CADD Master 5C
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PTC creo™

creo

Training Details

Course Code	: HCCC10
Course Name	: creo
Duration	: 2 Months
Eligibility	: Diploma / BE / ME / EIE / EEE / ECE /+2 / Equivalent

Syllabus:

1: INTRODUCTION TO CREO PARAMETRIC

- Introduction to Creo Parametric
- Feature-Based Nature
- Bidirectional Associative Property
- Parametric Nature
- System Requirements
- Getting Started with Creo Parametric
- Important Terms and Definitions
- File Menu Options
- Managing Files
- Menu Manager
- Model Tree
- Understanding the Functions of the Mouse Buttons
- Ribbon
- Toolbars
- Navigator
- Creo Parametric Browser
- Appearance Gallery
- Rendering in Creo Parametric
- Colour Scheme Used in this Book

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2: CREATING SKETCHES IN THE SKETCH MODE-I

- The Sketch Mode
- Working with the Sketch Mode
- Invoking the Sketch Mode
- The Sketcher Environment
- Working with a Sketch in the Sketch Mode
- Drawing a Sketch Using tools available in the Sketch Tab
- Placing a Point
- Drawing a Line
- Drawing a Centerline
- Drawing a Geometry Centerline
- Drawing a Rectangle
- Drawing a Circle
- Drawing an Ellipse
- Drawing an Arc
- Dimensioning the Sketch
- Converting a Weak Dimension into a Strong Dimension
- Dimensioning a Sketch Using the Normal Tool
- Dimensioning the Basic Sketched Entities
- Linear Dimensioning of a Line
- Angular Dimensioning of an Arc
- Diameter Dimensioning
- Radial Dimensioning
- Dimensioning Revolved Sections
- Working with Constraints
- Types of Constraints
- Disabling Constraints
- Modifying the Dimensions of a Sketch
- Using the Modify Button
- Modifying a Dimension by Double-Clicking on it
- Modifying Dimensions Dynamically
- Resolve Sketch Dialog Box
- Deleting the Sketched Entities
- Trimming the Sketched Entities
- Mirroring the Sketched Entities
- Inserting Standard/User-Defined Sketches
- Drawing Display Options

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3: CREATING SKETCHES IN THE SKETCH MODE-II

- Dimensioning the Sketch
- Dimensioning a Sketch Using the Baseline Tool
- Replacing the Dimensions of a Sketch Using the Replace Tool
- Creating Fillets
- Creating Circular Fillets
- Creating Elliptical Fillets
- Creating a Reference Coordinate System
- Working with Splines
- Creating a Spline
- Dimensioning of Splines
- Modifying a Spline
- Writing Text in the Sketcher Environment
- Rotating and Resizing Entities
- Importing 2D Drawings in the Sketch Mode
- 4: CREATING BASE FEATURES
- Creating Base Features
- Invoking the Part Mode
- The Default Datum Planes
- Creating a Protrusion
- Extruding a Sketch
- Revolving a Sketch
- Understanding the Orientation of Datum Planes
- Parent-Child Relationship
- Implicit Relationship
- Explicit Relationship
- Nesting of Sketches

5: DATUMS

- Datums
- Default Datum Planes
- Need for Datums in Modeling
- Selection Method in Creo Parametric
- Datum Options, Datum Planes
- Creating Datum Planes
- Datum Planes Created On-The-Fly
- Datum Axes, Datum Points

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- Creating Cuts, Removing Material by Using the Extrude Tool
- Removing Material by Using the Revolve Tool
- 6: OPTIONS AIDING CONSTRUCTION OF PARTS-I
- Options Aiding Construction of Parts
- Creating Holes, The Hole Dashboard
- Important Points to Remember While Creating a Hole
- Creating Rounds, Creating Basic Rounds
- Creating a Variable Radius Round
- Points to Remember While Creating Rounds
- Creating Chamfers, Corner Chamfer
- Edge Chamfer, Understanding Ribs
- Creating Trajectory Ribs
- Creating Profile Ribs
- Editing Features of a Model
- Editing Definition or Redefining Features
- Reordering Features
- Rerouting Features, Suppressing Features
- Deleting Features, Modifying Features

7: OPTIONS AIDING CONSTRUCTION OF PARTS-II

- Introduction, Creating Feature Patterns
- Uses of patterns, Creating Patterns
- Deleting a Pattern, Copying Features
- New Refs, Same Refs, Mirror, Move
- Select, Mirroring a Geometry
- Creating a Section of a Solid Model
- Work Region Method

8: ADVANCED MODELING TOOLS-I

- Other Protrusion Options
- Sweep Features
- Creating Sweep Protrusions
- Aligning a Sketched Trajectory to an Existing Geometry
- Creating a Thin Sweep Protrusion
- Creating a Sweep Cut, Blend Features
- Parallel Blend, Rotational Blend
- General Blend, Using Blend Vertex

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- Shell Feature, Creating a Constant Thickness Shell
- Creating a Variable Thickness Shell 8
- Datum Curves
- Creating a Datum Curve by Using the Curve Button
- Creating a Datum Curve by Sketching
- Creating a Curve by Using the Intersect Option
- Creating a Curve by Using the Project Option
- Creating a Curve by Using the Wrap Option
- Creating Draft Features

