PTC® university

creo™ elements/pro™

Curriculum Guide

Live Classroom Curriculum Guide

- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 4.0
- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 3.0
- Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Detailing with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Advanced Assembly Design with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Pro/ENGINEER Mechanica Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
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- Sheetmetal Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
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- Mechanism Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Mechanism Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Behavioral Modeling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Cabling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Piping using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Routed Systems Designer 9.0 for Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 2.0

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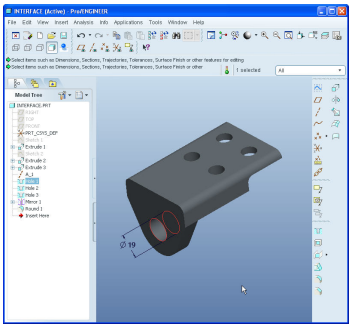
Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 4.0

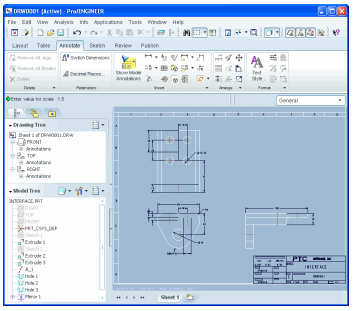
Overview

Course Code	TRN-2230-T
Course Length	1 Day



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0. First, you will become familiar with the enhancements to Sketcher, such as sketching datum features, parallelograms, and chamfers. Next, you will explore enhancements to Part mode, including creating trajectory ribs, point patterns, and the all-new resolve mode. You will also learn about new and enhanced Assembly capabilities such as the accessory window, the new explode interface, and enhancements to component repeat and restructure. Finally, you will examine the new ribbon interface for 2–D drawings, and review Sheetmetal enhancements such as patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 4.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 4.0.

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Sketcher Enhancements
Module	3	Part Modeling Enhancements
Module	4	Resolving Failures
Module	5	Assembly Enhancements
Module	6	Drawing Enhancements
Module	7	Sheetmetal Enhancements

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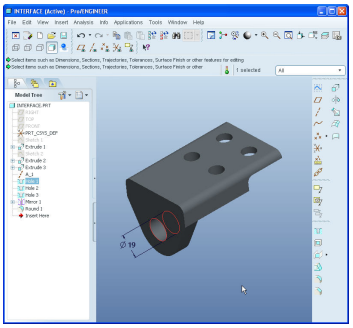
Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 3.0

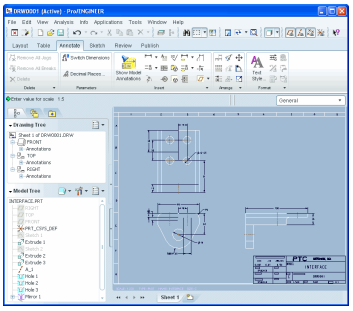
Overview

Course Code	TRN-2231-T
Course Length	2 Days



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0, when upgrading from Pro/ENGINEER Wildfire 3.0. First, changes to the interface will be explored, such as the new skin and screen colors. Next, you will become familiar with the enhancements to Sketcher geometry and tools, such as sketching parallelograms and chamfers, as well as using the sketcher diagnostic tools. Next, you will explore enhancements to Part mode, including creating trajectory ribs, pattern and UDF enhancements, the all-new resolve mode, and auto-round. You will also learn about Assembly enhancements such as the accessory window, the new reference viewer, improved simplified rep functionality, and mechanism enhancements to gears and belts. Next, you will examine the new ribbon interface and drawing tree for drawing mode, as well as numerous detailing enhancements. Finally, you will review Sheetmetal enhancements for flat and flange walls, in addition to patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Part Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Assembly Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 3.0.

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Sketcher Enhancements
Module	3	Part Modeling Enhancements
Module	4	Advanced Part Modeling Enhancements
Module	5	Resolving Failures

Day 2

Module	6	Assembly Enhancements
Module	7	Advanced Assembly Enhancements
Module	8	Drawing Enhancements
Module	9	Sheetmetal Enhancements

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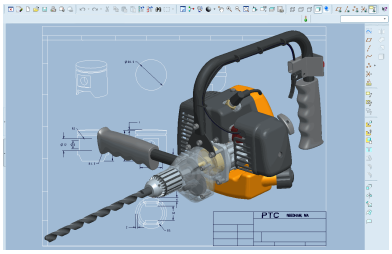
Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

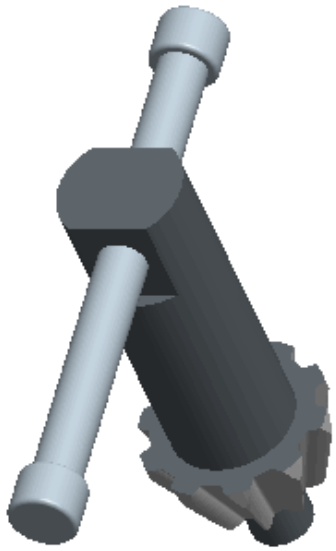
Overview

Course Code	TRN-2232–T
Course Length	5 Days



This course is designed for new users who want to become proficient with Pro/ENGINEER Wildfire 5.0 as quickly as possible. In this course, you will focus on learning core-modeling skills. Topics include sketching, part modeling, assemblies, drawings, and basic model management techniques. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts, assemblies, and drawings. At the end of each module, you will find a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole. After completing the course you will be well prepared to work effectively on product design projects using Pro/ENGINEER Wildfire.

