

Introduction to Creo Parametric 5.0

Overview

Course Code TRN-5301-T

Course Length 40 Hours

In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 5.0. Topics include sketching, part modeling, assemblies, drawings, and basic model management techniques. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts, assemblies, and drawings. After completing the course, you will be well prepared to work effectively on product design projects using Creo Parametric 5.0. 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

- Learn the basic Creo Parametric modelling process
- Understand Creo Parametric concepts
- 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 datum planes and datum axes
- Create extrudes, revolves, and profile ribs
- Utilize internal sketches and embedded datums
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Group, copy, and mirror items
- Create patterns
- Measure and inspect models
- Assemble with constraints
- Assemble with connections
- 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
- Explode assemblies
- Lay out drawings and create views
- Create drawing annotations
- Use layers
- Investigate parent/child relationships
- Capture and manage design intent
- Resolve failures and seek help
- Comprehensive two-part Design Project

Prerequisites

- None

Audience

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

Day 1

Module 1 Introduction to the Creo Parametric Basic Modeling Process

Module 2 Understanding Creo Parametric Concepts

Module 3 Using the Creo Parametric Interface

Module 4 Selecting Geometry, Features, and Models

Module 5 Editing Geometry, Features, and Models

Module 6 Creating Sketcher Geometry

Day 2

Module 7 Using Sketcher Tools

Module 8 Creating Sketches for Features

Module 9 Creating Datum Features: Planes and Axes

Module 10 Creating Extrudes, Revolves, and Ribs

Module 11 Sketcher Workflow

Module 12 Creating Sweeps and Blends

Day 3

Module 13 Creating Holes, Shells, and Draft

Module 14 Creating Rounds and Chamfers

Module 15 Project I

Module 16 Group, Copy, and Mirror Tools

Module 17 Creating Patterns

Module 18 Measuring and Inspecting Models

Day 4

Module 19 Assembling with Constraints

Module 20 Assembling with Connections

Module 21 Exploding Assemblies

Module 22 Drawing Layout and Views

Module 23 Creating Drawing Annotations

Module 24 Using Layers

Day 5

Module 25 Investigating Parent/Child Relationships

Module 26 Capturing and Managing Design Intent

Module 27 Resolving Failures and Seeking Help

Module 28 Project II