9: ADVANCED MODELING TOOLS-II

- Advanced Feature Creation Tools
- Variable Section Sweep Using the Sweep Option
- Swept Blend, Helical Sweep
- Blend Section to Surfaces
- Blend Between Surfaces

10: ADVANCED MODELING TOOLS-III

- Advanced Modeling Tools, Toroidal Bend
- Spinal Bend, Warp
- Transform Tool, Warp Tool
- Spine Tool, Stretch Tool
- Bend Tool, Twist Tool, Sculpt Tool

11: ASSEMBLY MODELING

- Assembly Modeling, Important Terms Related to the Assembly Mode
- Top-down Approach, Bottom-up Approach
- Placement Constraints, Package
- Creating Top-down Assemblies
- Creating Components in the Assembly Mode
- Creating Bottom-up Assemblies
- Inserting Components in an Assembly
- Assembling Components
- Displaying Components in a Separate Window
- Displaying Components in the Same Window
- 3D Dragger, Applying Constraints
- Status Area, Placement Tab

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- Move Tab, Packaging Components
- Creating Simplified Representations
- Redefining the Components of an Assembly
- Reordering Components
- Suppressing/Resuming Components
- Replacing, Assembling Repeated Copies of a Component
- Modifying the Components of an Assembly
- Modifying Dimensions of a Feature of a Component
- Redefining a Feature of a Component
- Creating the Exploded State
- References Tab Offset Tab, Explode Line Tab
- The Bill of Materials, Global Interference
- Pairs Clearance

12: GENERATING, EDITING, AND MODIFYING THE DRAWING VIEWS

- The Drawing Mode
- Generating Drawing Views
- Generating the General View
- Generating the Projection View
- Generating the Detailed View
- Generating the Auxiliary View
- Generating the Revolved Section View
- Generating the Copy and Align View
- Generating the 3D Cross-Section View
- Editing the Drawing Views
- Moving the Drawing View
- Erasing the Drawing View
- Deleting the Drawing View
- Adding New Parts or Assemblies to the Current Drawing
- Modifying the Drawing Views
- Changing the View Type
- Changing the View Scale
- Reorienting the Views
- Modifying the Cross-sections
- Modifying Boundaries of Views
- Adding or Removing the Cross-section Arrows
- Modifying the Perspective Views

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- Modifying Other Parameters
- Editing the Cross-section Hatching

13: DIMENSIONING THE DRAWING VIEWS

- Dimensioning the Drawing Views
- Show Model Annotations Dialog Box
- Adding Notes to the Drawing
- Adding Tolerances in the Drawing Views
- Dimensional Tolerances
- Geometric Tolerances
- Editing the Geometric Tolerances
- Adding Balloons to the Assembly Views
- Adding Reference Datums to the Drawing Views
- Modifying and Editing Dimensions
- Modifying the Dimensions Using the Dimension Properties Dialog Box
- Modifying the Drawing Items Using the Shortcut Menu
- Cleaning Up the Dimensions

14: OTHER DRAWING OPTIONS

- Sketching in the Drawing Mode
- Modifying the Sketched Entities
- User-Defined Drawing Formats
- Retrieving the User-Defined Formats in the Drawings
- Adding and Removing Sheets in the Drawing
- Creating Tables in the Drawing Mode
- Generating the BOM and Balloons in Drawings

15: SURFACE MODELING

- Surface Modeling
- Creating Surfaces in Creo Parametric
- Creating an Extruded Surface
- Creating a Revolved Surface
- Creating a Sweep Surface
- Creating a Blended Surface
- Creating a Swept Blend Surface
- Creating a Helical Sweep Surface
- Creating a Surface by Blending the Boundaries

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- Creating a Variable Section Sweep Surface Using the Sweep Tool
- Creating Surfaces the Using the Style Environment of Creo Parametric
- Style Dashboard, Surface Editing Tools
- Mirroring the Surfaces, Merging the Surfaces
- Trimming the Surfaces, Creating the Fill Surfaces
- Creating the Intersect Curves
- Creating the Offset Surfaces
- Adding Thickness to a Surface
- Converting a Surface into a Solid
- Creating a Round at the Vertex of a Surface
- Freestyle modelling environment
- Freestyle Dashboard

16: WORKING WITH SHEET METAL COMPONENTS

- Introduction to Sheet metal
- Invoking the Sheet metal Mode
- Introduction to Sheet metal Walls
- Creating the Planar Wall
- Creating the Unattached Revolve Wall
- Creating the Unattached Blend Wall
- Creating the Unattached Offset Wall
- Creating Reliefs in Sheet metal Components
- Creating a Flat Wall, Creating a Twist Wall
- Creating an Extend Wall
- Creating a Flange Wall
- Creating the Bend Feature
- Creating the Unbend Feature
- Creating the Bend Back
- Conversion to Sheet metal Part
- Creating Cuts in the Sheet metal Components

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