

Update to Creo Parametric from Creo Parametric

Overview

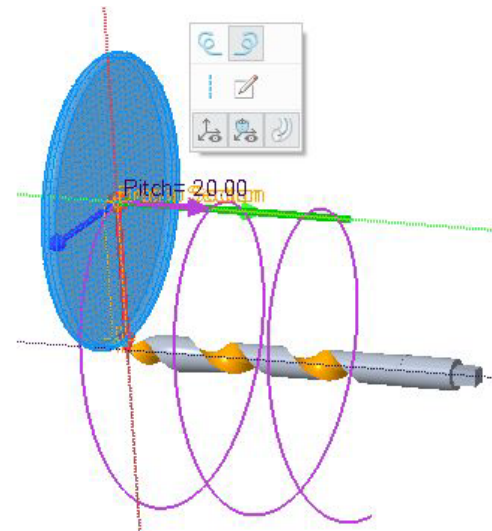
Course Length

8 Hours

In this course, you will learn how to use the new functionality enhancements in Creo Parametric, which will enable you to increase your productivity and improve your engineering designs. You will be introduced to new features and enhancements in Part Modeling, Assembly, Drawing, Surfacing, Sheetmetal, and Rendering, along with enhancements in the Interface. You will learn about the enhancements in Part Modeling features such as Use a Helix curve in Volume Helical Sweep, Create Datum Points by using the Project option, Configurable Drill Tip Angle for Hole feature, Default value of Extrude depth and Creating a Midplane using default Bisector. You will review various Drawing and Detailing enhancements such as Modernized Notes, Modernized Arrowheads, automatic Drawing naming, deleting Table Cell contents. You will learn about enhancements to Freestyle Surfacing features such as Mini toolbar, Modernized Freestyle tree, Suppress shapes in freestyle tree. You will also learn about enhancements to Style features such as improved Resolve Mode, creating default Connections in Style, use Soft Point Collector for editing a curve in Style. You will be introduced to enhancements of the Sheetmetal design features such as Flat Wall feature, Bend Offset Options, Mini toolbar for Bend Offset and Relief control options, Understand improved regeneration performance for Sheetmetal cuts. You will also be introduced to enhanced Rendering features such as using Rendering Studio for Mechanism and Design Animation, Use of Emissive Appearance to define the light source. You will learn about enhancements to Assembly design such as Identifying Active Components in the Model Tree, Using Zoom to Selected Reference from Reference Viewer, Use Access to Varied Items from the Inheritance feature.

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





Course Objectives

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

Prerequisites

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

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	Assembly Enhancements
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Module	4	Drawing Enhancements
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Module	5	Surfacing Enhancements
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Module	6	Sheetmetal Enhancements
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Module	7	Rendering Enhancements
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