

2015

Basic@CATIA



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CATIA

THE DIGITAL PRODUCT EXPERIENCE



Contents

Course	Subject	Code	Duration (day)
<u>2.1.1</u>	Basic@CATIA	CAT-A00	
	Fundamentals	CAT-A01	5
	Intermediate	CAT-A02	3

Please do not hesitate to call **MAWEA** at **03-7783 3459** for further info or **FAX to 03-77816818** for registration.

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CATIA

THE DIGITAL PRODUCT EXPERIENCE

CATIA V5 TRAINING – FUNDAMENTAL (CAT A01)



Venue : MAWEA TRAINING CENTRE (Petaling Jaya, Selangor)

Date : TBC

Duration/Time : 5 Days (9.00 a.m. to 5.00 p.m.)

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

Contents

1st day (CATIA Basics)

Introduction to CATIA
Manipulating CATIA Window
Managing CATIA Data Exchange

Interacting with CATIA
Analyzing the Geometry

2nd day (Solid Design Fundamentals)

Using of Dynamic Sketcher
Organizing the Part Structure
Further Operations

Creating the main Features
Modifying the Part
Adding Material Specifications

3rd day (Wireframe & Surface Design)

Defining the Wireframe
Modifying the Geometry
Hybrid Modeling and integration

Extracting the Curves

4th day (Assembly Design)

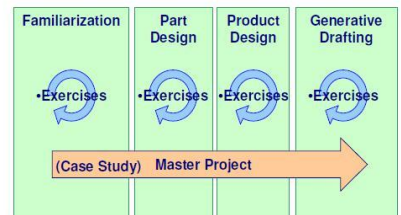
Defining and Assembly parts
Detecting Collision and Clearance
Generating the Exploded View and Bill of Material

Manipulating the Assembly Process

5th day (Drafting)

Extracting the Draw Views
Creating Texts & Dimensions
Managing the Standards

Preparing the Drawings
Plotting the Drawings



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CATIA

THE DIGITAL PRODUCT EXPERIENCE



CATIA V5 TRAINING – INTERMEDIATE (CAT-A02)



Venue : MAWEA TRAINING CENTRE (Petaling Jaya, Selangor)

Date : TBC

Duration/Time : 3 Days (9.00 a.m. to 5.00 p.m.)

Introduction

CATIA V5 Intermediate 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

Contents

1st day (Solid Expert)

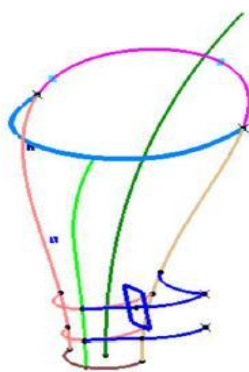
Sketch-Based Features
Dress-Up Features
Annotations

Part Manipulations
Part Analysis

2nd & 3rd day (Surface Expert)

Surface Design Overview
Wireframe Analysis and Repair
Creating Advanced Blend Surfaces
Additional Surface Design Tools

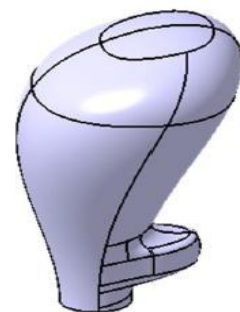
Creating Advanced Wireframe Geometry
Creating Advanced Swept Surfaces
Surface Analysis and Repair



Wireframe Geometry



Surface Geometry



Solid Geometry

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