This course constitutes a comprehensive review of configuring Creo Parametric with Windchill. In addition, the content is built to support topic-based course configurations. After completing this course, you will be prepared to configure Creo Parametric to work in a Windchill environment.

In this course, you will learn key capabilities and features of configuring Creo Parametric to work with Windchill, such as the Windchill architecture that is used to support Creo Parametric. Additionally, you will be taught about Creo Parametric configuration files and options, and the preferences in Windchill that together configure your working environment for the management of your Creo Parametric CAD documents.

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 Windchill 11.0 F000 / Creo Parametric 3.0 M070.
Course Objectives

- Understand the Windchill workspace architecture
- Understand the Creo Parametric installation folder structure
- Use configuration elements to configure Creo Parametric to work with Windchill
- Use and edit Creo Parametric Startup Files (PSF)
- Configure Windchill CAD document management preferences in Windchill
- Map Creo Parametric parameters to Windchill attributes
Prerequisites

- Locate and View Information with Windchill 11.0 or equivalent
- Basic understanding of Creo Parametric

Audience

- This course is intended for System Administrators and end users.
| Module | 1 | Creo Parametric Data Administration |