



Update to Creo Parametric 4.0 from Creo Parametric 3.0

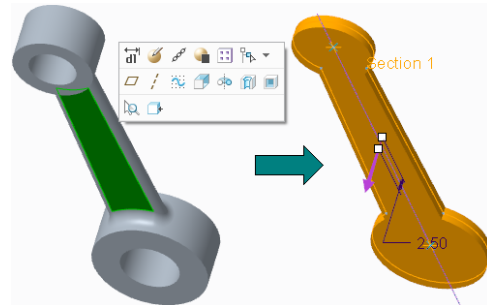
Overview

Course Code TRN-5100-T

Course Length 8 Hours

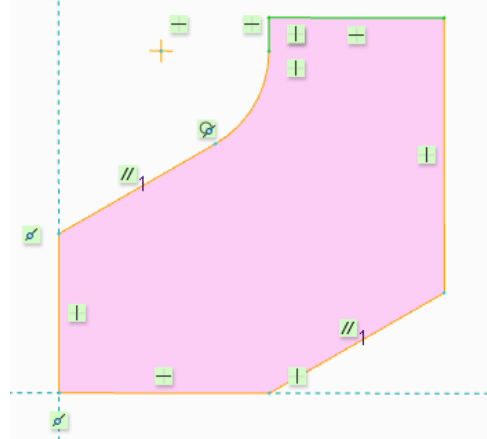
In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric 4.0. You will be introduced to user interface enhancements such as the mini toolbar and the new geometry selection filter. You will examine the Part Modeling enhancements to features such as negative depth, extrude features with offset depth, and new midplane datum plane type. You will learn about the new Sketcher enhancements, including the clipping the model, customizing line thickness, the new constraint appearance, and using sketch references. You will investigate the new Assembly capabilities such as automatic representations, mirror symmetry, and assigning materials. You will examine the new Datum Feature Symbol, Datum Target, Geometric Tolerance, and Driven Dimension functionality and workflows for 2-D drawings and review various detailing enhancements. You will learn the new Freestyle surfacing enhancements including using multiple objects, importing OBJ files, and using N-Gon faces, as well as new Style enhancements such as G3 connections, creating periodic and nonperiodic closed curves, and drop curves. Finally, you will learn to use the updated tools in Sheetmetal mode such as Twist Walls, Edge Bend Relief, and Close Corner edge treatment, as well as the enhancement for utilizing dependency control with punch and die forms. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0



Course Objectives

- Use the Interface enhancements in Creo Parametric 4.0
- Use the Part Modeling enhancements in Creo Parametric 4.0
- Use the Sketcher enhancements in Creo Parametric 4.0
- Use the Assembly Modeling enhancements in Creo Parametric 4.0
- Use the Drawing enhancements in Creo Parametric 4.0
- Use the Surfacing enhancements in Creo Parametric 4.0
- Use the Sheetmetal enhancements in Creo Parametric 4.0



Prerequisites

- Introduction to Creo Parametric 3.0, or equivalent experience with Creo Parametric 3.0

Audience

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

Day 1

Module	1	Interface Enhancements
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Module	2	Part Modeling Enhancements
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Module	3	Sketcher Enhancements
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Module	4	Assembly Enhancements
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Module	5	Drawing Enhancements
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Module	6	Surfacing Enhancements
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Module	7	Sheetmetal Enhancements
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Course Content

Module 1. Interface Enhancements

- i. Interface Appearance Enhancements
- ii. Understanding Creo Parametric Basic Controls
- iii. Mini Toolbar Enhancement
- iv. Using the Geometry Selection Filter
- v. Customizing the User Interface
- vi. Toggle Full Screen Enhancement
- vii. Column Visibility Toggle Enhancement
- viii. Hidden Items Display Enhancement
- ix. Double-byte Characters Enhancement
- x. Locked Config.sup Options Enhancement
- xi. General Feature Location Enhancements
- xii. Using Real-Time Rendering
- xiii. Model View Dialog Box Enhancement
- xiv. Line Style Display Enhancement
- xv. Creating Appearance States
- xvi. Understanding and Identifying Failures
- xvii. Understanding the Notification Center
- xviii. Analyzing Geometry Failures

Knowledge Check Questions

Module 2. Part Modeling Enhancements

- i. Negative Depth Direction Enhancement
- ii. Creating Extrude Features with Offset Depth
- iii. Creating On Point Holes
- iv. Using the Top Clearance Option
- v. Datum Midplane Enhancement
- vi. Using the Exclude Areas with Draft Option
- vii. Group Enhancements
- viii. Accuracy Handling when Scaling or Changing Units

Knowledge Check Questions

Module 3. Sketcher Enhancements

- i. Clip Model Enhancement
- ii. Customizable Line Thickness Enhancement
- iii. Box Selection Enhancement
- iv. Sketching with On-the-Fly Constraints
- v. Sketching Lines
- vi. Sketching Text
- vii. Utilizing Sketch References

Knowledge Check Questions

Module 4. Assembly Enhancements

- i. Regeneration Status Enhancement
- ii. Using Automatic Representations
- iii. Creating Mirrored Assemblies
- iv. Creating Mirrored Components
- v. Creating Mirrored Sub-Assemblies
- vi. Assigning Materials
- vii. Creating Flexible Components with Varied Material
- viii. Outdated Mass Properties Enhancement
- ix. Managing Reference Backups
- x. Mechanism Motors as Features Enhancement
- xi. Initial Conditions from Playback of Other Analysis Enhancement
- xii. Creating Local Copy Geometry Features in Parts

*Knowledge Check Questions***Module 5. Drawing Enhancements**

- i. MBD Enhancements
- ii. Annotation Workflow Enhancement
- iii. Understanding Semantic References
- iv. Understanding Syntax Checking
- v. Creating Datum Feature Symbols
- vi. Creating Datum Targets
- vii. Applying Geometric Tolerances
- viii. Creating Driven Dimensions
- ix. Embedding Images in Drawings
- x. Replacing View Models
- xi. Non-Linear Cross-Hatching Enhancement
- xii. Text and Symbol Fonts Enhancement

*Knowledge Check Questions***Module 6. Surfacing Enhancements**

- i. Using Multiple Objects
- ii. Importing and Exporting OBJ Files into Freestyle
- iii. Using N-Gon Faces
- iv. Freestyle Usability Improvements
- v. Using G3 Connections
- vi. Creating Periodic and NonPeriodic Closed Curves
- vii. Using the Drop Curve Option
- viii. Preserving Analytic Geometry Enhancement

*Knowledge Check Questions***Module 7. Sheetmetal Enhancements**

- i. Creating Twist Wall Features
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- ii. Utilizing Dependency Control with Punch and Die Forms
- iii. Creating Edge Bends
- iv. Flange Feature Close Corner Edge Treatment Enhancement
- v. Using Flexible Modeling in Sheetmetal Mode

Knowledge Check Questions



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