Course Objectives



- Learning the basic Pro/ENGINEER Design Process
- Understanding Pro/ENGINEER concepts
- Learning how to use the Pro/ENGINEER interface
- Selecting and editing items
- Sketching geometry and using tools
- Creating sketches for features
- Creating datum planes and datum axes
- Creating extrudes, revolves, and ribs
- Utilizing internal sketches and embedded datums
- Creating sweeps and blends
- Creating holes, shells, and drafts
- Creating rounds and chamfers
- Grouping, copying, and mirroring items
- Creating patterns
- Measuring and inspecting models
- Assembling with constraints
- Assembling with connections
- Exploding assemblies
- Laying out drawings and creating views
- Creating drawing annotations
- Using layers
- Investigating parent/child relationships
- Capturing and managing design intent
- Resolving failures and seeking help
- Comprehensive two part Design Project

Prerequisites

- None

Audience

- This course is intended for product designers, drafters, industrial/conceptual designers, and routed systems designers. People in related roles will also benefit from taking this course.

Agenda

Day 1

Module	1	Introduction to the Pro/ENGINEER Wildfire Basic Modeling Process
Module	2	Understanding Pro/ENGINEER Concepts
Module	3	Using the Pro/ENGINEER Interface
Module	4	Selecting and Editing
Module	5	Creating Sketcher Geometry

Day 2

Module	6	Using Sketcher Tools
Module	7	Creating Sketches for Features
Module	8	Creating Datum Features: Planes and Axes
Module	9	Creating Extrudes, Revolves, and Ribs
Module	10	Utilizing Internal Sketches and Embedded Datums
Module	11	Creating Sweeps and Blends

Day 3

Module	12	Creating Holes, Shells, and Draft
Module	13	Creating Rounds and Chamfers
Module	14	Project I
Module	15	Group, Copy, and Mirror Tools
Module	16	Creating Patterns
Module	17	Measuring and Inspecting Models

Day 4

Module	18	Assembling with Constraints
Module	19	Assembling with Connections
Module	20	Exploding Assemblies
Module	21	Drawing Layout and Views
Module	22	Creating Drawing Annotations

Module 23 Using Layers

Day 5

Module 24 Investigating Parent/Child Relationships

Module 25 Capturing and Managing Design Intent

Module 26 Resolving Failures and Seeking Help

Module 27 Project II

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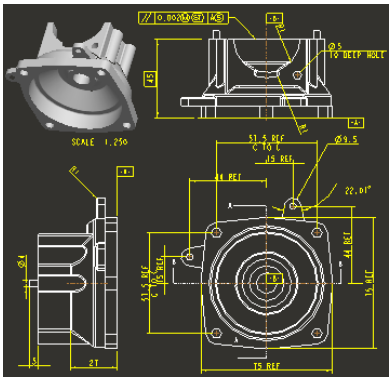
Detailing with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

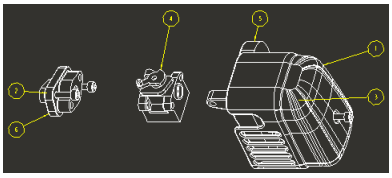
Course Code	TRN-2233–T
Course Length	3 Days



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.

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

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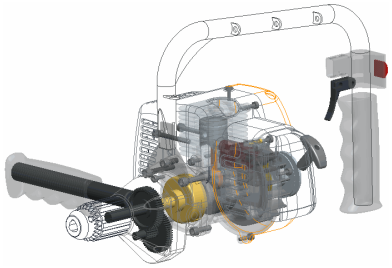
Advanced Assembly Design with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	TRN-2234-T
Course Length	3 Days

In this course, you will learn how to use Pro/ENGINEER Wildfire 5.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain design, increase your efficiency, and increase system performance when working with large assemblies. In addition, you will learn the basics of using and creating predefined assembly structures and skeletons, both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Using Advanced Assembly Constraints
- Creating and Using Component Interfaces
- Creating and Using Flexible Components
- Restructuring and Mirroring Assemblies
- Using Assembly Features and Shrinkwrap
- Replacing Components in an Assembly
- Understanding the Basics of Simplified Reps
- Creating Cross-Sections, Display Styles, Layer States, and Combined Views
- Substituting Components using User Defined, Envelopes, and Simplified Reps
- Understanding Advanced Simplified Rep Functionality
- Creating and Using Assembly Structure and Skeletons
- Project

Prerequisites

- Fast Track to Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers, mechanical designers, and related roles.

