

Find a Class

COURSE TYPE

- Technology

DURATION

- 3 Hours

DELIVERY METHOD

- Online
- Regional Classroom
- Onsite

DELIVERY LANGUAGES

- English
- Français
- Deutsch
- Italiano
- 日本語
- 中文(简体)
- 中文(繁體)

COURSE OVERVIEW

In this course you will gain a general understanding of different Creo capabilities, as well as some of the different workflows used. You learn the areas of the user interface, model navigation, and the different display styles. This course includes expert-led lectures, demos, and some exercises.

TARGET AUDIENCE

Users with minimal Creo Parametric experience will benefit from this course.

PREREQUISITES

- None

LEARNING OBJECTIVES

- Explore the Learn Online overview process
- Learn high-level Creo Parametric capabilities
- Review the Creo Design Process framework
- Tour of the Creo Parametric Fundamentals curriculum plan
- Learn the Creo user interface, model navigation, and different display styles
- Review basic Creo Parametric concepts and benefits of solid modeling

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COURSE OVERVIEW

In this course, you will learn how to create parts with PTC Creo Parametric using the basic sketch, editing, and feature creation techniques.

TARGET AUDIENCE

Users with minimal Creo Parametric experience will benefit from this course.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Explore the Part Modeling framework
- Edit existing features
- Explore solid modeling concepts
- Create extrude features from sketches
- Create revolve features from sketches
- Create sketches using design intent
- Model a part using techniques learned in this course

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COURSE OVERVIEW

In this course, you will review the Part Modeling framework used to create part models. You will also learn how to create parts with PTC Creo Parametric using basic feature creation techniques. These features round out the remainder of the basic geometry types you can use to create models.

TARGET AUDIENCE

Users with minimal Creo Parametric experience will benefit from this course.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Review the Part Modeling framework
- Create Sweep features with open and closed trajectories
- Create Blend features by selecting and sketching sections
- Create different types of Hole features
- Create basic Shell features
- Create basic Draft features
- Create Rounds and Chamfers by selecting various reference types
- Create Round and Chamfer sets

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COURSE OVERVIEW

In this course, you will learn how to create assembly systems using core assembly constraints. You will learn how to create explode states in an assembly. You will also learn how to use assembly features.

TARGET AUDIENCE

Users with little to no Creo Parametric experience will benefit from this course.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Explore the Assembly Systems framework
- Explore assembly concepts
- Create a new assembly
- Assemble components using various constraints
- Create assembly explode states
- Create assembly features

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COURSE OVERVIEW

In this course, you will explore the 2D drawing capabilities of Creo Parametric. You will learn how to create detailed 2D drawings from 3D models using PTC Creo Parametric.

TARGET AUDIENCE

Users with minimal Creo Parametric experience will benefit from this course.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Review the different types of Creo documentation
- Explore the 2D Drawing framework
- Create drawings with formats
- Add and modify drawing views
- Create tables in drawings
- Add BOM balloons
- Create and modify annotations
- Analyze drawing associativity

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COURSE OVERVIEW

In this course, you will explore the analysis capabilities of Creo Parametric within the modeling and assembly frameworks. You will learn how to analyze models and assemblies with PTC Creo Parametric using basic analysis techniques to validate design specifications.

TARGET AUDIENCE

Users with minimal Creo Parametric experience will benefit from this course.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Review the Analysis framework
- Explore the analyses available using Creo Parametric
- Explore model properties
- Use measurement tools on models and assemblies
- Identify and correct interferences
- Review the basic principles of Finite Element Analysis
- Perform a Finite Element Analysis using Creo Simulate Lite

Creo: Sketching Productivity Tools

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COURSE OVERVIEW

In this course, you will learn how to create robust sketches using PTC Creo Parametric. You will learn additional sketching techniques using the tools available for creating sketch geometry.

TARGET AUDIENCE

Users with fundamental knowledge of Creo Parametric looking to increase efficiency and productivity in the tool

PREREQUISITES

- Creo Fundamentals Certification

LEARNING OBJECTIVES

- Explore the Sketcher Creation framework
- Explore the various ways in which to reuse geometry
- Use additional sketching tools to facilitate model design
- Create different dimension schemes to capture design intent

Creo: Modeling Productivity Tools

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COURSE OVERVIEW

In this course, you will learn how to speed up and improve the quality of your product design using modeling productivity tools.

TARGET AUDIENCE

Users with a fundamental knowledge of Creo Parametric looking to increase efficiency and productivity in the tool.

PREREQUISITES

- Creo: Fundamentals Overview

LEARNING OBJECTIVES

- Manage design intent in sketches and features
- Edit features and parts using the reorder and insert mode
- Create features by copying and pasting feature(s)
- Use Mirror and Pattern tools to duplicate features
- Create and manage layers

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COURSE OVERVIEW

In this course you will learn how to speed up the assembly process using model management and creation tools. You will learn how to assemble models with PTC Creo Parametric using additional assembly techniques to create robust assemblies.

TARGET AUDIENCE

Users with fundamentals knowledge of Creo Parametric looking to increase efficiency and productivity in the tool.

PREREQUISITES

- Creo Fundamentals Certification

LEARNING OBJECTIVES

- Review the Assembly Systems Framework
- Manage components within an assembly
- Mirror components and assemblies
- Modify assembly states
- Create reference patterns
- Create component interfaces
- Manage assembly failures and interferences
- Replace components in an assembly

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COURSE OVERVIEW

In this course you will learn about managing large assemblies using PTC Creo Parametric. You will learn how to work with large assemblies to increase efficiency and loading times.

TARGET AUDIENCE

Users with a fundamental knowledge of Creo Parametric looking to increase productivity when working with large assemblies.

PREREQUISITES

- Creo Fundamentals Certification

LEARNING OBJECTIVES

- Review the Assembly Systems Framework
- Create shrinkwrap features and models
- Manage simplified representations in assemblies
- Examine simplified representation methods
- Manage simplified representations in parts