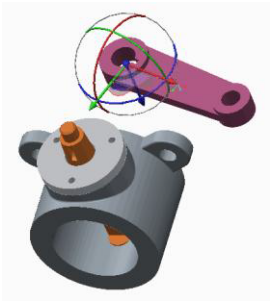
Creo for Industrial Designers

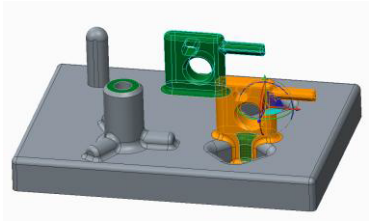
Overview

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.

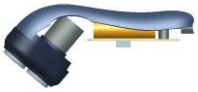


The Advanced Modeling using Creo Parametric 2.0 teaches you how to use advanced part modeling techniques to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse existing design geometry when creating new design models. After completing this course, you will be well prepared to work efficiently with complex product designs using Creo Parametric 2.0.

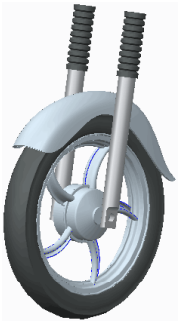
In this, you will learn how to use flexible modeling tools to edit existing geometry on parametric models. The flexible modeling process typically involves initially selecting model surfaces, then refining the selected surface set using smart selection tools, and finally modifying the selected geometry by applying transformation tools, patterning tools, or symmetry tools. Each stage of the process is described in detail and supported by step-by-step exercises.



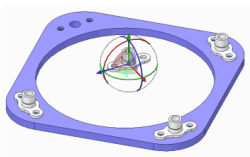
In this, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how manipulate surfaces using editing tools, and analyze surfaces for quality and desired characteristics. In addition, you will learn how to create solid features using the surfaces as references. After completing this course, you will be well prepared to create complex shaped models using surfaces in Creo Parametric.



In Creo Parametric, you can create freeform surface models using the Style and Freestyle modeling environments. Collectively, the use of these environments is often called Freeform surfacing. The Style modeling environment is a spline-based freeform modeler that enables you to combine the parametric feature-based modeling approach with the unconstrained freeform surface modeling approach. This gives you the flexibility to design complex-shaped products in a single modeling environment. The Freestyle modeling environment provides commands to create smooth and well defined B-spline surfaces quickly and easily using a polygonal control mesh. You will learn how to integrate style features with other parametric features in design models.



In this, you will learn direct modeling using PTC's new Creo Direct product. You will learn Creo Direct's interface and approach to direct modeling, including how to quickly create sketches with precision, transform sketches into 3-D shapes and directly manipulate existing geometry with ease.



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
- Advanced Selection, Creating Sweeps and Blends
- Sweeps with Variable Sections
- Helical Sweeps & Swept Blends
- Relations, Parameters & Family Tables
- Groups, Copy, Mirror & UDF's
- Measuring, Inspecting Models & Seeking Help
- Capturing, Managing Design Intent & Resolving Failures
- Introduction to Flexible Modeling
- Editing, Transformations & Recognition in Flexible Modeling
- Surface Modeling Overview
- Advanced Datum Features
- Basic & Boundary Blend Surfaces
- Surface Analysis Tools
- Extending, Trimming & Manipulating Surfaces
- Creating and Editing Solids using Quilts
- Introduction & Understanding Freeform Surface Modeling Concepts
- Creating Curves & Developing Surfaces in Freeform Surface Modeling
- Introduction to Creo Direct & Creating Sketches in 2-D Mode
- Creating Features & Assemblies in Creo Direct
- Selecting, Modifying, and Reusing 3-D Geometry in Creo Direct

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

- 120 Hrs (15 Days)
    - *40 Hrs for ATC's Part (Basic) & Assembly Modeling (Basic) - Creo 2.0*
    - *36 Hrs for ATC's Part (Adv) & Flexible Modeling - Creo 2.0*
    - *44 Hrs for ATC's Surface Design & Direct Modeling - 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 Part (Adv) & Flexible Modeling - Creo 2.0

11. Advanced Selection, Creating Sweeps and Blends

- Advanced Chain Selection
- Advanced Surface Selection
- Creating Sweeps with Open Trajectories
- Creating Sweeps with Closed Trajectories
- Creating Blends by Sketching Sections
- Creating Rotational Blends by Selecting Sections

12. Sweeps with Variable 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

13. Helical Sweeps & Swept Blends

- 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

14. Relations, Parameters & Family Tables

- Understanding Relation Theory
- Understanding Relation Types
- Creating Parameters
- Creating Relations
- Understanding Family Table Theory
- Creating a Family Table

