

Update to Creo Parametric from Creo Parametric

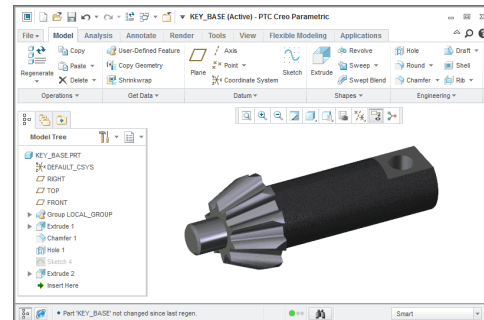
Overview

Course Length

1 Day

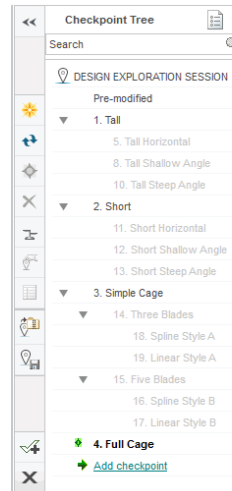
In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric. You will be introduced to user interface appearance enhancements such as the active window and right-click menu enhancements. You will examine the Part Modeling enhancements to features such as Draft, Pattern, UDF, Rounds, and Spinal Bend. You will experiment with the new Edit References dialog box on part models and study the surfacing enhancements to surface copy, flatten quilt, boundary blends, and the new connection analysis tool. Freestyle surfacing enhancements to the Join tool and aligning to geometry will also be covered. You will also learn about Flexible Modeling enhancements to tangency propagation, flexible patterns, round/chamfer recognition, and the flexible move tool. 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. You will examine the new Dimension and Note functionality for 2-D drawings and review various detailing enhancements. Finally, you will learn to use the many updated tools in Sheetmetal mode such as Die Forms, Bend Relief, and Rip, as well as the enhancement for bending in multiple planes.

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

- Use the Interface enhancements in Creo Parametric.
- Use the Part Modeling enhancements in Creo Parametric.
- Use the Surfacing enhancements in Creo Parametric.
- Use the Flexible Modeling enhancements in Creo Parametric.
- Use the Intelligent Fastener extension in Creo Parametric.
- Use the Design Exploration extension in Creo Parametric.
- Use the Drawing enhancements in Creo Parametric.
- Use the Sheetmetal enhancements in Creo Parametric.



Design Exploration session : Full Cage
Session name: FAN_DEX

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.



Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Part Modeling Enhancements
Module	3	Surfacing Enhancements
Module	4	Flexible Modeling Enhancements
Module	5	Utilizing Intelligent Fasteners
Module	6	Utilizing Design Exploration
Module	7	Drawing Enhancements
Module	8	Sheetmetal Enhancements