Agenda

Day 1

Module	1	Using Advanced Assembly Constraints
Module	2	Creating and Using Component Interfaces
Module	3	Creating and Using Flexible Components
Module	4	Restructuring and Mirroring Assemblies

Day 2

Module	5	Using Assembly Features and Shrinkwrap
Module	6	Replacing Components in an Assembly
Module	7	Understanding the Basics of Simplified Reps
Module	8	Creating Cross-Sections, Display Styles, Layer States, and Combined Views

Day 3

Module	9	Substituting Components using User Defined, Envelopes, and Simplified Reps
Module	10	Understanding Advanced Simplified Rep Functionality
Module	11	Creating and Using Assembly Structure and Skeletons
Module	12	Project

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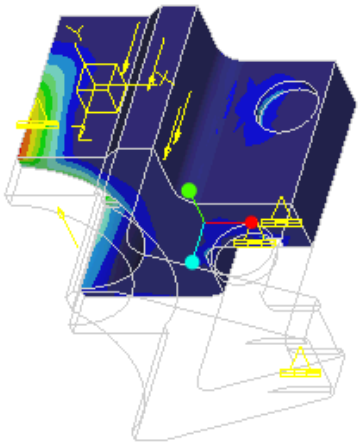
Pro/ENGINEER Mechanica Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

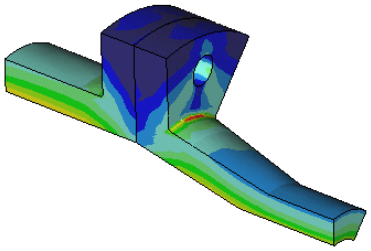
Course Code	TRN-2235-S
Course Length	5 Days



This course is designed for new users who want to test, validate, and optimize product designs with the Pro/ENGINEER Wildfire 5.0 Mechanica module. Mechanica enables you to simulate structural and thermal loads on product designs. In this course, you will complete comprehensive, hands-on lab exercises that simulate realistic analysis and design optimization activities. You will also learn about advanced topics such as dynamic analyses, combined mechanical and thermal analyses, and Fatigue Studies. A module on Mechanica Best Practices is also included to help users avoid some of the more common problems that new users encounter. After completing the course, you will be able to run engineering analyses and optimizations on your product design models. At the end of each module, you will find a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole. After completing the course you will be well prepared to complete Pro/MECHANICA analyses on product design projects in Pro/ENGINEER Wildfire 5.0.

Course Objectives

- Learning the basic Pro/MECHANICA Analysis Process
- Theory and Mechanica Model Topics
- Exploring Results
- Materials and Material Properties
- Understanding and Using Pro/MECHANICA idealizations
- Understanding and Using Structural Loads
- Understanding and Using Structural Constraints
- Running Structural Analyses
- Running Thermal Analyses
- Convergence
- Analyzing Assemblies with Pro/MECHANICA
- Completing Design and Sensitivity Studies
- Running Optimization Studies
- Advanced Topics
- Analysis Best Practices
- Analysis Projects



Prerequisites

- Three months of Pro/ENGINEER Wildfire 5.0 experience

Audience

- This course is intended for design engineers and mechanical designers. People in related roles will also benefit from taking this course.

Agenda

Day 1

Module	1	The Pro/ENGINEER Mechanica Process
Module	2	Theory and Mechanica Model Topics
Module	3	Results
Module	4	Materials and Material Properties

Day 2

Module	5	Idealizations
Module	6	Structural Loads
Module	7	Structural Constraints
Module	8	Structural Analysis I

Day 3

Module	9	Structural Analysis II
Module	10	Thermal Analysis
Module	11	Convergence
Module	12	Analyzing Assemblies I

Day 4

Module	13	Analyzing Assemblies II
Module	14	Design and Sensitivity Studies
Module	15	Optimization Studies
Module	16	Dynamic Analyses

Day 5

Module	17	Advanced Topics
Module	18	Analysis Best Practices
Module	19	Projects

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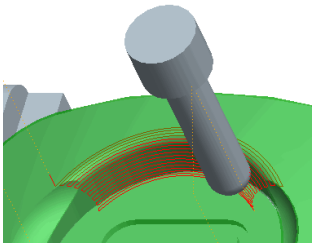
Milling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

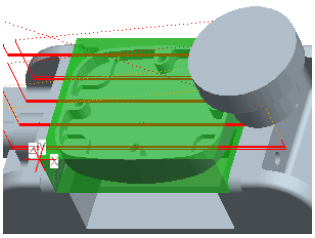
Overview

Course Code	TRN-2236–T
Course Length	5 Days



In this course, you will learn how to machine products using Pro/ENGINEER Wildfire 5.0 manufacturing tools. This course covers creating tool paths for 3 axis milling machines. During the course, you will learn how to complete each phase of the manufacturing process. You will start by creating manufacturing models and configuring the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will then learn how to create milling sequences and holemaking sequences, and post-process cutter location (CL) data to create machine code.After completing the course, you will be able to create numerical control (NC) programs for milling machines and post-process cutter location (CL) data to create machine specific code. 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



- Understanding the manufacturing process.
- Creating and configuring manufacturing models.
- Configuring the manufacturing environment.
- Creating and modifying milling sequences.
- Creating and modifying holemaking sequences.
- Using the process manager to create NC sequences.
- Post-processing cutter location (CL) data.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

- This course is intended for manufacturing engineers and NC machinists.

