

Update to Creo Parametric 4.0 from Creo Parametric 2.0

Overview

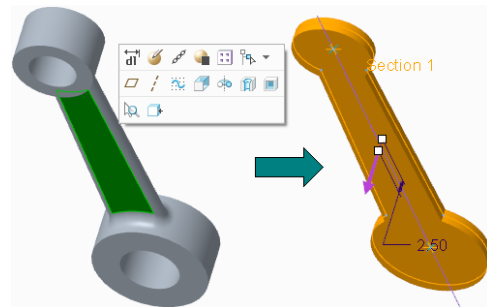
Course Code TRN-5125-T

Course Length 16 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. 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. Finally, you will investigate the new Assembly capabilities such as the built-in hardware library and the Intelligent Fastener extension, as well as enhanced Assembly functionality such as heterogeneous (multi-CAD) assembly and the Design Exploration extension.

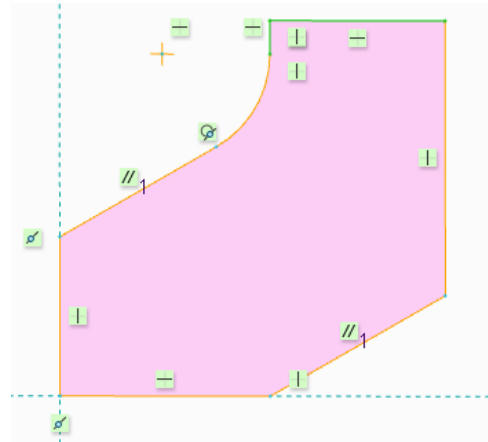
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0 (F000)



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
- Use the Intelligent Fastener extension in Creo Parametric 4.0
- Use the Design Exploration extension in Creo Parametric 4.0
- Use Unite Technology in Creo Parametric 4.0



Prerequisites

- Introduction to Creo Parametric 2.0, or equivalent experience with Creo Parametric 2.0

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
Module	2	Part Modeling Enhancements
Module	3	Sketcher Enhancements
Module	4	Assembly Enhancements
Module	5	Drawing Enhancements

Day 2

Module	6	Surfacing Enhancements
Module	7	Sheetmetal Enhancements
Module	8	Utilizing Intelligent Fasteners
Module	9	Utilizing Design Exploration
Module	10	Using Unite Technology

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
- xix. File Save Enhancements
- xx. Active Window Enhancements
- xxi. Graphics Enhancements
- xxii. Applying Realistic Appearances
- xxiii. Model Orientation Enhancement
- xxiv. Active Component Enhancements

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
 - ix. Draft Enhancements
 - x. Pattern Enhancements
 - xi. UDF and Paste Special Enhancements
 - xii. Collapse Feature Enhancements
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- xiii. Feature Operation Enhancements
- xiv. Creating Constant Width Rounds
- xv. Creating Spinal Bends
- xvi. Creating Toroidal Bends
- xvii. Editing Feature References
- xviii. Replacing Feature References
- xix. Displaying Missing References

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
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- ix. Embedding Images in Drawings
- x. Replacing View Models
- xi. Non-Linear Cross-Hatching Enhancement
- xii. Text and Symbol Fonts Enhancement
- xiii. Printing Enhancements
- xiv. Note Creation Enhancements

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
- ix. Boundary Blend Enhancements
- x. Untrimming Surface Copies
- xi. Flattening Quilts
- xii. Analyzing Connections
- xiii. Previewing Style Features
- xiv. Reusing Creo Sketch Curves
- xv. Creating Curves from Isolines
- xvi. Joining Freestyle Geometry
- xvii. Aligning Freestyle Geometry

Knowledge Check Questions

Module 7. Sheetmetal Enhancements

- i. Creating Twist Wall Features
- 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
- vi. Rip Enhancements
- vii. Flatten Form Enhancements
- viii. Bending in Multiple Planes
- ix. Creating Multiple Bend Reliefs
- x. Bend Line Relief Placement
- xi. Creating Die Forms
- xii. Creating Die Forms Using Annotations

Knowledge Check Questions

Module 8. Utilizing Intelligent Fasteners

- i. Understanding the Intelligent Fastener Extension
- ii. Assembling Intelligent Fasteners
- iii. Manipulating Intelligent Fasteners
- iv. Assembling Intelligent Fasteners Using Advanced Options
- v. Manipulating Intelligent Fasteners Using Advanced Options

Knowledge Check Questions

Module 9. Utilizing Design Exploration

- i. Understanding Design Exploration
- ii. Exploring Part and Assembly Designs
- iii. Creating Design Exploration Branches
- iv. Opening and Saving Design Exploration Sessions
- v. Using Design Exploration Options
- vi. Utilizing Update Control with Copy Geometry Features

Knowledge Check Questions

Module 10. Using Unite Technology

- i. Understanding Unite Technology
- ii. Creating and using Unite Profiles
- iii. Using Unite Technology to Assemble Components
- iv. Making Changes to Non-Creo Data
- v. Incorporating Changes Made to Non-Creo Data Outside of Creo

Knowledge Check Questions
