Creo Parametric Data Administration in Windchill 10.2

Overview

Course Code  WBT-4335-0
Course Length  1 Hour

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. You will also have a basic understanding of the Creo Parametric and Windchill architectures.

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 the course, you will complete a course assessment in Pro/FICENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

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

- Locate and View Information with Windchill 10.2 or equivalent
- Basic understanding of Creo Parametric

Audience

- This course is intended for System Administrators and end users.
<table>
<thead>
<tr>
<th>Module</th>
<th>Creo Parametric Data Administration</th>
</tr>
</thead>
</table>

Table of Contents