

Advanced PTC Creo Parametric Data Management Processes and Techniques Using PTC Windchill PDMLink 11.1

Overview

Course Code TRN-5232-T

Course Length 8 Hours

In this course, you will learn advanced Creo CAD data management tips, techniques, processes, and best practices that will enhance your Creo CAD data management strategies.

This course has been developed using Windchill 11.1 F000/Creo Parametric 4.0 M030.

Course Objectives

- Register the Windchill server with Creo.
- Configure the Creo Parametric user interface.
- Understand the use of Creo Parametric CAD data management configuration settings.
- Understand the use of Windchill preference to manage the CAD data.
- Understand how to handle family table in Windchill.
- Understand the use of the inheritance feature.
- Use ProjectLink as a design sandbox.
- Understand the importance of simplified representations.
- Create and check in external simplified representations.
- Understand the incomplete objects.
- Understand how to resolve incomplete objects.
- Understand the use of the Design Exploration extension to explore the part designs.

Prerequisites

- TRN-5230
- Basic understanding of using Windchill for Creo data management.
- Experience using Creo Parametric 4.0 for basic modeling and assembly.

Audience

- This course is intended for Creo Parametric and Windchill administrators. People in related roles will also benefit from taking this course.
 - Type course audience here. Course Audience will be listed in print.
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Agenda

Day 1

Module	1	Configuring Creo Parametric and Windchill
Module	2	Creo Parametric Configuration Settings and Windchill Preferences
Module	3	Family Tables
Module	4	Using ProjectLink As Design Sandbox
Module	5	Simplified Representations
Module	6	Incomplete Objects
Module	7	Using Design Exploration

Course Content

Module 1. Configuring Creo Parametric and Windchill

- i. Use the Creo Parametric Embedded Browser for Windchill CAD Operations
- ii. Configuring the Creo Parametric Browser
- iii. Configuring the Creo Parametric User Interface
- iv. Use Unique Base File Names
- v. Standardized Windchill Server Alias Name
- vi. Unique Local Cache Requirement for Each Windchill Account
- vii. Creo Parametric Start Parts and Windchill CAD Document Templates
- viii. Using Start Parts
- ix. Using CAD Templates in New Designs
- x. Using Library Parts
- xi. Retrieving Components from a Library

Module 2. Creo Parametric Configuration Settings and Windchill Preferences

- i. Environment Variables
- ii. CAD Data Management - Configuration Options
- iii. Managing Virtual Workspaces in the Server Management Utility
- iv. Undocking the Embedded Browser Window
- v. CAD Data Management Preferences
- vi. Revise Preferences
- vii. Save As Preferences
- viii. Mapping System Attributes and File Properties Preferences
- ix. Workspace Preferences
- x. Displaying Workspace
- xi. Enabling As Stored Configuration
- xii. Enabling Baseline Creation
- xiii. Using ModelCHECK to Authorize Check In
- xiv. Managing Secondary Content Files with Windchill

Module 3. Family Tables

- i. Family Table Overview
 - ii. Family Table Structure
 - iii. Viewing Family Table Information
 - iv. Creating Family Tables
 - v. Managing Family Tables
 - vi. Understanding the Family Table Generic Usage
 - vii. Generic Modification Matrix
 - viii. Individual Instance Modification Matrix
 - ix. Modifying Family Tables
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- x. Saving Family Table Items As New Items
- xi. Converting a Family Table Instance into a Standalone Object
- xii. About Inheritance Features
- xiii. Creating Inheritance Features
- xiv. Defining Inheritance Feature Dependency
- xv. Defining Varied Items in an Inheritance Feature

Module 4. Using ProjectLink As Design Sandbox

- i. Windchill Integrated Environment
- ii. Using ProjectLink As Design Sandbox
- iii. Adding PDM Objects to a Project
- iv. Add to Project Option
- v. Sharing CAD Integration Objects
- vi. Performing a PDM Check Out
- vii. PDM Objects in Projects - Version Scheme

Module 5. Simplified Representations

- i. Simplified Representations
- ii. Retrieving Simplified Representations On-the-Fly
- iii. Using External Simplified Representations

Module 6. Incomplete Objects

- i. Incomplete Objects
- ii. Types of Incomplete Objects
- iii. Causes of Incomplete Objects
- iv. Preference-Incomplete Object Resolution
- v. Resolving Incomplete Objects from the Information Page
- vi. Resolving Incomplete Objects During Check In
- vii. Resolving Incomplete Objects Using Reference Viewer

Module 7. Using Design Exploration

- i. Understanding Design Exploration
- ii. Exploring Part and Assembly Designs
- iii. Opening and Saving Design Exploration Sessions



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