Agenda

Day 1

Module	1	Introduction to Manufacturing
Module	2	Creating Manufacturing Models
Module	3	Configuring Operations
Module	4	Using Reference Models
Module	5	Using Workpiece Models
Module	6	Creating and Using NC Model Assemblies
Module	7	Creating and Configuring Workcells

Day 2

Module	8	Creating and Configuring Tools
Module	9	Using Template Manufacturing Models
Module	10	Using Manufacturing Parameters
Module	11	Creating Face Milling Sequences

Day 3

Module	12	Creating Volume Milling Sequences
Module	13	Creating Profile Milling Sequences
Module	14	Creating Straight Cut Surface Milling Sequences
Module	15	Creating From Surface Isolines Surface Milling Sequences

Day 4

Module	16	Creating Cut Line Surface Milling Sequences
Module	17	Advanced Surface Milling Options
Module	18	Creating Roughing and Re-roughing Sequences
Module	19	Creating Finishing Sequences

Day 5

Module	20	Creating Trajectory Milling Sequences
Module	21	Creating Holemaking Sequences

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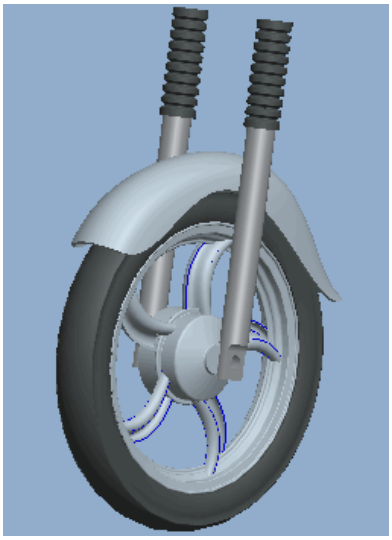
Freeform Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

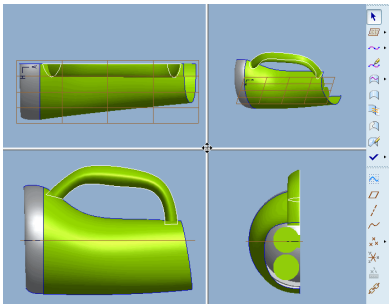
Course Code	TRN-2237–T
Course Length	2 Days



In Pro/ENGINEER Wildfire 5.0 you can create freeform surface models using the interactive surface design extension (ISDX) modeling environment, often called Freeform Surfacing or Style surfacing. The Style tool is a spline-based freeform modeler that enables you to combine the parametric feature-based modeling approach with the unconstrained freeform surface modeling approach. This gives you the flexibility to design complex-shaped products in a single modeling environment. In this course, you will learn how to use the Style tool to create and manipulate freeform curves, freeform surfaces, freeform surface details, and advanced freeform surface models. You will also learn how to integrate style features with other parametric features in design models. After completing this course, you will be well prepared to design complex-shaped freeform surface models in Pro/ENGINEER Wildfire 5.0. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Introduction to the Freeform Surface Modeling Process
- Understanding Freeform Surface Modeling Concepts
- Creating Initial Freeform Curves
- Developing Freeform Surface Models
- Advanced Tools and Techniques for Defining Freeform Shapes
- Creating Smooth Freeform Surface Models
- Integrating Style and Parametric Features
- Techniques for Creating Common Detailed Shapes
- Creating Complex, High Quality Freeform Models



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0.

Audience

- Design engineers, mechanical designers, who have a need to create styled surface geometry.

Agenda

Day 1

Module	1	Introduction to the Freeform Surface Modeling Process
Module	2	Understanding Freeform Surface Modeling Concepts
Module	3	Creating Initial Freeform Curves
Module	4	Developing Freeform Surface Models

Day 2

Module	5	Advanced Tools and Techniques for Defining Freeform Shapes
Module	6	Creating Smooth Freeform Surface Models
Module	7	Integrating Style and Parametric Features
Module	8	Techniques for Creating Common Detailed Shapes
Module	9	Creating Complex, High Quality Freeform Models

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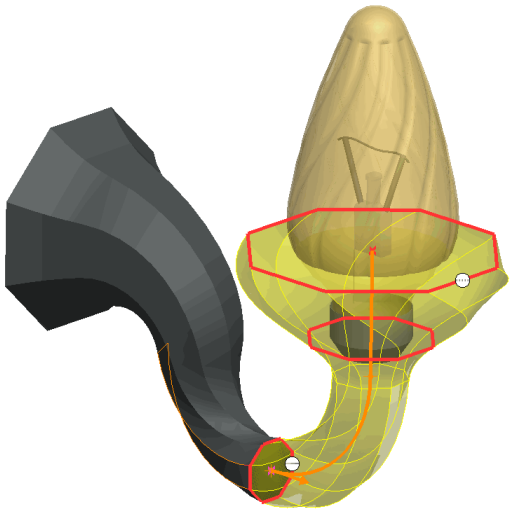
Advanced Modeling with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	TRN-2238–T
Course Length	3 Days



The Advanced Part Modeling with Pro/ENGINEER Wildfire 5.0 training course teaches you how to use advanced part modeling techniques in Pro/ENGINEER Wildfire 5.0 to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse existing design geometry when creating new design models. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment via your PTC University account, or your instructor will provide training on how to do this after the class. After completing this course, you will be well prepared to work efficiently with complex product designs using Pro/ENGINEER Wildfire 5.0.