15. Groups, Copy, Mirror & UDF's

- Creating Local Groups
- Copying and Pasting Features
- Moving and Rotating Copied Features
- Mirroring Selected Features
- Creating UDF's
- Placing UDF's

16. Measuring, Inspecting Models & Seeking Help

- Viewing and Editing Model Properties
- Investigating Model Units
- Analyzing Mass Properties
- Creating Planar Part Cross-Sections
- Using Creo Parametric Help

17. Capturing, Managing Design Intent & Resolving Failures

- Handling Children of Deleted and Suppressed Items
- Reordering Features
- Inserting Features
- Redefining Features and Sketches
- Understanding and Identifying Failures
- Analyzing Geometry Failures
- Analyzing Open Section Failures
- Analyzing Missing Part Reference Failures

18. Introduction to Flexible Modeling

- Understanding Flexible Modeling
- Flexible Modeling Process
- Using the Selection Filter
- Applying Shape Selection

- Applying Boss Selections

19. Editing, Transformations & Recognition in Flexible Modeling

- Applying Flexible Move using Dragger
- Using Flexible Mirror
- Using the Edit Round Feature
- Working with Pattern Recognition
- Using the Flexible Attach Feature

ATC's Surface Design & Direct Modeling - Creo 2.0

20. Surface Modeling Overview

- Introduction to Surfacing
- Surface Modeling Uses
- Surface Modeling Paradigms
- Blending Surface Modeling Paradigms
- Surfacing Terms

21. Advanced Datum Features

- Creating Datum Coordinate Systems
- Creating Points On or Offset from Entities
- Creating Points using an Offset Coordinate System
- Creating Curves Through a Point or Vertex
- Creating a Curve Through a Point Array
- Creating Composite Curves
- Projecting and Wrapping Curves
- Trimming Curves

22. Basic & Boundary Blend Surfaces

- 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

23. Surface Analysis Tools

- Analyzing Surfaces Theory
- Defining Curvature
- Defining Curvature Continuity
- Analyzing Curvature of Curves
- Analyzing Curvature of Surfaces
- Using Shaded Curvature Analysis for Surfaces

- Using the Dihedral Angle Analysis Option
- Using the Draft Analysis Option
- Using the Reflection Analysis Option

24. Extending, Trimming & Manipulating Surfaces

- Extending Surfaces
- Creating a Surface Trim
- Trimming Surfaces with Geometry
- Trimming Surfaces with Quilts Options
- Copying and Pasting Surfaces
- Offsetting Surfaces
- Moving and Rotating Quilts
- Mirroring Quilts
- Merging Surfaces

25. Creating and Editing Solids using Quilts

- Thickening Surface Quilts
- Solidifying Quilts to Add Material
- Solidifying Quilts to Remove Material

26. Introduction & Understanding Freeform Surface Modeling Concepts

- Introduction to Freeform Surface Modeling
- Typical Pro/ENGINEER Freeform Modeling Process
- Understanding Freeform Surface Modeling Concepts
- Understanding Style Features
- Understanding the Style Modeling Environment
- Using Style Tool Shortcut Menus
- Using Style Tool Key Combinations
- Understanding Active Planes

27. Creating Curves & Developing Surfaces in Freeform Surface Modeling

- Creating Initial Freeform Curves
- Understanding Style Curves
- Creating Basic Style Curves
- Defining Endpoint Tangency
- Editing Curves
- Analyzing Curves
- Developing Freeform Surface Models
- Understanding Style Surfaces
- Creating Boundary Surfaces
- Creating Loft Surfaces
- Creating N-Sided Surfaces

28. Introduction to Creo Direct & Creating Sketches in 2-D Mode

- Understanding the User Interface
- Orienting and Positioning the Model
- Understanding 2-D Mode
- Using the Line-Arc Chain Tool
- Sketching Rectangles, Circles, and Arcs
- Understanding Direct Modeling
- Trimming Sketched Entities

29. Creating Features & Assemblies in Creo Direct

- Understanding Sketches and Regions
- Creating Extrusions
- Revolving Sketches
- Creating Holes
- Creating Rounds and Chamfers
- Assembling Components

30. Selecting, Modifying, and Reusing 3-D Geometry in Creo Direct

- Understanding Selection Filters
- Using Shape Selection
- Understanding the CoPilot
- Using the Move/Rotate Operation
- Editing Rounds
- Removing Surfaces From a Solid