

## **CATIA Syllabus**

### **1: INTRODUCTION TO CATIA**

- Introduction to CATIA
- CATIA Workbenches
- System Requirements
- Getting Started with CATIA
- Important Terms and Definitions
- Understanding the Functions of the Mouse Buttons
- Toolbars
- Hot Keys
- Color Scheme

### **2: DRAWING SKETCHES IN THE SKETCHER WORKBENCH-I**

The Sketcher Workbench

Starting a New File

Invoking the Sketcher Workbench

- Invoking the Sketcher Workbench Using the Sketch Tool

- Invoking the Sketcher Workbench Using the Positioned Sketch Tool

Setting the Sketcher Workbench

- Modifying Units

- Modifying the Grid Settings

Understanding Sketcher Terms

- Specification Tree

- Grid

- Snap to Point

- Construction/Standard Element

- Select Toolbar

- Inferencing Lines

Drawing Sketches Using Sketcher Tools

- Drawing Lines

- Drawing Center Lines

- Drawing Rectangles, Oriented Rectangles, and Parallelograms

Drawing Rectangles  
Creating Points  
Drawing Circles  
Drawing Arcs  
Drawing Profiles  
Drawing Display Tools

Fit All In  
Pan  
Zoom In  
Zoom Out  
Zoom Area  
Normal View  
Splitting the Drawing Area into Multiple Viewports  
Hiding and Showing Geometric Elements  
Swapping Visible Space

### **3: DRAWING SKETCHES IN THE SKETCHER WORKBENCH-II**

Other Sketching Tools in the Sketcher Workbench

Drawing Ellipses  
Drawing Splines  
Connecting Two Elements by a Spline or an Arc  
Drawing Elongated Holes  
Drawing Cylindrical Elongated Holes  
Drawing Keyhole Profiles  
Drawing Hexagons  
Drawing Centered Rectangles  
Drawing Centered Parallelograms  
Drawing Conics

Editing and Modifying Sketches

Trimming Unwanted Sketched Elements  
Extending Sketched Elements  
Trimming by Using the Quick Trim Tool

- Filleting Sketched Elements
- Chamfering Sketched Elements
- Mirroring Sketched Elements
- Mirroring Elements without Duplication
- Translating Sketched Elements
- Rotating Sketched Elements
- Scaling Sketched Elements
- Offsetting Sketched Elements
- Modifying Sketched Elements
- Deleting Sketched Elements

#### **4: CONSTRAINING SKETCHES AND CREATING BASE FEATURES**

Constraining Sketches

Concept of Constrained Sketches

- Iso-Constraint

- Under-Constraint

- Over-Constrained

- Inconsistent

- Not Changed

Applying Geometrical Constraints

- Applying Geometrical Constraints Automatically

- Applying Additional Constraints to the Sketch

- Applying Dimensional Constraints

- Applying Contact Constraints

- Applying Fix Together Constraints

- Applying Auto Constraints

- Editing Multiple Dimensions

Analyzing and Deleting Over-Defined Constraints

- Analyzing Sketch using the Sketch Analysis Tool

Exiting the Sketcher Workbench

Creating Base Features by Extrusion

- Creating a Thin Extruded Feature



Extruding the Sketch Using the Profile Definition Dialog Box

Extruding the Sketch along a Directional Reference

Creating Base Features by Revolving Sketches

Creating Thin Shaft Features

Dynamically Rotating the View of the Model

Rotating the View Using the Rotate Tool

Rotating the View Using the Compass

Modifying the View Orientation

Display Modes of the Model

Shading (SHD)

Shading with Edges

Shading with Edges without Smooth Edges

Shading with Edges and Hidden Edges

Shading with Material

Wireframe (NHR)

Customize View Parameters

Creating Sections Dynamically

Maneuvering the Section Plane

Position of Section Planes

Assigning a Material to the Model

**5: REFERENCE ELEMENTS AND SKETCH-BASED FEATURES**

Importance of Sketching Planes

Reference Elements

Reference Planes

Creating New Planes

Creating Points

Creating Reference Lines

Other Sketch-Based Features

Creating Drafted Filleted Pad Features

Creating Multi-Pad Features

Feature Termination Options

Creating Pocket Features

Creating Drafted Filleted Pocket Features

Creating Multi-Pocket Features

Creating Groove Features

Extruding and Revolving Planar and Non-planar Faces

Projecting 3D Elements

## **6: CREATING DRESS-UP AND HOLE FEATURES**

Advanced Modeling Tools

Creating Hole Features

Creating Fillets

Creating Chamfers

Adding a Draft to the Faces of the Model

## **7: EDITING FEATURES**

Editing Features of a Model

Editing Using the Definition Option

Editing by Double-Clicking

Editing the Sketch of a Sketch-Based Feature

Redefining the Sketch Plane of Sketches

Deleting Unwanted Features

Managing Features and Sketches by using the Cut, Copy,  
And Paste Functionalities

