Creo Parametric   Duration-80 hours

Session	Topics

Introduction to the Creo Parametric Basic Modeling Process

- Creo Parametric Basic Modeling Process

Understanding Creo Parametric Concepts

- Understanding Solid Modeling Concepts
- Understanding Feature-Based Concepts
- Understanding Parametric Concepts
- Understanding Associative Concepts
- Understanding Model-Centric Concepts
- Recognizing File Extensions

Using the Creo Parametric Interface

- Understanding the Main Interface
- Understanding the Folder Browser
- Understanding the Web Browser
- Understanding the Ribbon Interface
- Setting the Working Directory and Opening and Saving Files
- Managing Files in Creo Parametric

Session 1

Creating Sketcher Geometry

- Reviewing Sketcher Theory
- Understanding Design Intent
- Modifying the Sketcher Display
- Utilizing Constraints
- Sketching with on the fly constraints
- Sketching Lines
- Sketching Centerlines
- Sketching Rectangles
- Sketching Circles
- Sketching Arcs
- Sketching Circular Fillets and Chamfers

Using Sketcher Tools

- Understanding Construction Geometry Theory

- Sketching Points
- Using Geometry Tools within Sketcher
- Manipulating Sketches within Sketcher
- Dimensioning Entities within Sketcher
- Modifying Dimensions within Sketcher
- Sketcher Conflicts
- Placing Sections into Sketcher

Creating Sketches for Features

- Creating Sketches ('Sketch' Feature)
- Specifying the Sketch Setup
- Utilizing Sketch References
- Using Entity from Edge within Sketcher

Advanced Sketching

- Sketching Ellipses
- Sketching Elliptical Fillets
- Sketching Splines
- Modifying Splines (Basic & Advanced)
- Importing and Exporting Spline Points
- Sketching Conics
- Sketching Text
- Analyzing Sketcher Convert Options
- Analyzing Sketcher Dimension Options
- Sketcher Diagnostic Tools

Creating Extrudes, Revolves, and Ribs

- Creating Solid Extrude Features
- Adding Taper to Extrude Feature
- Common Dashboard Options: Extrude Depth
- Common Dashboard Options: Feature Direction
- Common Dashboard Options: Thicken Sketch
- Creating Solid Revolve Features
- Common Dashboard Options: Revolve Angle
- Automatically Adding and Removing Material
- Creating Rib Features

Session 2

Selecting and Editing

- Understanding Creo Parametric Basic Controls
- Using Drag Handles and Dimension Draggers
- Using Keyboard Shortcuts
- Understanding the Model Tree
- Selecting Items using Direct Selection
- Selecting Items using Query Selection
- Selecting Multiple Components
- Using the Search Tool
- Understanding Selection Filters
- Renaming Objects
- Editing Features and Regenerating
- Activating and Editing Models
- Deleting and Suppressing Items

Creating Datum Features: Planes and Axes

- Creating Datum Features Theory
- Creating Datum Axes
- Creating Datum Planes

Utilizing Internal Sketches and Embedded Datums

- Creating Internal Sketches
- Creating Embedded Datum Features

Creating Sweeps

- Creating Sweeps with Open Trajectories
- Creating Sweeps with Closed Trajectories
- Analyzing Sweep Feature Attributes

Creating Blends

- Creating a Parallel Blend Protrusion or Cut
- Creating a Rotational Blend Protrusion or Cut

Session 3   ## Creating Holes and Shells

- Common Dashboard Options: Hole Depth
- Creating Coaxial Holes
- Creating Linear Holes
- Creating Radial and Diameter Holes

- Exploring Hole Profile Options
- Creating Shell Features

Advanced hole Creation

- Creating Standard Holes
- Creating Sketched Holes
- Creating On Point Holes

Advanced Shells

- Analyzing Shell References and Thickness Options
- Excluding Surfaces from Shells
- Extending Shell Surfaces
- Analyzing Shell Corner Options

