IFS Academy

Training For The Future!!

CATIA V5 Mechanical Designer

Course Curriculum (Duration: 80 Hrs.)

Chapter 1: CATIA V5 Fundamentals

1.1] Introduction to CATIA
   a. Design Intent
   b. Understand CATIA Software
   c. PLM-Product Life Cycle Management
   d. CATIA within the PLM Solution
   e. CATIA Coverage
   f. What is CATIA V5?
   g. Open CATIA
   h. Starting CATIA using the Start Menu
   i. Opening an existing Document
   j. Understand the CATIA Interface
   k. The Windows Philosophy
   l. Introduction to V5 Documents
   m. The Workbench Concept
   n. CATIA User Interface
   o. Workbenches
   p. Menus and Toolbars
   q. The Specification Tree
   r. Manipulating the Specification Tree
   s. Selecting Objects with the Mouse
   t. The Object/Action and Action/Objects Approaches
   u. Using Dialog Boxes
   v. Moving Objects with the Mouse
   w. What is the Compass?
   x. Graphic Properties
   y. Changing the Graphic Properties
   z. Rendering Styles
   aa. Hide/Show
   bb. Help Documentation
   cc. Message Bar

1.2] Profile Creation
   a. Create a new Part
   b. Select and Appropriate Sketch Support
   c. Create Sketched Geometry
   d. Constrain the Sketch
   e. Create Pad Feature
   f. Save and Close the Document

1.3] Basic Features
   a. Determine a suitable Base Feature
   b. Create Pad and Pocket Features

c. Create Holes

d. Create Fillets and Chamfers

1.4] Additional Part Features

    a. Create Feature Profiles and Axis System

    b. Create Multi-profile Sketch Features

    c. Create Wireframe Geometry

    d. Create Shaft and Groove Features

    e. Shell the Model

1.5] Dress-up Features

    a. Apply a Draft

    b. Create a Stiffener

    c. Create Threads and Taps

    d. Edit Features

1.6] Reusing Data

    a. Duplicate Features

    b. Transform a Body

    c. Copy and Paste the Data

    d. Insert Data from a Catalog

1.7] Finalizing Design Intent

    a. Apply Material Properties

    b. Analyze the Model

    c. Create Formulas and Parameters

1.8] Assembly Design

    a. Create a new CATProduct

    b. Assemble in the Base Component

    c. Manipulate the position of the Component

    d. Assemble and Fully Constrain Components

    e. Save the Assembly

1.9] Designing in Context

    a. Open an existing assembly

    b. Insert a new model

    c. Create a Sketch in Context

    d. Create Assembly-Level Features

1.10] Drafting:

    a. Introduction to Generative Drafting

    b. General Process

    c. Accessing the Workbench

    d. The Drawing Environment

    e. Drafting Toolbars and Objects

    f. Start a New Drawing

    g. Setting the Drawing Sheet Format and Drafting Standards

    h. Starting a Drawing with a Blank Sheet

    i. Sheet Properties

    j. Title Blocks

    k. 2D catalogs

    l. Inserting Catalog Items

    m. Create Views

2.5] Assembly Design
- a. Manage the product structure
- b. Create flexible sub-assemblies
- c. Analyze the assembly
- d. Create scenes
- e. Create annotations
- f. Generate reports

2.6] Contextual Design
- a. Clarify the display
- b. Create Contextual Parts
- c. Create Assembly-Level Features
- d. Manipulate the Contextual Components
- e. Save the Contextual Models

2.7] Complex Assembly Design
- a. Create a Skeleton Model
- b. Create the Published Elements
- c. Use the Published Elements

2.8] Exercises and Workshops

Chapter 3: CATIA Surface Design (Generative Shape Design)

3.1] Introduction to Surface Design
- a. Introduction to Surface Design
- b. The Generative Shape Design Workbench
- c. Surface Design Workbench User Interface
- d. Surface Design Workbench Terminology
- e. Surface Design Workbench General Process

3.2] Creating Wireframe Geometry
- a. Why create 3D Wireframe Geometry?
- b. Creative Points, Lines, Planes and Curves in 3D
- c. Creating Wireframe Geometry
- d. To Sum Up

3.3] Shape Design Common Tools
- a. Why use common Tools?
- b. Stacking Commands
- c. Managing Geometrical Sets
- d. To Sum Up

3.4] Creating Surfaces
- a. Why create Surface Geometry?
- b. Creating Basic Surfaces
- c. Creating a Swept Surface
- d. Creating a Surface Offset from a Reference
- e. Creating a Surface from Boundaries
- f. Creating a Multi-Section Surface

3.5] Performing Operations on the Geometry
- a. Why are operations on Geometry needed?
- b. Joining Elements

c. Splitting/Trimming
d. Creating Fillets
e. Transforming Elements
f. Extrapolating Elements
g. Disassembling Elements
h. Additional Methods for Operations

3.6] Completing the Geometry in Part Design
a. Why complete geometry in part design?
b. Creating a Solid from surfaces?
c. Completing Geometry recommendations

3.7] Modifying the Geometry
a. What about modifying the Geometry?
b. Editing Surface and Wireframe Definition
c. Modifying Geometry Recommendations

3.8] Using Tools
a. What about using Tools?
b. Creating Datum Features
c. Checking Connections between Elements
d. Updating a part

3.9] Exercise and Workshops

Chapter 4: CATIA Generative Sheetmetal Design

4.1] Getting Started
a. Introduction to Sheetmetal Design
b. Generative Sheetmetal Design Overview
c. Defining Sheetmetal Parameters
d. Recommendations for Getting Started

4.2] Sheetmetal Walls
a. What is Sheetmetal Wall?
b. Defining the First Wall
c. Secondary Walls
d. Part Recognition
e. Recommendation for Sheetmetal Walls
f. Summary

4.3] Bends and Unfolded Mode
a. Bends
b. Unfloded Mode
c. Corner Relief
d. Recommendation for Bends
e. Summary

4.4] Flanges
a. Creating Flanges
b. Recommendation for Flanges

c. Summary

4.5] Sheetmetal Features
   a. Cutouts
   b. Holes and Circular Cutouts
   c. Corners and Chamfers
   d. Standard Stamps
   e. User-Defined Stamps
   f. Recommendation for Sheet Metal Features
   g. Summary

4.6] Transformations and Duplication
   a. Transformation of Parts
   b. Patterns
   c. Mirror
   d. PowerCopies
   e. Recommendation for Duplication
   f. Summary

4.7] Mapping & Output
   a. Point & Curve Mapping
   b. Output
   c. Summary

4.8] Advanced Topics in Sheet Metal Design
   a. Sheet Metal Standard Files
   b. Multi-Body Methodology
   c. Summary

4.9] Exercise and Workshops

Case Study & Tool Test: Student has to complete any one live project case study and submit report of the same followed by Tool Test.

Deliverables: *Soft copy of Training Manual, Dassault Systemes Certificate of Training, Dassault Systemes CATIA Certification Exam and Online access to Companion Learning Space for DS Study Material.*