



# CATIA V5 TRAINING – FUNDAMENTAL V5R22

## Who should attend:

Engineer and designer

## Duration:

5 days

## Methodology:

Practical hands-on by using CATIA software

## Introduction

CATIA V5 Fundamental provides you the knowledge in high-end CAD modeling including the architecture of CATIA V5 interface, 3D solid modeling, 3D surface modeling, assemblies of parts and drafting.

## Objective:

- Understand the CATIA V5 interface
- Plan the construction of a part in order to properly convey its visual and functional aspects
- Create simple parts in CATIA V5
- Understand how to manage parts in the context of an assembly
- Produce simple drawings and assembly layouts

## Course outline

### 1st day (CATIA Basics)

Introduction to CATIA  
Analyzing the Geometry

Interacting with CATIA  
Managing CATIA Data Exchange

Manipulating CATIA Window

### 2nd day (Solid Design Fundamentals)

Using of Dynamic Sketcher  
Modifying the Part

Creating the main Features  
Further Operations

Organizing the Part Structure  
Adding Material Specifications

### 3rd day (Wireframe & Surface Design)

Defining the Wireframe  
Hybrid Modeling and integration

Extracting the Curves

Modifying the Geometry

### 4th day (Assembly Design)

Defining and Assembly parts  
Detecting Collision and Clearance

manipulating the Assembly Process  
Generating the Exploded View and Bill of Material

### 5th day (Drafting)

Extracting the Draw Views  
Creating Texts & Dimensions  
Managing the Standards

Preparing the Drawings  
Plotting the Drawings

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