

Update to latest version of Creo Parametric from older versions

Overview

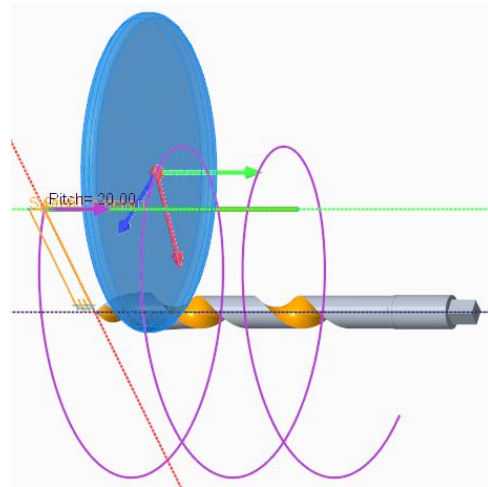
Course Length

16 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 Modelling, Assembly, Drawing, Surfacing, Sheetmetal and AR design visualization, along with enhancements in Mini Toolbar. You will learn about the new Part Modeling features such as Volume Helical Sweep and Sketch Region selection. You will examine the Part Modeling enhancements to features such as Draft controls for Rounds and Chamfers, Point Pattern workflows, Mirror, and Unite for the Inventor. You will review various Drawing and Detailing enhancements such as improved Undo/Redo support, Arrow Styles, new configuration options for Cross-hatching Pattern types. You will learn about enhancements to Freestyle Surfacing features such as Align to a non-G2 chain, Align Curvature Connection, Enable Box Creation Mode, Add Edge functionality, Symmetric Style Curve, along with new feature for Slice Mesh by Plan. You will be introduced to enhancements of the Sheet metal design such as Closed sections for the Bend Relief, new Corner Relief types, Sheetmetal Part Conversion, improved Sheetmetal Rounds and Chamfer behavior, and Flatten Representations. You will also be introduced to enhanced Mini Toolbar for the Style, For 3D and 2D Annotations, and Sketch Regions.

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 Sketcher 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 Augmented reality 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	Sketcher Enhancements
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Module	4	Assembly Enhancements
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Day 2

Module	5	Drawing Enhancements
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Module	6	Surfacing Enhancements
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Module	7	Sheetmetal Enhancements
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Module	8	Augmented Reality Design for Visualization
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