Course Objectives

- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Create advanced holes
- Create advanced drafts and ribs
- Create advanced shells
- Create advanced rounds and chamfers
- Use relations and parameters
- Create advanced blends
- Create variable section sweeps
- Create helical sweeps
- Create swept blends
- Learn advanced layer techniques
- Learn how to use different advanced reference management techniques
- Create family tables
- Reuse features
- Learn advanced copy techniques
- Create advanced patterns

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Pro/ENGINEER Wildfire 5.0 Update from Pro/ENGINEER Wildfire 4.0.

Audience

- This course is intended for mechanical designers, design engineers and related roles. The topics in this course are also available as Web-based training courses.

Agenda

Day 1

Module	1	Advanced Selection
Module	2	Advanced Datum Features
Module	3	Advanced Sketching
Module	4	Advanced Hole Creation
Module	5	Advanced Drafts and Ribs
Module	6	Advanced Shells
Module	7	Advanced Rounds and Chamfers

Day 2

Module	8	Relations and Parameters
Module	9	Advanced Blends
Module	10	Variable Section Sweeps
Module	11	Helical Sweeps
Module	12	Swept Blends

Day 3

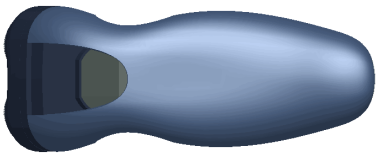
Module	13	Advanced Layers
Module	14	Advanced Reference Management
Module	15	Family Tables
Module	16	Reusing Features
Module	17	Advanced Copy
Module	18	Advanced Patterns

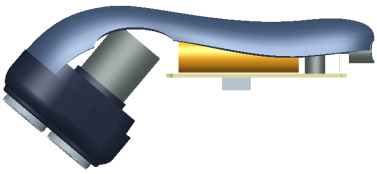
PTC university

Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

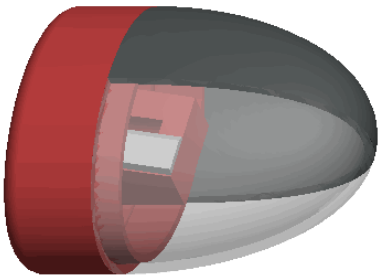
Course Code	TRN-2239–T
Course Length	3 Days





In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how manipulate surfaces using editing tools, and analyze surfaces for quality and desired characteristics. In addition, you will learn how to create solid features using the surfaces as references. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment by using your PTC University account, or your instructor will provide training on how to do this after the class. After completing this course, you will be well prepared to create complex shaped models using surfaces in Pro/ENGINEER Wildfire 5.0.

Course Objectives



- Describe surface modeling and its terminology
- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Learn basic surfacing tools
- Create various boundary surfaces
- Create variable section sweep surfaces
- Create helical sweep surfaces
- Create swept blend surfaces
- Utilize surface analysis tools
- Extend and trim surfaces
- Manipulate surfaces
- Create and edit solid models using surface quilts
- Utilize the master model technique

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Pro/ENGINEER Wildfire 5.0 Update from Pro/ENGINEER Wildfire 4.0.

Audience

- This course is intended for mechanical designers, design engineers, industrial designers, and related roles. The topics in this course are also available as Web-based training courses.

Agenda

Day 1

Module	1	Surface Modeling Overview
Module	2	Advanced Selection
Module	3	Advanced Datum Features
Module	4	Advanced Sketching
Module	5	Basic Surfacing Tools
Module	6	Boundary Blend Surfaces

Day 2

Module	7	Additional Boundary Surfaces
Module	8	Variable Section Sweeps
Module	9	Helical Sweeps
Module	10	Swept Blends
Module	11	Analyzing Surface Curvature
Module	12	Additional Surface Analysis Tools

Day 3

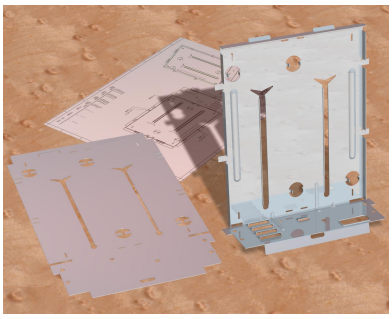
Module	13	Extending and Trimming Surfaces
Module	14	Manipulating Surfaces
Module	15	Creating and Editing Solids using Quilts
Module	16	Master Model Technique
Module	17	Project

