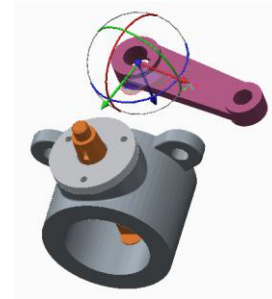


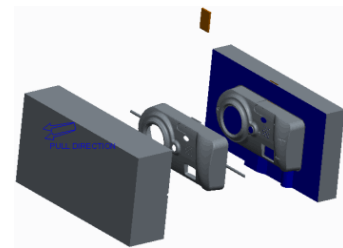
Creo for Production Engineers

Overview

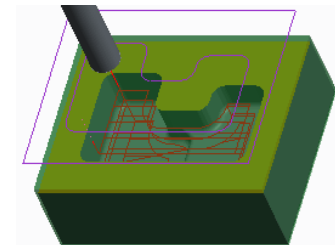
In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will become familiar with the Sketcher workflow and reference enhancements. Part modeling enhancements to features such as Extrude, Corner Chamfer, Sweeps, Blends, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities, such as selecting multiple components and enhancements for dragging components.



The Mold application provides the tools to create a mold model from start to finish by using the mold design process within Creo Parametric. In this course, you will learn how to create, modify, and analyze mold components and assemblies. Any changes made to the design model automatically propagate to the mold components and assemblies. You will learn how to create final extract components that reflect the geometry of the design model, along with shrinkage considerations, adequate drafting, mold features, and cooling systems. After completing the course, you will have a better understanding of the mold design process and how to create molded products by using the mold design process.



In this, you will learn how to machine products using Creo Parametric manufacturing tools. This course covers creating tool paths for three axis milling machines. During the course, you will learn how to complete each phase of the manufacturing process. You will start by creating manufacturing models and configuring the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will then learn how to create milling sequences, holmaking sequences, and post-process cutter location (CL) data to create machine code. After completing the course, you will be able to create numerical control (NC) programs for milling machines and post-process cutter location (CL) data to create machine specific code.



Course Objectives

- Introduction & Understanding to Creo Parametric Concepts
- Using Creo Parametric Interface
- Selecting & Editing of Geometry, Features, Models
- Creating Sketcher Geometry & Using Sketcher Tools
- Using Sketches & Datum Features
- Creating Extrudes & Revolves
- Creating Holes, Shells, Draft & Patterns
- Creating Rounds, Chamfers & Using Layers
- Assembling with Constraints
- Exploding, Replacing Components, Cross-Sections in Assemblies
- Introduction & Design Model Preparation
- Mold Model, Shrinkage & Workpieces
- Mold Volume, Parting Surface
- Splitting Mold Volumes & Mold Component Extraction
- Mold Features Creation & Opening the Mold
- Introduction to Manufacturing

- Using NC Model Assemblies, Workcells & Parameters
- Creating Face Milling & Profile Milling Sequences
- Creating Volume Milling Sequences
- Process Manager & Post-Processing CL Data

Prerequisites

- None

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers
- People in related roles can also benefit from taking this course

Duration

- 80 Hrs (10 Days)
 - 40 Hrs for ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0
 - 40 Hrs for ATC's Mold Design & Manufacturing - Creo 2.0

Agenda

ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0

1. Introduction & Understanding to Creo Parametric Concepts

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

2. Using Creo Parametric Interface

- Understanding the Main Interface
- Understanding the Folder Browser
- Setting the Working Directory and Opening and Saving Files
- Understanding the Ribbon Interface
- Managing Files in Creo Parametric
- Understanding Datum Display Options
- Analyzing Basic 3-D Orientation
- Understanding the View Manager
- Setting Up New Part Models

3. Selecting & Editing of Geometry, Features, Models

- Understanding Creo Parametric Basic Controls

- Using Drag Handles and Dimension Draggers
- Understanding the Model Tree
- Selecting Items using Direct Selection
- Selecting Items using Query Selection
- Using the Smart Selection Filter
- Utilizing Undo and Redo Operations
- Understanding Regeneration and Auto Regeneration
- Editing Features
- Editing Features using Edit Definition
- Deleting and Suppressing Items

4. Creating Sketcher Geometry & Using Sketcher Tools

- Reviewing Sketcher Theory
- Understanding Design Intent
- Utilizing Constraints
- Sketching Lines
- Sketching Rectangles and Parallelograms
- Sketching Circles
- Sketching Arcs
- Understanding Construction Geometry Theory
- Using Geometry Tools within Sketcher
- Dimensioning Entities within Sketcher
- Modifying Dimensions within Sketcher

