

# SLOG SOLUTIONS PRIVATE LIMITED 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

- ✓ 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 SSSstandard/User-Defined Sketches
- Drawing Display Options

## ❖ 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

- **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**

- ✓ **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**

## 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

- ✓ 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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