



QUICKSTEP COMPUTER CENTER

National Accreditation Board of Education Training.
(NABET)- Quality council of India) An ISO 9001:2008

***** CATIA-V5*****

Module 1 .

- ❖ Introduction to CATIA V5
- ❖ CATIA V5 Workbenches
- ❖ System Requirements
- ❖ Getting Started with CATIA V5
- ❖ Important Terms and Definitions
- ❖ Understanding the Functions of Mouse Buttons
- ❖ Toolbars
- ❖ Hot Keys
- ❖ Color Scheme

Module 2 .Drawing Sketches in the Sketcher Workbench-

- ❖ 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 Grid Settings
- ❖ Understanding Sketcher Terms
- ❖ Specification Tree
- ❖ Grid
- ❖ Snap to Point
- ❖ Construction/Standard Element
- ❖ Select Toolbar
- ❖ Inference Lines
- ❖ Drawing Sketches Using Sketcher Tools
- ❖ Drawing Display Tools

Module 3 .Drawing Sketches in the Sketcher Workbench-II

- ❖ Other Sketching Tools in the Sketcher Workbench
- ❖ Editing and Modifying Sketches
- ❖ Constraining Sketches and Creating Base Features
- ❖ Constraining Sketches
- ❖ Concept of Constrained Sketches
- ❖ Applying Geometrical Constraints
- ❖ Analyzing and Deleting Over-Defined Constraints
- ❖ Exiting the Sketcher Workbench
- ❖ Creating Base Features by Extrusion
- ❖ Creating Base Features By Revolving Sketches
- ❖ Dynamically Rotating the View of the Model
- ❖ Modifying the View Orientation
- ❖ Display Modes of the Model
- ❖ Creating Sections Dynamically
- ❖ Assigning a Material to the Model

Module 4 .Reference Elements and Sketch-Based Features

- ❖ Importance of Sketching Planes
- ❖ Reference Elements
- ❖ Other Sketch-Based Features

Module 5 .Creating Dress-Up and Hole Features

- ❖ Advanced Modeling Tools

• Module 6 .Editing Features

- ❖ Editing Features of a Model
- ❖ Measuring Elements

Module 7 .Transformation Features and Advanced Modeling Tools-I

- ❖ Transformation Features
- ❖ Working With Additional Bodies
- ❖ Adding Stiffeners to a Model
- ❖ Generating Solid Combine

Module 8 .Advanced Modeling Tools-II

- ❖ Advanced Modeling Tools

Module 9 .Editing and Modifying Surfaces

- ❖ FTP From
- ❖ FTP To

Module 10 .Compress Components with Surfacing

- ❖ The Surface modeling
- ❖ Sketching surface's
- ❖ Introduction to Surface methods
- ❖ Creating surfacing
- ❖ Creating complex surface
- ❖ Creating a surface blends to sheet and solid
- ❖ Surface Operations
- ❖ Solidifying Surface Models

Module 11 .Assembly Modeling

- ❖ Assembly Modeling
- ❖ Creating Bottom-up Assemblies
- ❖ Creating Top-down Assemblies
- ❖ Editing Assemblies

Module 12 .Working with the Drafting Workbench-I

- ❖ The Drafting Workbench
- ❖ Type of Views
- ❖ Generating Drawing Views
- ❖ Generating the Exploded View
- ❖ Working with Interactive Drafting in CATIA V5
- ❖ Editing and Modifying Drawing Views
- ❖ Modifying the Hatch Pattern of Section Views

Module 13 .Working with the Drafting Workbench-II

- ❖ Inserting Sheets in the Current File
- ❖ Inserting the Frame and the Title Block
- ❖ Adding Annotations to the Drawing Views
- ❖ Editing Annotations
- ❖ Generating the Bill of Material (BOM)
- ❖ Generating Balloons

Module 14 .Working with Sheet Metal Components

- ❖ The Sheet metal Component
- ❖ Setting Sheet Metal Parameters
- ❖ Introduction to Sheet Metal Walls
- ❖ Creating Extrusions
- ❖ Creating Swept Walls
- ❖ Creating a Bend
- ❖ Bend From Flat
- ❖ Folding and Unfolding Sheet Metal Parts
- ❖ Creating Flat Patterns of Sheet Metal Components
- ❖ Viewing a Sheet Metal Component in Multiple Windows
- ❖ Using Views Management
- ❖ Stamping
Deployment