



*Hands on, Measurable
Training Programs*

CATIA V5 Fundamentals for Managers and Non-Designers

Course Code	EDU-CAT-en-V5FFMAND-F
Brand & Release	CATIA V5R21, V5R23 (V5-6R2013)
Duration	2 days
Level	Fundamentals
Prerequisites	None

Objectives:

Introductory course for any users of CATIA V5 that will not be responsible for creating new designs. Through a combination of instructor lectures, demonstrations, and hands-on exercises students gain an understanding of the CATIA V5 interface and how to view, measure, and manipulate existing CATIA V5 model data as well as how to capture images and create simple assemblies and drawings.

Class Structure:

Lesson 1: Introduction to CATIA

- Open CATIA
- Understand the CATIA Interface
- Recap Ex. CATIA V5 User Interface, Case Study

Lesson 2: Profile Creation

- Create a new part
- Select an appropriate sketch support
- Create sketched geometry
- Recap Ex. Profile Creation & Sketch Constraints

Lesson 3: Basic Features

- Determine a Base Feature
- Create Pad and Pocket features
- Create Holes
- Create Fillets & Chamfers
- Recap Exercises, Basic Feature Creation

Lesson 4: Additional Part Features

- Create Basic Wireframe Geometry
- Create Shaft & Groove features
- Shell the model
- Recap Ex. Shaft & Groove, Thin Pad & Shell

Lesson 5: Dress-Up Features

- Edit Features
- Recap Ex. Feature Definition, Features Activation

Lesson 7: Finalizing Design Intent

- Apply Material Properties
- Measure the model
- Recap Ex. Material and Measures

Lesson 8: Assembly Design

- Assemble the Base Component
- Manipulate the position of the component
- Assemble & fully constrain components
- Save the Assembly
- Recap Ex. Positioning & Constrain Components

Lesson 10: Drafting

- Start a new drawing
- Create views
- Create dimensions and annotations
- Create additional views
- Save & print the drawing
- Recap Ex. Drawing Creation