The PTC Windchill PDMLink 11.0 MCAD Data Management Process for PTC Creo Parametric

Overview

Course Code       WBT-4730-0
Course Length     3 Hours

In this Process-based course, you will learn about the Windchill PDMLink Creo Parametric data management solution. You will review how to register the Windchill server with Creo and learn about the Creo Windchill access points. Additionally, you will be taught about the processes of Managing Design Data and Managing Design Development.

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 and Creo Parametric 3.0 M070.

Course Objectives

- Understand the purpose of CAD documents
- Understand the purpose of Windchill parts
- Register the Windchill server with Creo
- Understand the Manage Design Data process
- Understand the Manage Design Development process
Prerequisites

Audience

- This course is intended for any user who creates and/or modifies data using Creo Parametric. People in related roles will also benefit from taking this course.
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