Update to Creo Elements/Pro Data Management with Windchill PDMLink 10.0 from Windchill PDMLink 8.0

Overview

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<th>Course Code</th>
<th>WBT-3142</th>
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<td>Course Length</td>
<td>1 Hour</td>
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In this course, you will learn about enhancements to managing Creo Elements/Pro data in Windchill. Specifically, you will learn about the update to Creo Elements/Pro Data Management with Windchill PDMLink 10.0 from Windchill PDMLink 8.0. These enhancements include general usability, changes to the user interface, and the addition of new features available through the workgroup manager and Windchill Web pages.

At the end of each module, you will complete a skills assessment. The questions are used to help reinforce your understanding of the module topics and form the basis for review of any topics, if necessary.

Course Objectives

- Identify Windchill user interface enhancements that improve working with Creo Elements/Pro data
- Use advanced Check out and Revise options
- Use the Move action to relocate design documents
- Use the Auto-associate page to create relationships between WTParts and CAD documents
- Identify Creo Elements/Pro file dependencies
- View and collect related data using Creo Elements/Pro dependencies in Windchill
- Compare structures
- Identify features to perform Top Down Design
- Use workspace frames
- Identify new features for Heterogeneous Design in Context (HDIC)
Prerequisites

• Working knowledge of Windchill PDMLink 8.0 Pro/ENGINEER Data Management

Audience

• This course is designed for users who need to update their Creo Elements/Pro data management knowledge and skills from Windchill PDMLink 8.0 to work in the Windchill PDMLink 10.0 environment
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