OBJECTIVE OF THE COURSE

CATIA (**computer-aided three-dimensional interactive application**) is a multi-platform software suite for computer-aided design (CAD), computer-aided manufacturing (CAM), computer-aided engineering (CAE), 3D modeling and product lifecycle management (PLM), developed by the French company Dassault Systèmes.

DURATION OF THE COURSE : 60 HOURS (4 WEEKS).
MINIMUM ELIGIBILITY CRITERIA AND PRE-REQUISITE : BASIC KNOWLEDEGE OF Drafting.

Module 1 (Week 1 – 16 hours):

Introduction to the CATIA V5 Modeling Process

- CATIA V5 Basic Modeling Process
- Starting CATIA V5

Understand the CATIA Interface

- Windows Philosophy
- The Workbench Concept
- CATIA User Interface
- Menus and Toolbars
- Moving Objects with the Mouse
- Compass
- Graphic Properties

Sketcher

Create a New Part

- Creating a New Part
- Part Design Workbench

Select an Appropriate Sketch Support

- Reference Planes
- What is a Sketch?

Create Sketched Geometry

  - Basic Sketching
  - Sketcher Workbench
  - Grid

Create Sketched Geometry

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

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

Module 2 (Week 2 - 14 hours):

Basic Features in Part Design

- Part Design Terminology

Create Pad and Pocket Features

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

Create Holes

Create Fillets and Chamfers

Create Feature Profiles and Axis system

Sketcher Relimitation Tools

- Trim Options
- Quick Trim Options

Sketcher Transformation Tools

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

Module 3 (Week 3 – 16 hours):

Create Multi-profile Sketch Features

- Multiple Profiles
- Multi-Pads/Pockets

Create Shaft and Groove Features

- Creating an Axis
- Dimensioning to an Axis
- Revolved Features

- Axis of Revolution
- Shafts
- Creating Grooves

Shell the Model

- Shelling
- Shelling a Part
- Thin Features

Multi-Sections Solid

- Multi-Sections Solids
- Multi-Sections Solids: Closing Point and Orientation
- Multi-Sections Solid Creation : Spine
- Multi-Sections Solid Creation : Tangent Surfaces

Apply a draft

- Basic Drafts
- Reflect Draft
- Variable Draft
- Selecting Faces to Draft
- Create Fillets After Drafts

Create a stiffener

- Introduction to Stiffeners

Create Threads

Duplicate Features

- Mirror
- Patterns
- Rectangular Patterns
- Circular Patterns

Apply Material Properties

  - Material Properties
  - Applying Material Properties
  - Viewing Material on the Model

Module 4 (Week 4 – 14 hours):

Introduction to Assembly Design

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**
- **Snapping Components**
- **Fixing a Component**

Assemble & Fully Constrain Components

- Assembly Constraints

- Available Constraints and their Symbols
- Defining a Coincidence Constraint
- Defining a Contact Constraint
- Defining an Offset Constraint
- Creating an Angle Constraint

Save the Assembly

- Saving an Assembly Document
- Saving a Document With Another Name

Introduction to Generative Drafting

- General Process
- The Drawing Environment