CATIA - Syllabus

CATIA V5 Foundation

Course Length: 10 Days (80 hours) – Theory - **40 hrs** Practical – **40 hrs**

Session 1
Introduction to the CATIA V5 Modeling Process

- About CATIA V5
- CATIA V5 Basic Modeling Process
- Starting CATIA V5

Understand the CATIA Interface

- Windows Philosophy
- Introduction to V5 Documents
- The Workbench Concept
- CATIA User Interface
- Workbenches
- Menus and Toolbars
- Finding Tools
- The Specification Tree
- Manipulating the Specification Tree
- Selecting Objects with the Mouse
- The Object/Action and Action/Object Approaches
- Using the CATIA Dialog Boxes
- Using Dialog Boxes and Right-click
- Moving Objects with the Mouse
- Compass
- Graphic Properties

Session 1a
Sketcher
Create a New Part

- Creating a New Part
- Part Design Workbench

Select an Appropriate Sketch Support

- Reference Planes
- What is a Sketch?
- Sketch Support

Create Sketched Geometry

- Basic Sketching
- Positioned Sketching
- Sketcher Workbench
- Grid

Session 2
Create Sketched Geometry

- Geometry Creation
- Points
- Lines
- Circles
- Ellipse, Parabola, Hyperbola, and Spline
- Conics
- Pre-defined Profiles
- User-defined Profiles
- Construction Geometry

Session 2a
Constrain the Sketch

- Constraining the Sketch
- Geometric and Dimensional Constraints
- Fully-Constrained Sketches

Create the Pad Feature

- Completing the Feature
- Using a Pad to Create the First Feature

Save and Close the Document

- Saving Documents
- Saving a Document with a New Name

Closing a Document

Session 3
Basic Features in Part Design

- Determine a Suitable Base Feature
- Part Design Terminology
- Creating a Base Feature
- Selecting a Base Feature
- Features that Add or Remove Material

Create Pad and Pocket Features

- Creating Pads
- Creating a Simple Pocket
- Pad and Pocket Limits
- Restrictions for Pad/Pocket Profile Sketches
- Open Profiles

Session 3a
Create Holes
Create Fillets and Chamfers
Session 4
Create Feature Profiles and Axis system
Additional Sketcher Tools
Sketcher Relimitation Tools

- Sketcher Re-limitation Tools
- Re-limitations
- Trim Options
- Quick Trim Options

Sketcher Transformation Tools

- Mirror and Symmetry Options
- Translation
- Rotation
- Scale
- Offset Propagation Modes
- Offset

Project 3D Elements

- 3D Geometry Elements

- Isolate Projected Elements

Sketch Analysis

- Sketch Analysis Window
- Performing a Quick Geometry Diagnosis

Session 4a
Create Relationships between Dimensions

- Create Relationships between Dimensions
- Creating Equivalent Dimensions
- Creating a Formula

Create an Axis System

- Create an Axis System
- Types of Axis System

Create Multi-profile Sketch Features

- Multiple Profiles
- Multi-Pads/Pockets

Session 5
Create Shaft and Groove Features

- Creating an Axis
- Dimensioning to an Axis
- Revolved Features
- Axis of Revolution
- Shafts
- Creating Grooves
- Restrictions for Revolved Features

Shell the Model

- Shelling
- Shelling a Part
- Thin Features

Session 5a
Multi-Sections Solid

- Multi-Sections Solids
- Multi-Sections Solids: Closing Point and Orientation
- Creating a Simple Multi-Sections Solid
- Multi-Sections Solid Creation: Guides
- Multi-Sections Solid Creation : Spine
- Multi-Sections Solid Creation : Tangent Surfaces
- Multi-Sections Solid Creation : Coupling
- Multi-Sections Solid : Manual Coupling
- Multi-Sections Solid Relimitation

Session 6
Apply a draft

- Basic Drafts
- Reflect Draft
- Variable Draft
- Selecting Faces to Draft
- Using the Draft Analysis Tool
- Recommendations for Drafts
- Parting and Neutral Elements
- Create Fillets After Drafts

Create a stiffener

- Introduction to Stiffeners

Creation of Stiffeners
Create Threads and Taps
Session 6a
Edit Features

- Editing Features
- Model View Options
- Hide/Show
- Investigating the Model
- Parent-Child Relationships
- Reordering Features
- Define in Work Object
- Deactivate/Activate

