

# Creo Parametric Data Management with Windchill PDMLink 10.2

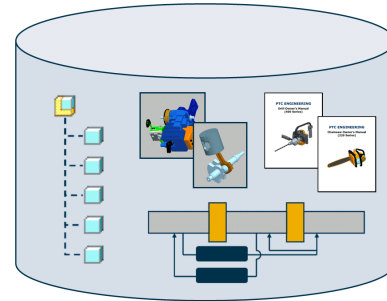
## Overview

Course Code TRN-4300-T

Course Length 1 Day

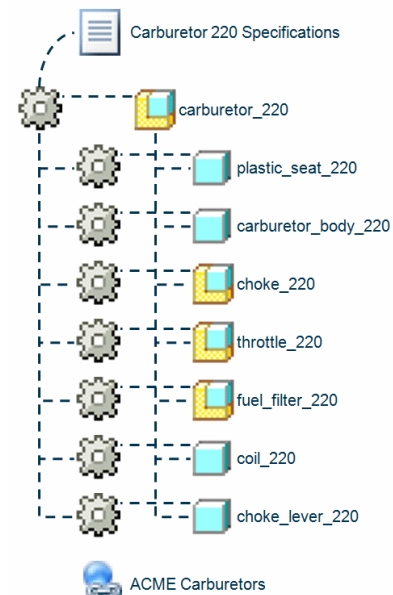
In this course, you will learn key capabilities and features of Windchill PDMLink CAD Data Management and how to use Windchill PDMLink to manage Creo Parametric design information. Specifically, you will focus on the day-to-day data management tasks required by the Creo Parametric user community, including how to use the CAD data structure to associate and build the product structure. You will also learn how to use Windchill to perform basic PDM functions on Creo Parametric CAD Documents and their associated Windchill parts. After completing this course, you will be prepared to complete basic Windchill PDMLink CAD data management tasks and activities. You will also have a better understanding of Windchill CAD data management processes.

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 Pro/FICIENCY intended to evaluate your understanding of the course as a whole.



## Course Objectives

- Locate design information
- View design information
- Create new designs and share design information
- Modify and manage existing design information
- Understand Windchill parts and their association to CAD documents
- Understand file relationships and dependencies
- Manage family tables
- Use workspace frames



## Prerequisites

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- A basic working knowledge of Creo Parametric 2.0
- Completion of any one of the following courses:
  - Introduction to Windchill PDMLink 10.2 for Light Users
  - Introduction to Windchill PDMLink 10.2 for Heavy Users
  - Introduction to Windchill PDMLink 10.2 for the Implementation Team

## Audience

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- This course is intended for end users and System Administrators. People in related roles will also benefit from taking this course.
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## Agenda

### Day 1

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Module	1	Getting Started with Creo Parametric Data Management
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Module	2	Initiating New Designs and Sharing Design Data
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Module	3	Modifying and Managing Design Data
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Module	4	Understanding File Relationships and Dependencies
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Module	5	Managing Family Tables
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Module	6	Managing CAD Data
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## Course Content

### Module 1. Getting Started with Creo Parametric Data Management

- i. Creo Parametric and Windchill Connectivity
- ii. Windchill Integration
- iii. Windchill Integration – Creo Parametric Menus
- iv. Windchill Integration – Model Tree
- v. Windchill Integration – Folder Browser
- vi. Windchill Integration – Search
- vii. Windchill Integration – Embedded or Separate Browser
- viii. Windchill Integration – Event Management
- ix. Registering a Windchill Server
- x. Introduction to the Workspace
- xi. Workspace Layout
- xii. Creating Workspaces
- xiii. Changing the Active Workspace
- xiv. Working with Workspaces
- xv. Deleting Workspaces
- xvi. Changing Workspace Table Views
- xvii. Customizing Workspace Table Views
- xviii. Workspace Preferences – General
- xix. Workspace Preferences – Document Configuration Specification
- xx. Workspace Preferences – Part Configuration Specification
- xxi. Locating Design Information with Creo Parametric
- xxii. Searching for Design Information
- xxiii. Browsing for Design Information
- xxiv. Viewing CAD Document Information
- xxv. Viewing CAD Document Content
- xxvi. Exposing PDM Information using Creo Parametric File Open
- xxvii. Exposing PDM Information in the Model Tree
- xxviii. Duplicate Names