5. Using Sketches & Datum Features

- Creating Sketches ('Sketch' Feature)
- Specifying and Manipulating the Sketch Setup
- Utilizing Sketch References
- Using Entity from Edge within Sketcher
- Creating Datum Features Theory
- Creating Datum Axes
- Creating Datum Planes

6. Creating Extrudes & Revolves

- Creating Solid Extrude Features
- Adding Taper to Extrude Features
- Common Dashboard Options: Extrude Depth
- Creating Solid Revolve Features
- Common Dashboard Options: Revolve Angle

7. Creating Holes, Shells, Draft & Patterns

- Common Dashboard Options: Hole Depth
- Creating Coaxial Holes

- Creating Linear Holes
- Creating Radial and Diameter Holes
- Creating Shell Features
- Creating Draft Features
- Creating Basic Split Drafts
- Direction Patterning in the First Direction
- Axis Patterning in the First Direction
- Creating Reference Patterns of Features

8. Creating Rounds, Chamfers & Using Layers

- Creating Rounds Theory
- Creating Rounds by Selecting Edges
- Creating Rounds by Selecting a Surface and Edge
- Creating Rounds by Selecting Two Surfaces
- Creating Full Rounds
- Creating Chamfers by Selecting Edges
- Analyzing Basic Chamfer Dimensioning Schemes
- Understanding Layers
- Utilizing Layers in Part Models
- Creating and Managing Layers

9. Assembling with Constraints

- Understanding Assembly Theory
- Creating New Assembly Models
- Understanding Constraint Theory
- Assembling Components using the Default Constraint
- Creating Coincident Constraints using Geometry
- Creating Coincident Constraints using Datum Features
- Creating Distance Constraints
- Creating Parallel, Normal, and Angle Constraints
- Assembling using Automatic

10. Exploding, Replacing Components, Cross-Sections in Assemblies

- Creating and Managing Explode States
- Animating Explode States
- Understanding Component Replace
- Replacing Components using Family Table
- Understanding Assembly Cross-Sections
- Creating Assembly Cross-Sections
- Creating Offset Assembly Cross-Sections
- Creating Display Styles

ATC's Mold Design & Manufacturing - Creo 2.0

11. Introduction & Design Model Preparation

- Creo Parametric Basic Mold Process
- Understanding Mold Theory
- Preparing Design Models for the Mold Process
- Analyzing Design Models Theory
- Performing a Draft Check

12. Mold Model, Shrinkage & Workpieces

- Creating New Mold Models
- Creating the Reference Model
- Analyzing Reference Model Orientation
- Understanding Shrinkage
- Applying Shrinkage by Scale
- Creating a Workpiece Automatically
- Creating and Assembling a Workpiece Manually

13. Mold Volume, Parting Surface

- Understanding Mold Volumes
- Sketching Slider Mold Volumes
- Understanding Parting Lines and Parting Surfaces
- Creating an Automatic Parting Line using Silhouette Curves
- Analyzing Silhouette Curve Options: Slides
- Creating a Skirt Surface
- Creating a Parting Surface Manually

14. Splitting Mold Volumes & Mold Component Extraction

- Splitting the Workpiece
- Splitting Mold Volumes
- Splitting Volumes using Multiple Parting Surfaces
- Extracting Mold Components from Volumes
- Applying Start Models to Mold Components

15. Mold Features Creation & Opening the Mold

- Creating Sprues and Runners
- Creating Ejector Pin Clearance Holes
- Creating a Molding
- Opening the Mold
- Interference Checking a Mold Opening Step

16. Introduction to Manufacturing

- Manufacturing Process Overview
- Creating Manufacturing Models
- Configuring Operations

- Using Reference Models
- Using Workpiece Models

17. Using NC Model Assemblies, Workcells & Parameters

- Creating and Using NC Model Assemblies
- Creating and Configuring Workcells
- Understanding Manufacturing Parameter Concepts
- Configuring Parameter Values

18. Creating Face Milling & Profile Milling Sequences

- Basic Face Milling
- Lateral Control Face Milling Parameters
- Depth Control Face Milling Parameters
- Entry and Exit Face Milling Parameters
- Basic Profile Milling
- Depth and Lateral Control Profile Milling Parameters
- Lead In and Lead Out Motions

19. Creating Volume Milling Sequences

- Basic Volume Milling
- Volume Milling with Mill Windows
- Scanning Volume Milling Parameters
- Depth and Lateral Control Volume Milling Parameters
- Stock Allowance Volume Milling Parameters
- Gathering Mill Volumes
- Modifying Volume Milling Toolpaths

20. Process Manager & Post-Processing CL Data

- Using Process Manager Tools
- Editing Process Items
- Creating and Post-Processing CL Data Files