PTC university

Sheetmetal Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

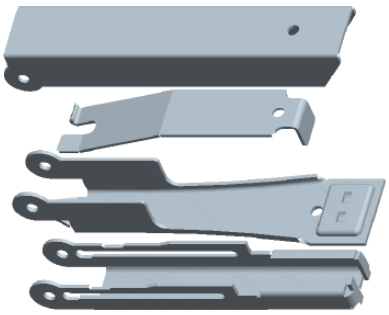
Overview

Course Code	TRN-2240-T
Course Length	2 days



Sheetmetal Design using Pro/ENGINEER Wildfire 5.0 is a comprehensive training course that teaches you how to create sheetmetal parts in Pro/ENGINEER. The course builds upon the basic lessons you learned in Introduction to Pro/ENGINEER Wildfire 5.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including sheetmetal production drawings. All the functions needed to create sheetmetal parts, drawings, and assemblies are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model, and document both in production drawings. At the end of each day, you use the Pro/FICIENCY skills assessments to reinforce your understanding of the course topics.

Course Objectives



- The Sheetmetal Design Process
- Sheetmetal Model Creation, Conversion, and Display
- Methods of Developed Length Calculation
- Primary Wall Features
- Secondary Wall Features
- Partial Walls
- Bend Relief
- Unbend and Bend Back Features
- Sheetmetal Bend Features
- Flat Patterns
- Sheetmetal Cuts
- Forms
- Notch and Punch Features
- Sheetmetal Environment Setup
- Sheetmetal Design Information Tools
- Sheetmetal Design Rules
- Detailing Sheetmetal Designs
- Sheetmetal Design Project

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers. People in related roles can also benefit from taking this course.

Agenda

Day 1

Module	1	Introduction to the Pro/ENGINEER Wildfire Sheetmetal Design Process
Module	2	Sheetmetal Model Fundamentals
Module	3	Creating Primary Sheetmetal Wall Features
Module	4	Creating Sheetmetal Secondary Wall Features

Day 2

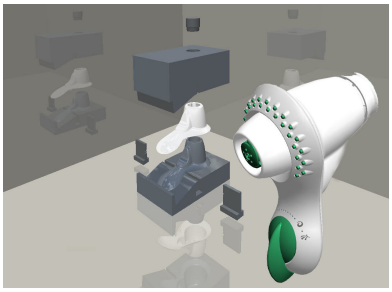
Module	5	Modifying Sheetmetal Models
Module	6	Sheetmetal Bends and Setting Up the Sheetmetal Environment
Module	7	Special Sheetmetal Tools
Module	8	Detailing Sheetmetal Designs
Module	9	Design Project

PTC university

Mold Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

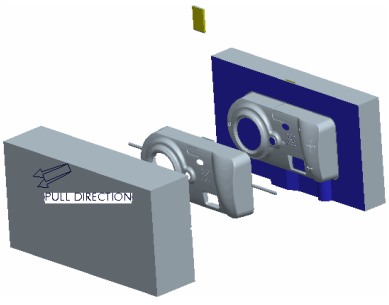
Course Code	TRN-2241–T
Course Length	2 days



Pro/MOLDESIGN provides the tools to create a mold model from start to finish by using the mold design process within Pro/ENGINEER Wildfire 5.0. In this course, you will learn how to create, modify, and analyze mold components and assemblies. Any changes made to the design model automatically propagate to the mold components and assemblies. You will learn how to create final extract components that reflect the geometry of the design model, along with shrinkage considerations, adequate drafting, mold features, and cooling systems. After completing the course, you will have a better understanding of the mold design process and how to create molded products by using the mold design process. 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

- Learn the basic mold process
- Prepare design models for the mold process
- Analyze design models to ensure their readiness for molding
- Create mold models
- Apply shrinkage to the reference model
- Create and assemble workpieces into the mold model
- Create mold volumes
- Create parting lines and parting surfaces
- Split mold volumes
- Extract mold components
- Create mold features
- Learn how to fill and open the mold



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Basic understanding of industry standard Mold design terminology and processes.
- Knowledge of Pro/ENGINEER surfacing techniques a plus.

Audience

- This course is intended for designers, machinists, and manufacturing engineers. The topics in this course are also available as a Web-based training course.

Agenda

Day 1

Module	1	Introduction to the Pro/ENGINEER Basic Mold Process
Module	2	Design Model Preparation
Module	3	Design Model Analysis
Module	4	Mold Models
Module	5	Shrinkage
Module	6	Workpieces
Module	7	Mold Volume Creation

Day 2

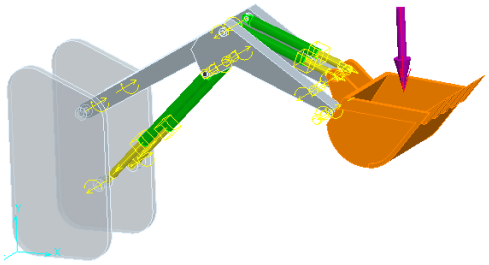
Module	8	Parting Line and Parting Surface Creation
Module	9	Splitting Mold Volumes
Module	10	Mold Component Extraction
Module	11	Mold Features Creation
Module	12	Filling and Opening the Mold