Understanding the Concept of Update Diagnosis

Cut, Copy, and Paste Features and Sketches

Copying Features Using Drag and Drop

Copying and Pasting Part Bodies

Deactivating Features

Activating Deactivated Features

Defining Features in Work Object

Reordering Features

Understanding the Parent-Child Relationships

Measuring Elements

Measuring between Elements

Measuring Items

Measuring Inertia

## **8: TRANSFORMATION FEATURES AND ADVANCED MODELING TOOLS-I**

Transformation Features

Translating Bodies

Rotating Bodies

Creating Symmetry Features

Transforming the Axis System

Mirroring Features and Bodies

Creating Rectangular Patterns

Creating Circular Patterns

Creating User Patterns

Uniform Scaling of Model

Non-uniform Scaling of Model

Working with Additional Bodies

Inserting a New Body

Inserting Features in the New Body

Applying Boolean Operations to Bodies

Adding Stiffeners to a Model

Generating Solid Combine

## **9: ADVANCED MODELING TOOLS-II**

Advanced Modeling Tools

Creating Rib Features

Creating Slot Features

Creating Multi-Sections Solid Features

## **10: WORKING WITH THE WIREFRAME AND SURFACE DESIGN WORKBENCH**

Need of Surface Modeling

Wireframe and Surface Design Workbench

Starting the Wireframe and Surface Design Workbench

Creating Wireframe Elements

Creating Circles

Creating Splines

Creating a Helix

#### Creating Surfaces

Creating Extruded Surfaces

Creating Revolved Surfaces

Creating Spherical Surfaces

Creating Cylindrical Surfaces

Creating Offset Surfaces

Creating Sweep Surfaces

Creating Fill Surfaces

Creating Multi-Sections Surfaces

Creating Blended Surfaces

#### Operations on Shape Geometry

Joining Surfaces

Splitting Surfaces

Trimming Surfaces

### 11: EDITING AND MODIFYING SURFACES

#### Surface Operations

Creating Projection Curves

Creating Intersection Elements

Healing Geometries

Disassembling Elements

Untrimming a Surface or a Curve

Creating Boundary Curves

Extracting Geometry

Transformation Features

Extrapolating Surfaces and Curves

Splitting a Solid Body with a Surface

#### Solidifying Surface Models

Adding Thickness to a Surface

Creating a Solid Body from a Closed Surface Body

Sewing a Surface to a Solid Body

## **12: ASSEMBLY MODELING**

Assembly Modeling

Types of Assembly Design Approaches

Creating Bottom-up Assemblies

Inserting Components in a Product File

Moving Individual Components

Applying Constraints

Creating Top-down Assemblies

Creating Base Part in the Top-Down Assembly

Creating Subsequent Components in the Top-Down Assembly

Creating Subassemblies in the Top-Down Assembly

Editing Assemblies

Deleting Components

Replacing Components

Editing Components inside an Assembly

Editing Subassemblies inside an Assembly

Editing Assembly Constraints

Simplifying the Assembly

Interference Detection

Sectioning an Assembly

Exploding an Assembly

## **13: WORKING WITH THE DRAFTING WORKBENCH-I**

The Drafting Workbench

Starting a New File in the Drafting Workbench

Type of Views

Generating Drawing Views

Generating Views Automatically

Generating Individual Drawing Views

Generating the Exploded View



Working with Interactive Drafting in CATIA

Editing and Modifying Drawing Views

- Changing the Scale of Drawing Views

- Modifying the Project Plane of the Parent View

- Deleting Drawing Views

- Rotating Drawing Views

- Hiding Drawing Views

Modifying the Hatch Pattern of Section Views

## **14: WORKING WITH THE DRAFTING WORKBENCH-II**

Inserting Sheets in the Current File

Inserting the Frame and the Title Block

- Automatic Insertion of the Frame and the Title Block

- Creating the Frame and the Title Block Manually

Adding Annotations to the Drawing Views

- Generating Dimensions

- Adding Reference Dimensions

- Adding Datum Features

- Adding Geometric Tolerance to the Drawing Views

- Adding Surface Finish Symbols

- Adding Welding Symbols

- Applying Weld

Editing Annotations

Generating the Bill of Material (BOM)

Generating Balloons

## **15: WORKING WITH SHEET METAL COMPONENTS**

The Sheet metal Component

- Starting a New File in Generative Sheet Metal Workbench

Setting Sheet Metal Parameters

- Parameters Tab

- Bend Extremities Tab

- Bend Allowance Tab

## Introduction to Sheet Metal Walls

- Creating the Base Wall

- Creating the Wall on Edge

## Creating Extrusions

## Creating Swept Walls

- Creating Flanges on the Sheet Metal Component

- Creating Hems on the Sheet Metal Component

- Creating a Tear Drop on the Sheet Metal Component

- Creating a User Flange on the Sheet Metal Component

## Creating a Bend

- Creating a Conical Bend

## Bend From Flat

## Creating Rolled Walls

- Creating a Hopper Wall

- Creating a Rolled Wall

## Folding and Unfolding Sheet Metal Parts

- Unfolding Sheet Metal Parts

- Folding Unfolded Parts

- Mapping the Geometry

## Creating Flat Patterns of Sheet Metal Components

## Viewing a Sheet Metal Component in Multiple Windows

## Using Views Management

## Stamping

- Creating a Surface Stamp

- Creating a Bead Stamp

- Creating a Curve Stamp

- Creating a Flanged Cut out Stamp

- Creating a Louver Stamp

- Creating a Bridge Stamp

- Creating a Flanged Hole Stamp

- Creating a Circular Stamp

Creating a Stiffening Rib Stamp

Creating a Dowel Stamp

## **16: DMU KINEMATICS (Optional)**

Introduction to DMU Kinematics

Designing a Mechanism

Creating the Revolute Joint

Creating the Prismatic Joint

Creating the Cylindrical Joint

Creating the Screw Joint

Creating the Rigid Joint

Creating the Spherical Joint

Creating the Planar Joint

Creating the Point Curve Joint

Creating the Slide Curve Joint

Creating the Roll Curve Joint

Creating the Point Surface Joint

Creating the Universal Joint

Creating the CV Joint

Creating the Gear Joint

Creating the Rack Joint

Creating the Cable Joint

Converting Assembly Constraints into Joints