

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.

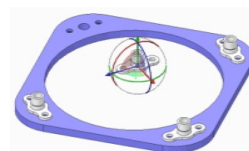
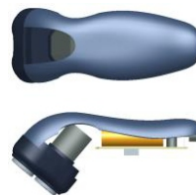
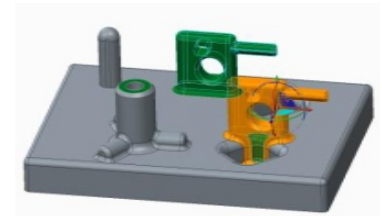
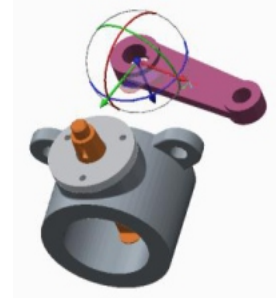
The Advanced Modeling using Creo Parametric 2.0 teaches you how to use advanced part modeling techniques to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse 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 to 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 Dragers
- 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