

Detailing with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

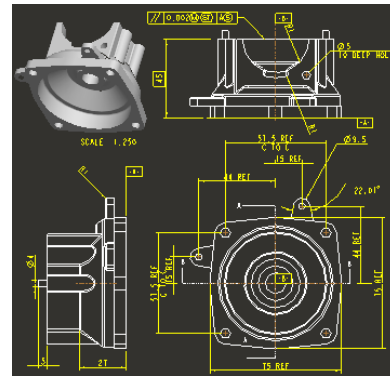
Overview

Course Code **TRN-2233-T**

Course Length **3 Days**

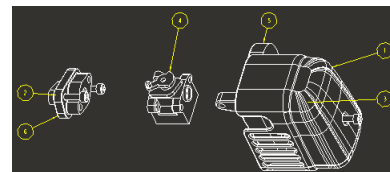
Detailing with Pro/ENGINEER Wildfire 5.0 is a comprehensive training course that teaches you how to quickly create detailed drawings using information captured within 3-D design models. In this course, you learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and associative nature of Pro/ENGINEER Wildfire 5.0 when configuring drawings.

After completing this course, you will be able to create production drawings suitable for manufacturing.



Course Objectives

- Understand the drawing development process.
- Create new drawings using formats and drawing templates.
- Create different types of views in drawings.
- Create dimensions and notes.
- Control display options using layers.
- Apply dimensional and geometric tolerances in drawings.
- Add draft geometry and symbols to drawings.
- Use layers in drawings to control the display of views and detail items.
- Create drawing tables and a bill of materials.
- Create drawing formats.
- Configure the drawing environment.
- Manage large drawings.



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0.

Audience

- This course is intended for mechanical designers, design engineers, and related roles.
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Agenda

Day 1

Module 1 Introduction to Drawings

Module 2 Creating New Drawings

Module 3 Creating Drawing Views

Day 2

Module 4 Adding Model Details to Drawings

Module 5 Adding Notes to Drawings

Module 6 Adding Tolerance Information

Module 7 Adding Draft Geometry and Symbols

Day 3

Module 8 Using Layers in Drawings

Module 9 Creating and Using Tables in Drawings

Module 10 Using Report Information in Drawings

Module 11 Creating Drawing Formats

Module 12 Configuring the Drawing Environment

Module 13 Managing Large Drawings
