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* 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
* **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
* **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
* Placing a Point
* Drawing a Line
* Drawing a Centreline
* Drawing a Geometry Centreline
* 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 Too
* 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 SSStandard/User-Defined Sketches
* Drawing Display Options

**TECHNOLOGY : CREO**

**DURATION: MODULE 1 (4 WEEKS)**

**MODULE 1 + 2 (6 WEEKS)**

* **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
* 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
* 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
* 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
* Creating Cuts
* Removing Material by Using the Extrude Tool
* Removing Material by Using the Revolve Tool
* **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
* **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
* **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
* 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
* **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
* 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

**MODULE 2(6 WEEKS)**

* **GENERATING, EDITING, AND MODIFYING THE DRAWING VIEWS**
* The Drawing Mode
* Generating the General View
* Generating the Projection View
* Generating the Detailed View
* Generating the 3D Cross-Section View
* Generating Drawing Views
* Editing the Drawing Views
* 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
* Modifying Other Parameters
* Editing the Cross-section Hatching
* **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
* **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
* **SURFACE MODELING**
* Surface Modeling
* Creating Surfaces in Creo Parametric
* Creating an Extruded Surface
* Creating a Revolved Surface
* Creating a Sweep Surface
* 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
* **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

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