

Creo Parametric 5.0 for SolidWorks Users

Overview

Course Code TRN-5326-T

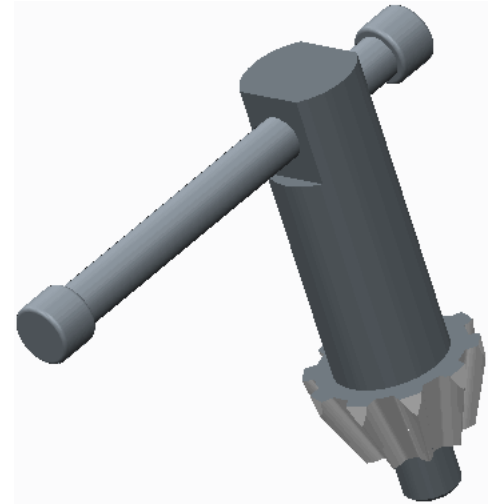
Course Length 24 Hours

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.

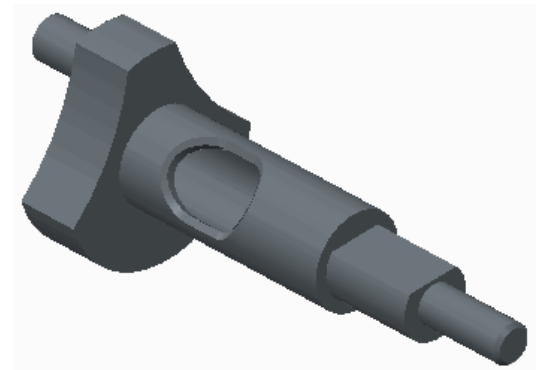
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 5.0.1.0.



Course Objectives

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



Prerequisites

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

Audience

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

Agenda

Day 1

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

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

Module 9 Project I

Module 10 Assembling with Constraints

Module 11 Introduction to Drawings

Module 12 Resolving Failures and Seeking Help
