

# Update to Creo Parametric 2.0 from Creo Elements/Pro 5.0

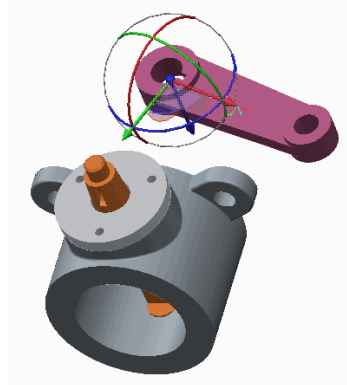
## Overview

Course Code TRN-3900-T

Course Length 1 Day

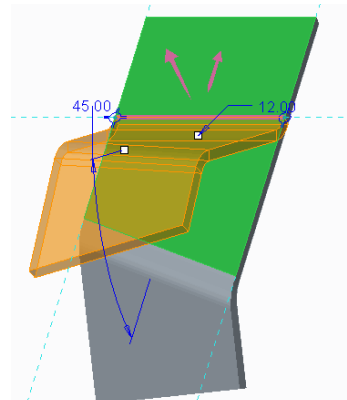
In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 2.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will become familiar 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, the new relationship constraints, and enhancements for dragging components. Then, you will examine the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the many updated Wall, Bend, and Relief tools, as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.



## Course Objectives

- Utilize the Interface enhancements in Creo Parametric
- Utilize the Sketcher enhancements in Creo Parametric
- Utilize the Modeling enhancements in Creo Parametric
- Utilize the Assembly enhancements in Creo Parametric
- Utilize the Drawing enhancements in Creo Parametric
- Utilize the Sheetmetal enhancements in Creo Parametric



## Prerequisites

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- Introduction to Pro/ENGINEER Wildfire 5.0, or equivalent experience with Pro/ENGINEER Wildfire 5.0 or Creo Elements/Pro 5.0

## Audience

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

### Day 1

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

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## Course Content

### Module 1. Interface Enhancements

- i. Understanding the Main Interface
- ii. Creo Graphics Enhancements
- iii. Understanding the Ribbon Interface
- iv. Customizing the Ribbon Interface
- v. Configuring Creo Options
- vi. Selection Enhancements
- vii. Locate in Model Tree Enhancement
- viii. Using the Measure Tools
- ix. Using the Measure Summary Tool
- x. Creating Planar Part Cross-Sections
- xi. Creating Assembly Cross-Sections
- xii. Anti-Alias Edge Display Enhancement

#### *Knowledge Check Questions*

### Module 2. Sketcher Enhancements

- i. Sketcher Interface Enhancements
- ii. Sketcher Workflow Enhancements
- iii. Sketching Rectangles and Parallelograms

#### *Knowledge Check Questions*

### Module 3. Part Modeling Enhancements

- i. Feature Preview Enhancements
  - ii. Understanding Regeneration and Auto Regeneration
  - iii. Editing Features
  - iv. Automatically Adding and Removing Material
  - v. Adding Taper to Extrude Features
  - vi. Creating Corner Chamfers
  - vii. Variable Section and Simple Sweep Consolidation
  - viii. Creating Sweeps with Open Trajectories
  - ix. Creating Sweeps with Closed Trajectories
  - x. Analyzing Sweep Feature Attributes
  - xi. Helical Sweep Enhancements
  - xii. Creating Helical Sweeps for Springs
  - xiii. Analyzing Helical Sweep Profile and Pitch Variations
  - xiv. Utilizing Variable Sections in Helical Sweeps
  - xv. Creating Blends by Selecting Parallel Sections
  - xvi. Creating Blends by Selecting Non-Parallel Sections
  - xvii. Creating Blends by Sketching Sections
  - xviii. Analyzing Blend Section Tools
  - xix. Analyzing Blend Options
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- xx. Analyzing Blend Tangency
- xxi. Creating Rotational Blends by Selecting Sections
- xxii. Creating Rotational Blends by Sketching Sections
- xxiii. Analyzing Rotational Blend Options
- xxiv. Analyzing Rotational Blend Tangency
- xxv. Creating Geometry Patterns
- xxvi. Creating a Curve From Equation
- xxvii. Creating a Curve from a Cross-Section
- xxviii. Creating Curves Through a Point or Vertex
- xxix. Creating a Curve Through a Point Array
- xxx. Creating Cosmetic Sketches
- xxxi. Creating Cosmetic Threads

*Knowledge Check Questions*

**Module 4. Assembly Enhancements**

- i. Reference Dialog Enhancements
- ii. Assembly Mass Properties Enhancements
- iii. Component Rename Enhancements
- iv. Assembly Appearance Enhancements
- v. Assembly Model Tree Enhancements
- vi. Selecting Multiple Components
- vii. Orienting Components
- viii. Creating Coincident Constraints using Geometry
- ix. Creating Coincident Constraints using Datum Features
- x. Creating Distance Constraints
- xi. Creating Parallel, Normal, and Angle Constraints
- xii. Assembling using Automatic
- xiii. Retrieving Assembly Subsets
- xiv. Lightweight Graphics Representations
- xv. Restructuring and Reordering Assembly Components
- xvi. Configuring Constraint Sets with Parameters
- xvii. Reviewing Assembly Changes

*Knowledge Check Questions*

**Module 5. Drawing Enhancements**

- i. Detail Options Enhancements
  - ii. Detailing Ribbon and Selection Enhancements
  - iii. Detailing Graphics Enhancements
  - iv. Prefix and Suffix Enhancements
  - v. Feature-Specific Text Enhancements
  - vi. Detailing Arrow Enhancements
  - vii. Detail and Section View Enhancements
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- viii. Dimension Text Orientation Enhancements
- ix. View Scale Enhancements
- x. BOM Enhancements
- xi. Inserting Tables
- xii. Editing Table Properties
- xiii. Creating Tables from File
- xiv. Creating BOM Balloons
- xv. Dimension and Annotation Enhancements
- xvi. Creating Combination States and Showing Annotations
- xvii. Creating Annotation Planes and Annotation Features
- xviii. Managing Annotation Display

*Knowledge Check Questions*

#### **Module 6. Sheetmetal Enhancements**

- i. Sheetmetal Properties Enhancements
- ii. Miscellaneous Sheetmetal Enhancements
- iii. Extending and Trimming Walls
- iv. Creating Offset Walls
- v. Creating Bend Features
- vi. Adding Transition to Bends
- vii. Creating Planar Bends
- viii. Creating Edge Bends
- ix. Creating Unbend Features
- x. Creating Bend Back Features
- xi. Joining Walls
- xii. Creating Sketched Forms
- xiii. Flattening Forms and Unstamping Edges
- xiv. Creating Split Area Features
- xv. Creating Corner Relief
- xvi. Creating Rip Features
- xvii. Converting Solid Models to Sheetmetal
- xviii. Using Conversion Features
- xix. Previewing and Creating Flat Patterns
- xx. Creating Flat States

*Knowledge Check Questions*

#### **Module 7. Appendix**

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