

Mastering CAD Driven Product Structure Management with PTC Windchill PDMLink 11.1

Overview

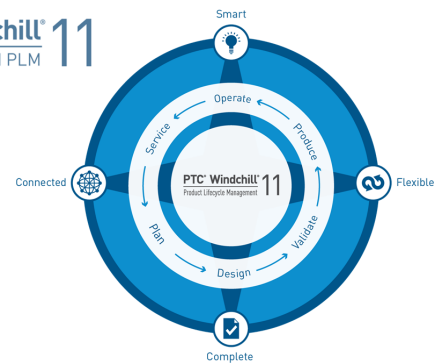
Course Code TRN-5231-T

Course Length 8 Hours

In this course, you will learn how to create and manage a CAD-driven product structure. You will learn how to work with product structures using a combined Enterprise top-down and bottom-up approach. Additionally, you will learn about the Windchill tools, processes, and best practices for advanced component creation and management techniques in the product development process.

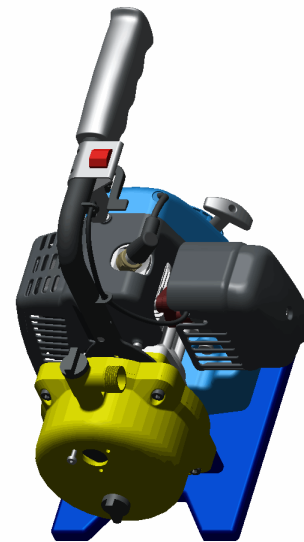
This course has been developed using Windchill 11.1 F000 / Creo Parametric 4.0 M030.

PTC® Windchill® 11
Smart Connected PLM



Course Objectives

- Define CAD-driven product structure management
- Understand the Windchill tools and processes that support CAD-driven product structure management
- Understand how to use Model Check
- Understand Enterprise top-down and bottom-up design methods
- Identify the best method for auto associating WTParts
- Use Creo View to position components
- Manipulate quantities in both Creo Parametric and Windchill
- Use bulk items in Creo Parametric designs



Prerequisites

- TRN-5211
- TRN-5230
- TRN-5215
- Thorough understanding of using Windchill for Creo CAD data management
- Experience using Creo Parametric 3.0 for basic modeling and assembly

Audience

- This course is intended for end users and system administrators seeking a better understanding of CAD-driven design. People in related roles will also benefit from taking this course.
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Agenda

Day 1

Module	1	Configuring for CAD Data in PDMLink
Module	2	Controlling Behaviour of CAD and WTPart Association in PDMLink
Module	3	Making the Most of Product Structure Browser (PSB)
Module	4	Data Management for CAD Documents
Module	5	Using Visualization Capabilities
Module	6	Enterprise Top-Down Design

Course Content

Module 1. Configuring for CAD Data in PDMLink

- i. CAD-Driven Product Structure Management Definition
- ii. CAD-Driven Product Structure Management Practice
- iii. System Administration, Configuration and Process considerations
- iv. Understanding ModelCHECK
- v. Example ModelCHECK Start Check File
- vi. ModelCHECK Results Example
- vii. Creo Parametric Restricted Parameter File
- viii. Understanding Control Characteristics
- ix. Defining Control Characteristics
- x. Multiple CAD Document Subtypes
- xi. CAD Document Subtype Attributes

Module 2. Controlling Behaviour of CAD and WTPart Association in PDMLink

- i. Multiple WTPart Subtypes
- ii. Mapping CAD Document Subtype to a WTPart Subtype
- iii. WTPart Build Service Preferences
- iv. WTPart CAD Document Links
- v. WTPart CAD Document Links Example
- vi. Understanding Calculated and Content Links
- vii. Associating WTParts with CAD Documents
- viii. Automatically Associating WTParts in the Workspace
- ix. Independent Revision and Association Updates of CAD Documents and Related Windchill Parts
- x. Specifying the Build Status at Check In
- xi. Powered Pump 6600 Dataset
- xii. Disabling the WTPart Build for Specific CAD Documents
- xiii. Other Auto Associate Windchill Preferences

Module 3. Making the Most of Product Structure Browser (PSB)

- i. PSB Uses Tab
- ii. Usages and Occurrences
- iii. Show and Hide Related Information

Module 4. Data Management for CAD Documents

- i. Revising Design Documents
 - ii. Revising from the Non-Latest Version
 - iii. Revise and Check Out Action
 - iv. Using Save As for Duplicating CAD and Product Structures
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- v. Save As Support Using a Same Base Number Scheme for Creating New CAD Documents and Windchill Parts
- vi. Locking CAD Documents in the Workspace
- vii. Creo Parametric Custom and Auto Check In
- viii. Understanding the Check In Options
- ix. Understanding Gathering Parts
- x. Specifying a Gathering Part Using Creo Parametric
- xi. Using the Workspace Auto Associate Action

Module 5. Using Visualization Capabilities

- i. PSB Visualization Tab
- ii. Translating Components in Creo View MCAD
- iii. Rotating Components in Creo View MCAD
- iv. Capturing Component Locations in Creo View MCAD

Module 6. Enterprise Top-Down Design

- i. Enterprise Top-Down Design
 - ii. Enterprise Top-Down Design Process
 - iii. Creating Windchill Parts
 - iv. Building Product Structures
 - v. Using Compare
 - vi. Compare Tool Layout
 - vii. Opening a Compare Tool Reverse Build CAD Assembly in Creo Parametric
 - viii. PSB Occurrences Tab
 - ix. Adding Quantities Using Windchill
 - x. Viewing Windchill Changes Using Creo Parametric
 - xi. Removing Occurrences Using Windchill
 - xii. Using Bulk Items
 - xiii. Windchill Line and Find Numbers
 - xiv. Modifying Line and Find Numbers Top-Down Support
 - xv. Modifying Line and Find Numbers Bottom-Up Support
 - xvi. Showing Line and Find Numbers in Creo Parametric Drawings
 - xvii. CAD-Driven Process Specific Recommendations
 - xviii. CAD-Driven CAD Specific Recommendations
 - xix. CAD-Driven WTPart Specific Recommendations
 - xx. CAD-Driven Windchill Configuration Specific Recommendations
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