PTC university

Mechanism Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

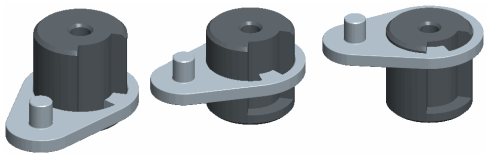
Course Code	TRN-2242
Course Length	1 Day



Mechanism Design using Pro/ENGINEER Wildfire 5.0 is designed for experienced users who want to add motion to their models by creating mechanism connections and servo motors. In Pro/ENGINEER Wildfire 5.0 you can add motion to your models using the standard mechanism functionality, often referred to as the Mechanism Design Extension (MDX). In this course, you will learn about creating mechanism connections, configuring the mechanism model, creating a kinematic analysis, and evaluating results. These topics will enable you to simulate the range of motion between components in your moving assemblies, create gear connections that simulate the gear ratios, create Cam connections that enable Pro/ENGINEER parts to "push" other parts they come into contact with, and check for collisions between moving components. After completing this course, you will be prepared to work on mechanism designs using Pro/ENGINEER Wildfire Mechanism Design. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment via your PTC University account, or your instructor will provide training on how to do this after the class.

Course Objectives

- Introduction to the mechanism design process
- Creating mechanism connections
- Configuring motion and analysis
- Evaluating analysis results



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers and mechanical designers who need to add and evaluate the motion of their assemblies.

Agenda

Day 1

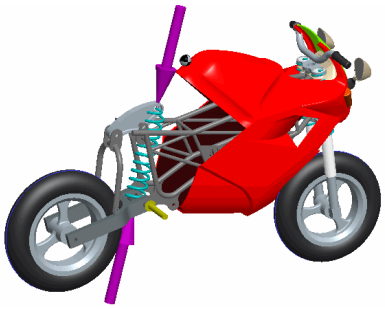
Module	1	Introduction to the Mechanism Design Process
Module	2	Creating Mechanism Connections
Module	3	Configuring Motion and Analysis
Module	4	Evaluating Analysis Results

PTC university

Mechanism Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

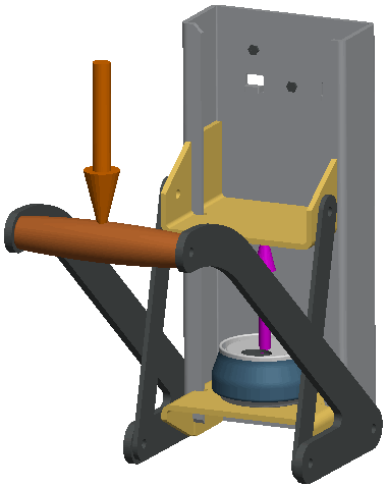
Course Code	TRN-2243–T
Course Length	1 Day



This course is designed for experienced users who want to add motion to their products and analyze dynamic reactions of moving components. In this course, you will focus on learning advanced modeling and analysis skills in this comprehensive, hands-on course. Topics include developing the 3-D model, analyzing the mechanism model, and evaluating results. These topics will enable you to measure dynamic reactions of components, measure the force required to keep a mechanism balanced, and determine the resting state of a mechanism. After completing this course, you will be prepared to work on mechanism designs using Pro/ENGINEER Wildfire Mechanism Dynamics Option (MDO). 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

- Understanding the mechanism dynamics option
- Applying force motors, springs, and dampers to assemblies
- Applying forces, torques, and gravity to assemblies
- Creating dynamic analyses
- Creating force balance analyses
- Creating static analyses
- Measuring forces, velocities, accelerations, and other reactions
- Evaluating results



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Mechanism Design using Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers and mechanical designers who need to add and evaluate the motion of their assemblies.

Agenda

Day 1

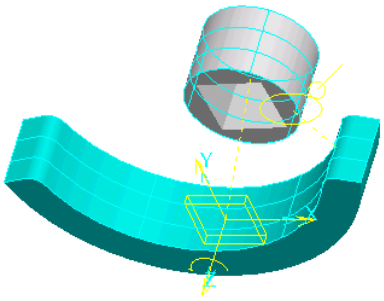
Module	1	Introduction to the Mechanism Design Process
Module	2	Adding Dynamic Entities to a Mechanism
Module	3	Analyzing the Mechanism Model
Module	4	Evaluating Analysis Results
Module	5	Project

PTC university

Behavioral Modeling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

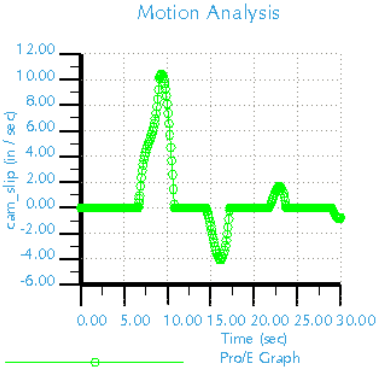
Course Code	TRN-2244-T
Course Length	1 Day



