CATIA

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

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

- 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
- DimensionsAnalyzing 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 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