Creating Rounds and Chamfers

- Creating Rounds by Selecting Edges
- Creating Rounds by Selecting a Surface and Edge
- Creating Rounds by Selecting Two Surfaces
- Creating Full Rounds
- Creating Round Sets
- Creating Chamfers by Selecting Edges
- Creating Chamfer Sets

Advanced Rounds and Chamfers

Session 4
- Creating Rounds Through Curve
- Creating Variable Radius Rounds
- Auto Round
- Creating Corner Chamfers
- Creating Rounds and Chamfers by Reference
- Using Intent Edges for Rounds and Chamfers
- Using Chamfer Transitions
- Using Round Transitions

Drafts

- Creating Draft Features
- Creating Basic Split Drafts
- Analyzing Draft Hinges and Pull Direction

- Drafting Intent Surfaces
- Creating Drafts with Multiple Angles
- Using the Extend Intersect Surfaces Draft Option
- Creating Drafts Split at Sketch
- Creating Drafts Split at Curve
- Creating Drafts Split at Surface
- Creating Drafts with Variable Pull Direction

Variable Section Sweeps

- Creating Variable Section Sweeps using a Constant Section
- Creating Variable Section Sweeps Normal to Trajectory
- Creating Variable Section Sweeps using Constant Normal Direction
- Creating Variable Section Sweeps Normal to Projection

Helical Sweeps

- Understanding Helical Sweeps Theory
- Creating Helical Sweeps for Springs
- Creating Helical Sweeps for Threads

Swept Blends

- Understanding Swept Blend Theory
- Creating Swept Blends by Selecting Sections
- Creating Swept Blends by Sketching Sections

Creating Patterns

Session 5

- Direction Patterning in the First Direction
- Direction Patterning in the Second Direction
- Axis Patterning in the First Direction
- Axis Patterning in the Second Direction
- Creating Reference Patterns of Features
- Creating Reference Patterns of Components
- Deleting Patterns or Pattern Members

Advanced Patterns

- Creating Dimension Patterns in One Direction
- Creating Dimension Patterns in Two Directions

- Creating Rotational Dimension Patterns
- Understanding Pattern Regeneration Options
- Creating Fill Patterns
- Specifying Fill Pattern Settings
- Creating Curve Patterns
- Creating Point Pattern

Group, Copy, and Mirror Tools

- Creating Local Groups
- Copying and Pasting Features
- Moving and Rotating Copied Features
- Mirroring Selected Features
- Mirroring All Features
- Creating Mirrored Parts

Assembling with Constraints

- Understanding Assembly Theory
- Creating New Assembly Models
- Understanding Constraint Theory
- Understanding Assembly Constraint Status
- Assembling Components using the Default Constraint
- Constraining Components using Automatic
- Constraining Components using Distance
- Constraining Components using Angle Offset
- Constraining Components using Parallel
Session 6
- Constraining Components using Coincident
- Constraining Components using Normal
- Constraining Components using Coplanar
- Constraining Components using Centred
- Constraining Components using Tangent
- Constraining Components using Fix

Exploding Assemblies

- Creating and Managing Explode States
- Creating Offset Lines between Exploded Components

Creating Drawing Views
Session 7

- Analyzing Drawing Concepts and Theory

- Analyzing Basic 2-D Orientation
- Creating New Drawings and Applying Formats
- Creating and Orienting General Views
- Adding Drawing Models and Sheets
- Creating Projection Views

Creating Drawing Views

- Creating Cross-Section Views
- Creating Auxiliary Views
- Creating 2D Cross Section Views
- Creating New Drawings using Drawing Templates
- Modifying Drawing Views
- Creating Assembly and Exploded Views

Creating Drawing Details

- Analyzing Detail Concepts and Types
- Showing and Erasing Detail Items
- Cleaning Up Dimensions
- Manipulating Dimensions
- Creating Driven Dimensions

Session 8

Creating Drawing Details

- Creating Notes
- Showing a Bill of Materials
- Analyzing Drawing Associativity