- Resolving Feature Failures
- Properties

Duplicate Features

- Mirror
- Patterns
- Rectangular Patterns
- Circular Patterns
- User Patterns
- Recommendations for Patterns
- Exploding a Pattern
- Removing Individual Instances in a Pattern

Keep Specification

Session 7
Advanced Dress-Up Features

- Creating Thickness
- Ignoring Faces While Creating Thickness
- Remove Faces
- Replace Face

Transform a Body

- Using Transformations
- Translation
- Rotation
- Symmetry
- Scaling

Session 7a
Copy and Paste Data
Insert Data from a Catalog

- Power Copy
- User Feature
- Catalog Creation

- Inserting Catalog Items

Multi-Body Method

- Boolean Operation: Assemble
- Boolean Operation: Add
- Boolean Operation: Remove
- Boolean Operation: Intersect
- Boolean Operation: Union Trim
- Boolean Operation: Removing Lump

Apply Material Properties

- Material Properties
- Applying Material Properties
- Viewing Material on the Model
- Measure the Model
- Measure Between Modes
- Measure Between
- Measure Item
- Mass Properties
- Measure Inertia

Session 8
Create Formulas and Parameters

- Formulas
- Identifying Parameters
- Creating User-Defined Parameters
- Renaming Parameters
- Filters

Design Table
Session 8a
Introduction to Assembly Design
Terminology
Create a New CATProduct

- Defining a New Assembly Document
- Assigning Product Properties

Assemble the Base Component

- Inserting an Existing Component
- Assigning Component Properties
- Positioning the Compass to Move a Component
- Moving a Component Using the Compass
- Snapping Components
- Fixing a Component in Space
- Fixing a Component
- Product Structure Reordering
- Copy and Paste a Component
- Duplicating Multiple Instances of a Component
- Duplicating Components Using an Existing Pattern
- Components from a Catalog

Session 9
Assemble & Fully Constrain Components

- Degrees of Freedom
- Setting Assembly Constraints
- Assembly Constraints
- Available Constraints and their Symbols
- Defining a Coincidence Constraint
- Defining a Contact Constraint
- Defining an Offset Constraint
- Creating an Angle Constraint
- Fixing Together Components
- Updating Constraints

Save the Assembly

- Saving an Assembly Document
- Saving a Document With Another Name
- Saving All Modified Documents
- Save Management

Session 9a
Working in the Context of an Assembly
Design and Visualization Mode
Create Assembly-Level Features

- Creating an Assembly Split

- Creating an Assembly Hole
- Creating an Assembly Pocket
- Adding a Body to an Assembly
- Removing a Body from an Assembly

Session 10
Introduction to Generative Drafting

- General Process
- The Drawing Environment

Start a New Drawing

- Setting the Drawing Sheet Format and Drafting Standards
- Starting a Drawing with a Blank Sheet
- Sheet Properties
- Drawing Title Blocks 10-15
- Introduction to 2D Catalogs
- Inserting Catalog Items

Create Views

- Types of Views
- Creating a Front View
- Using the Compass
- About the Projection Plane
- Adding Projection Views
- Adding an Isometric View
- Generating views using the View Wizard

Create Additional Views

- Section Views and Section Cut
- Adding a Simple Section View on a Drawing
- Creating a Section View Using a 3D Profile Definition
- Adding a Detail View
- Creating a Clipping View
- Creating a Broken View
- Creating a Breakout View
- Adding an Auxiliary View

View Modifications

- Repositioning Views
- Deleting Views
- View Properties
- Modifying the Links of a View
- Replacing the Projection Plane of a View
- Invert Section View Profile
- Replace Section View Profile

Session 10a
Create Dimensions and Annotations

- Dimensions System
- Creating Dimensions
- Dimensioning a Length
- Dimensioning a Distance
- Dimensioning a Hole
- Dimensioning a True Length
- Dimensioning a Simple Angle
- Dimensioning a Simple Radius
- Dimensioning a Diameter
- Dimensioning a Chamfer
- Dimensioning a Thread
- Chain Dimensions
- Stacked Dimensions
- Cumulated Dimensions
- Annotations