 3D components, complex assemblies, and detailed production-ready engineering drawings using CATIA V5. They will gain advanced capabilities in surface modeling, assembly management, and application of GD&T in real-world design scenarios. Participants will be equipped to manage structured product design workflows with improved accuracy, efficiency, and compliance with industry standards, enabling them to contribute effectively to professional engineering environments..

Course Outline: The course comprises **40-hours** of theory and labs and is divided into **15** different chapters. Each chapter will be followed by hands-on lab exercises to reinforce learning and gauge understanding of the topics covered.

Chapter 1. Introduction to CATIA V5

- Overview of CATIA V5 Platform
 - Understanding Workbenches
 - File Types (.CATPart, .CATProduct, .CATDrawing)
 - User Interface & Customization
 - Mouse Controls and Navigation
 - Working with Specifications Tree
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Chapter 2. Drawing Sketches in the Sketcher Workbench – I

- Entering Sketcher Workbench
 - Profile Tools (Line, Circle, Arc, Rectangle)
 - Spline and Conic Tools
 - Sketch Editing Tools
 - Sketch Analysis
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Chapter 3. Drawing Sketches in the Sketcher Workbench – II

- Advanced Sketch Tools
 - Project 3D Elements
 - Axis and Construction Geometry
 - Creating Complex Profiles
 - Best Practices for Sketching
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Chapter 4. Constraining Sketches and Creating Base Features

- Geometric Constraints
 - Dimensional Constraints
 - Fully Constrained Sketch Concept
 - Creating Pad (Extrude)
 - Creating Pocket (Cut)
 - Shaft (Revolve)
 - Groove Feature
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Chapter 5. Reference Elements and Sketch-Based Features

- Creating Planes
 - Creating Axis System
 - Points and Lines
 - Rib & Slot
 - Multi-Section Solid
 - Remove Multi-Section
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Chapter 6. Creating Dress-Up and Hole Features

- Fillet
 - Chamfer
 - Draft
 - Shell
 - Hole Types (Simple, Counterbore, Countersink)
 - Thread Representation
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Chapter 7. Editing Features

- Editing Sketch
- Editing Feature Parameters
- Reordering Features
- Replace Face
- Boolean Operations
- Design Intent

Chapter 8. Transformation Features and Advanced Modeling Tools – I

- Rectangular Pattern
- Circular Pattern
- Mirror
- Scaling
- Translation & Rotation
- Thickness Feature

Chapter 9. Advanced Modeling Tools – II

- Advanced Boolean Operations
- PowerCopy Introduction
- Parameters and Formulas
- Design Table Basics
- Knowledgeware Introduction

Chapter 10. Working with the Wireframe and Surface Design Workbench

- Wireframe Elements
- Sweep
- Loft
- Multi-Section Surface

- Join, Trim, Split
 - Surface to Solid Conversion
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Chapter 11. Editing and Modifying Surfaces

- Healing Surfaces
 - Boundary & Fill
 - Extrapolate
 - Blend
 - Surface Analysis Tools
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Chapter 12. Assembly Modeling

- Creating CATProduct
 - Inserting Components
 - Assembly Constraints
 - Degrees of Freedom
 - Clash Detection
 - Exploded View
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Chapter 13. Working with the Drafting Workbench – I

- Creating Drawing from 3D Model
 - Standard Views
 - Section Views
 - Projection Methods
 - Dimensioning
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Chapter 14. Working with the Drafting Workbench – II

- Annotation Tools

- Tolerances & Surface Symbols
 - Bill of Materials (BOM)
 - Title Block Editing
 - Printing & Plot Settings
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Chapter 15. Working with Sheet Metal Components

- Sheet Metal Parameters
- Wall Creation
- Flange
- Bend & Unbend
- Corner Relief
- Flat Pattern Generation
- Generating Sheet Metal Drawings