#### *Knowledge Check Questions*

### Module 2. Initiating New Designs and Sharing Design Data

- i. Introduction to Windchill Parts
  - ii. Sharing Creo Parametric Designs
  - iii. Introducing New Designs to the Workspace
  - iv. Using CAD Templates in New Designs
  - v. Initiating New Designs Using New CAD Document
  - vi. Workspace Save As
  - vii. Introducing New Designs to the Commonsplace
  - viii. Understanding Check In
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- ix. Commonsplace Save As
- x. Introducing New WTParts
- xi. Generating Viewables
- xii. Creating Representations
- xiii. Monitoring Publish Jobs
- xiv. Adding CAD Documents to the Workspace
- xv. Removing CAD Documents from the Workspace
- xvi. Understanding Workspace Architecture
- xvii. Understanding Save, Upload, and Check In
- xviii. Understanding Web Browser Access to the Workspace
- xix. Naming – Creo Parametric New Object
- xx. Naming and Numbering Design Documents
- xxi. Naming and Numbering Policy Example
- xxii. Parameter to Attribute Mapping

#### *Knowledge Check Questions*

### **Module 3. Modifying and Managing Design Data**

- i. Understanding Workspace Objects
  - ii. Modifying Creo Parametric Designs
  - iii. Modification Process – Check Out a CAD Document
  - iv. Modification Process – Modify a CAD Document
  - v. Modification Process – Check In a CAD Document
  - vi. Methods of Checking Out CAD Documents
  - vii. Check Out of the Non-Latest Iteration
  - viii. Understanding Check Out Status and Options
  - ix. Understanding Download and Check Out
  - x. Understanding Advanced Check In Options
  - xi. Canceling Modifications
  - xii. Workspace Status
  - xiii. Locking CAD Documents in the Workspace
  - xiv. Understanding Gathering Parts
  - xv. Using Library Parts in Assemblies
  - xvi. Associating WTParts with CAD Documents
  - xvii. Automatically Associating Windchill Parts
  - xviii. Renaming CAD Documents
  - xix. Renaming WTParts
  - xx. Moving Design Documents
  - xxi. Revising Design Documents
  - xxii. Revising from the Non-Latest Version
  - xxiii. Revise and Check Out Action
  - xxiv. Modifying CAD Document Attributes
  - xxv. Creo Parametric Asynchronous Mode
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*Knowledge Check Questions***Module 4. Understanding File Relationships and Dependencies**

- i. Managing Dependencies and Links
- ii. WTPart CAD Document Links
- iii. WTPart CAD Document Links Example
- iv. Copying Viewables to Non-Owner Associated WTParts
- v. Sample Creo Parametric Dependencies
- vi. CAD Document Fundamentals for Models
- vii. CAD Document Fundamentals for Drawings
- viii. Viewing Dependency Information in Windchill
- ix. Processing Dependencies
- x. Managing Secondary Content Files with Windchill
- xi. Downloading Non-Latest Configurations
- xii. Using Creo Parametric to Compare PDM Model Versions
- xiii. Intelligent Hyperlinks within a Drawing Revision Block
- xiv. Typical Data Management Problems
- xv. Regeneration Failures
- xvi. Incomplete Objects
- xvii. Resolving Incomplete Objects

*Knowledge Check Questions***Module 5. Managing Family Tables**

- i. Family Table Overview
- ii. Viewing Family Table Information
- iii. Creating Family Tables
- iv. Managing Family Tables
- v. Generic Modification Matrix
- vi. Individual Instance Modification Matrix
- vii. Modifying Family Tables
- viii. Modify Instance Parameters/Attributes in a Workspace
- ix. Saving Family Table Items as New Items

*Knowledge Check Questions***Module 6. Managing CAD Data**

- i. Part Compare Browser Layout
  - ii. Part Compare
  - iii. Enterprise Top-Down Design
  - iv. Introduction to Workspace Frames
  - v. Accessing Workspace Frames
  - vi. Workspace Frames Details
  - vii. Updating and Synchronizing Workspace Objects
  - viii. Updating CAD Documents in the Workspace
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- ix. Synchronizing Objects in the Workspace
- x. Importing and Exporting CAD Data
- xi. Importing CAD Data
- xii. Exporting CAD Data
- xiii. Retrieving Simplified Representations On-the-Fly
- xiv. Using Flexible Components
- xv. Heterogeneous Design in Context (HDIC)
- xvi. Understanding Control Characteristics
- xvii. Defining Control Characteristics

*Knowledge Check Questions*

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