

# Introduction to Creo Parametric for SolidWorks Users

## Overview

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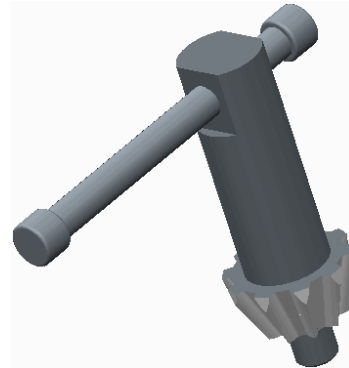
Course Length

24 Hours

In this course, you will learn about core modeling skills. This comprehensive, hands-on course is specifically designed for existing SolidWorks users who want to become proficient with Creo Parametric as quickly as possible. Topics include understanding the interface and basic Creo Parametric concepts, selecting and editing, sketching tools, and basic feature, part, and assembly creation. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing the course, you will be well prepared to work effectively on product design projects using Creo Parametric.

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.

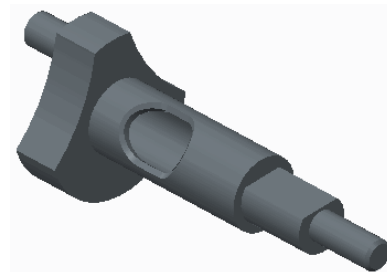
This course has been developed using Creo Parametric.




## Course Objectives

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- Learn the basic Creo Parametric modeling process
- Learn how to use the Creo Parametric interface
- Select and edit geometry, features, and models
- Sketch geometry and use tools
- Create sketches for features
- Create extrudes, revolves, and profile ribs
- Utilize internal sketches
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Assemble with constraints
- Lay out drawings and create views
- Create drawing annotations



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- Investigate parent/child relationships
  - Resolve failures and seek help
  - Complete a comprehensive design project

## **Prerequisites**

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- Knowledge of SolidWorks, including an understanding of solid modeling, feature-based, parametric, and associative concepts

## **Audience**

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- This course is intended for product designers, drafters, and industrial/conceptual designers. People in related roles can also benefit from taking this course.
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## Agenda

### Day 1

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Module	1	Introduction to the Creo Parametric Basic Modeling Process
Module	2	Using the Creo Parametric Interface
Module	3	Editing Geometry, Features, and Models
Module	4	Sketcher Geometry and Tools

### Day 2

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Module	5	Creating Extrudes, Revolves, and Ribs
Module	6	Creating Sweeps and Blends
Module	7	Creating Holes, Shells, and Draft
Module	8	Creating Rounds and Chamfers

### Day 3

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Module	9	Project I
Module	10	Assembling with Constraints
Module	11	Introduction to Drawings
Module	12	Resolving Failures and Seeking Help

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