



## CATIA V5 Training - Fundamental V5R22

### WHO SHOULD ATTEND

Engineer and designer

### DURATION

5 days

### METHODOLOGY

Practical hands-on by using  
CATIA software

Time:

9:00am to 5:00pm

Venue:

Mawea Industries Sdn Bhd

### 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.

### OBJECTIVES:

- 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
- Interacting with CATIA
- Manipulating CATIA Window
- Managing CATIA Data Exchange
- Analyzing the Geometry

#### 2nd day (Solid Design Fundamentals)

- Using of Dynamic Sketcher
- Creating the main Features
- Organizing the Part
- Structure Modifying the Part
- Further Operations
- Adding Material Specifications

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

- Defining the Wireframe
- Extracting the Curves
- Modifying the Geometry
- Hybrid Modeling and integration

#### 4th day (Assembly Design)

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

#### 5th day (Drafting)

- Extracting the Draw Views
- Preparing the Drawings
- Creating Texts & Dimensions
- Plotting the Drawings
- Managing the Standards

### CONTACT

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