Update to Creo Parametric 5.0 from Creo Parametric 4.0

Overview
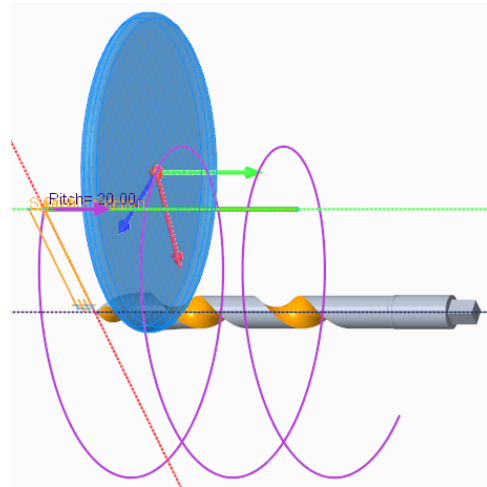
Course Code	TRN-5300-T
Course Length	8 Hours

In this course, you will learn how to use the new functionality enhancements in Creo Parametric 5.0, 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 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, and 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, and Symmetric Style Curve, along with a new feature for Slice Mesh by Plan. You will be introduced to enhancements of the Sheetmetal design such as Closed sections for the Bend Relief, new Corner Relief types, Sheetmetal Part Conversion, improved Sheetmetal Rounds and Chamfer behavior, and Flattened Representations. You will also be introduced to enhanced Mini Toolbar for the Style, For 3-D and 2-D 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 5.0 F000

Course Objectives

- Use the Interface enhancements in Creo Parametric 5.0
- Use the Part Modeling enhancements in Creo Parametric 5.0
- Use the Sketcher enhancements in Creo Parametric 5.0
- Use the Assembly Modeling enhancements in Creo Parametric 5.0
- Use the Drawing enhancements in Creo Parametric 5.0
- Use the Surfacing enhancements in Creo Parametric 5.0
- Use the Sheetmetal enhancements in Creo Parametric 5.0

Prerequisites

- Introduction to Creo Parametric 5.0, or equivalent experience with Creo Parametric 4.0

Audience

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

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Part Modeling Enhancements
Module	3	Assembly Enhancements
Module	4	Drawing Enhancements
Module	5	Surfacing Enhancements
Module	6	Sheetmetal Enhancements
Module	7	Augmented Reality Design for Visualization