Creo for Mechanical Engineers

Duration: 50 Days

Syllabus Content

Module 1: Introduction to Creo Parametric Concept

- Creo Parametric Basic Modeling Process
- Understanding Parametric Concepts
- Recognizing File Extensions
- Understanding the Folder Browser
- Setting the Working Directory and Opening and Saving Files
- Setting Up New Part Models
- Creating Sketcher Geometry
- Utilizing Constraints
- Using Geometry Tools within Sketcher
- Modifying Dimensions within Sketcher

Module 2: Solid Modeling in Creo

- Creating Solid Extrude Features
- Adding Taper to Extrude Features
- Common Dashboard Options: Extrude Depth
- Creating Solid Revolve Features
- Common Dashboard Options: Revolve Angle
- Creating Holes
- Creating Shell Feature
- Creating Draft Feature
- Creating Basic Split Drafts
- Pattern
- Creating Rounds Theory (By Edge, Surface and Edge, by two Surface)
- Creating Full Rounds
- Creating Chamfers

Module 3: Advanced Selection, Creating Sweeps and Blends

- Creating Sweeps with Open Trajectories
- Creating Sweeps with Closed Trajectories
- Creating Blends by Sketching Sections
- Creating Rotational Blends by Selecting Sections
- Understanding Sweeps with Variable Sections Theory
- Creating Sweeps Normal to Trajectory
- Creating Sweeps using Constant Normal Direction
- Creating Sweeps with Variable Sections Normal to Projection
- Creating Sweeps with Variable Sections Utilizing Multiple Trajectories
- Understanding Helical Sweeps Theory

- Creating Helical Sweeps for Springs
- Understanding Swept Blend Theory
- Creating Swept Blends by Selecting Sections
- Creating Swept Blends by Sketching Sections

Module 4: Relations, Parameters & Family Tables

- Understanding Relation Theory
- Creating Relations
- Creating Parameters
- Creating a Family Table
- Copying and Pasting Features

Module 5: Measuring, Inspecting Models

- Viewing and Editing Model Properties
- Investigating Model Units
- Analyzing Mass Properties
- Creating Planar Part Cross-Sections

Module 6: Introduction to Flexible Modeling

- Flexible Modeling Process
- Using the Selection Filter
- Applying Shape Selection
- Applying Flexible Move using Dragger
- Using Flexible Mirror
- Using the Edit Round Feature
- Working with Pattern Recognition

PRACTICE MODULE FOR PART DESIGN

Module 7: Introduction to Assembly & Restructuring

- Understanding Component Interfaces
- Using a Placing Component Interface
- Using a Receiving Component Interface
- Placing Flexible Components in an Assembly
- Creating Assembly Structure
- Constraint the Assembly
- Reorder the Parts in Assembly
- Editing the Constraint

PRACTICE MODULE FOR ASSEMBLY

Module 8: Surfacing Modeling in Creo

- Introduction to Surfacing

- Creating Surface Extrude Features
- Creating Surface Revolve Features
- Creating Fill Surfaces
- Understanding Boundary Curve Concepts
- Creating Boundary Blends in One Direction
- Creating Boundary Blends in Two Directions
- Analyzing Surfaces Theory
- Extending Surfaces
- Creating a Surface Trim
- Trimming Surfaces with Quilts Options
- Copying and Pasting Surfaces
- Offsetting Surfaces
- Moving and Rotating Quilts
- Mirroring Quilts
- Merging Surfaces
- Thickening Surface Quilts
- Solidifying Quilts to Add Material

PRACTICE MODULE FOR SURFACING

Module 9: Drafting in Creo

- Understanding Drawing Concepts
- Creating Drawings Using Formats and Sheets
- Adding General Views
- Adding Projection Views
- Editing Drawing Views
- Editing Visible View Area
- Adding 2-D Cross-Section Views
- Adding Assembly Exploded Views
- Understanding Annotations in Drawings
- Showing, Erasing, and Deleting Annotations
- Adjusting Dimensions and Detail Items
- Understanding Dimensional Tolerances
- Configuring Dimensional Tolerances
- Understanding Geometric Tolerances
- Setting Up Geometric Tolerance References
- Applying Geometric Tolerances
- Adding and Editing Notes
- Using Surface Finish Symbols
- Creating BOM Balloons

Module 10: Sheetmetal Design in Creo

- Sheetmetal Model Fundamentals
- Understanding Developed Length
- Creating a New Sheetmetal Model in Part Mode
- Creating Planar Walls
- Extruded Sheetmetal Wall Features
- Revolved Sheetmetal Wall Features

- Understanding Secondary Walls
- Creating Secondary Flat Walls
- Using Flange Walls
- Using Extruded Walls
- Understanding Relief
- Creating Bend Features
- Adding Transition to Bends
- Creating Unbend Features
- Creating Bend Back Features
- Creating Flat States
- Sheetmetal Cuts
- Die Form Features
- Punch Form Features
- Creating Rip Features

PRACTICE MODULE FOR SHEETMETAL

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