

Update to Creo Parametric from Pro/ENGINEER Wildfire

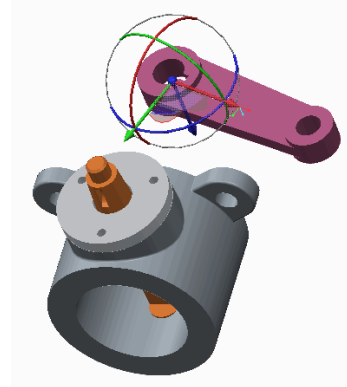
Overview

Course Length

2 Days

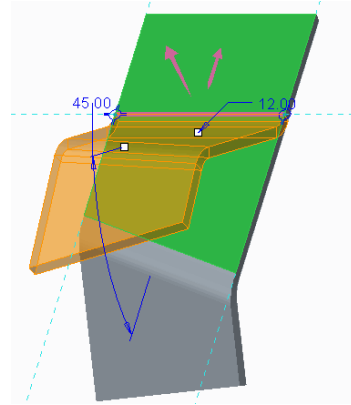
In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. The new measure and sectioning interfaces will also be examined. Next, you will study the Sketcher workflow and reference enhancements, as well as Sketcher constraint, geometry, and diagnostics enhancements. Part modeling enhancements to features such as Extrude, Trajectory Rib, Point Pattern, Corner Chamfer, Sweeps, Blends, UDFs, 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, enhancements for dragging components, explode enhancements, simplified rep enhancements, and dynamic gear enhancements. Then you will learn how to identify and resolve part and assembly failures on the fly, without entering Resolve mode. Next, you will examine the new Drawing tree and drawing sheets tabs, as well as showing annotations. Also in Drawing mode, you will learn 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 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

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

Audience

- 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

Module	1	Interface Enhancements
Module	2	Sketcher Enhancements
Module	3	Part Modeling Enhancements
Module	4	Assembly Enhancements

Day 2

Module	5	Resolving Failures
Module	6	Drawing Enhancements
Module	7	Sheetmetal Enhancements
Module	8	Appendix