In this course, you will focus on learning advanced analysis skills unrelated to structural or thermal analysis. Also, you will learn how to analyze your models and create analysis features that can enforce your design intent. You will also learn how to create sensitivity and feasibility studies that aid you in determining how to reach your design goals. Furthermore, you will learn how to create optimization design studies that enable you to configure the dimensions and parameters that Pro/ENGINEER can change in order to meet your design specifications. This course is designed for experienced users who want to add additional features that enable you to meet or exceed the design specifications of your products. At the end of each module, you will find a set of review questions to reinforce critical topics from that module. After completing this course, you will be prepared to work on critical component designs using Pro/ENGINEER Wildfire Behavioral Modeling. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Applying the Behavioral Modeling process and concepts to your designs
- Creating measurement analysis features
- Creating relation, motion, Mechanica, and MS Excel analysis features
- Creating user-defined analysis features
- Conducting sensitivity analyses
- Conducting feasibility and optimization studies
- Design Project



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0 or equivalent experience.
- Experience with MS Excel, Mechanica, Mechanism Design, and Mechanism Dynamics is useful but not required.

Audience

- This course is intended for product designers and engineers. Related roles will also benefit from taking this course.

Agenda

Day 1

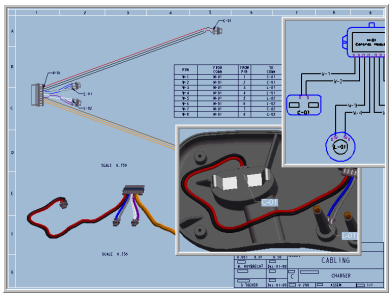
Module	1	Introduction to the Behavioral Modeling Process
Module	2	Creating Measurement Features on Pro/ENGINEER Models
Module	3	Creating Model Property Features on Pro/ENGINEER Models
Module	4	Creating Analysis Features on Pro/ENGINEER Models
Module	5	Creating User-Defined Analysis Features on Pro/ENGINEER Models
Module	6	Conducting Design Studies and Optimizing Models
Module	7	Project

PTC university

Cabling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	TRN-2245–T
Course Length	3 Days



In this course, you will learn how to create 3-D electrical harnesses using Pro/ENGINEER Wildfire 5.0. This includes using Routed Systems Designer 9.0 to pass schematic diagram information into the 3-D harness designs created within Pro/ENGINEER Wildfire 5.0.

You will learn how to route electrical harnesses both with and without schematic diagram information, create flattened harnesses for manufacturing, and document harness designs by creating flattened harness drawings that include customized BOM tables and wire list information.

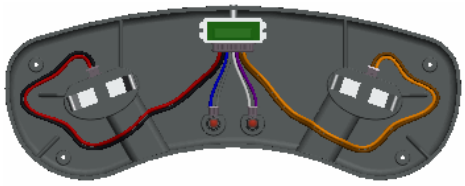
A significant portion of the course is devoted to a cabling design project, where students create a full wiring harness with minimal 'picks and clicks' to solidify techniques learned previously in the course.

At the end of each module, you will complete on-line learning assessments that reinforce your understanding of the course topics. There is also a wrap-up assessment at the end of the course.

After successfully completing the course, you will be able to create 3-D electrical harnesses and associated manufacturing deliverables using Pro/ENGINEER Wildfire 5.0.

Optionally, students may wish to attend the Routed Systems Designer 9.0 for Pro/ENGINEER Wildfire 5.0 course. This will enable a full understanding of the schematic design process used to provide schematic data for the creation of electrical harness assemblies in Pro/ENGINEER Wildfire 5.0.

Course Objectives



- Learning the basic Pro/ENGINEER Cabling Process
- Creating harness assembly structures
- Setting up for cabling
- Routing wires and cables
- Modifying wire routings
- Routing and utilizing networks
- Establishing logical references
- Routing wires and cables using logical data
- Creating harness components and cosmetics
- Creating flat harnesses
- Documenting harness designs
- Comprehensive Design Project

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0.
- Routed Systems Designer 9.0 for Pro/ENGINEER Wildfire 5.0 (Optional).

Audience

- This course is intended for engineers involved in the 3-D routing and documenting of electrical wiring and cabling harnesses. The topics in this course are also available as Web-based training courses.

Agenda

Day 1

Module	1	Introduction to the Pro/ENGINEER Wildfire Basic Cabling Process
Module	2	Creating Harness Assembly Structures
Module	3	Setting Up for Cabling
Module	4	Routing Wires and Cables
Module	5	Modifying Wire Routings

Day 2

Module	6	Routing and Utilizing Networks
Module	7	Establishing Logical References
Module	8	Routing Wires and Cables using Logical Data
Module	9	Creating Harness Components and Cosmetics
Module	10	Creating Flat Harnesses
Module	11	Documenting Harness Designs

Day 3

Module	12 A	Project (RSD-Based)
Module	12 B	Project (Manual Routing)

PTC university

Piping using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	TRN-2246-T
Course Length	3 Days

In this course, you will learn how to manually create (non-specification driven) mechanical piping designs using Pro/ENGINEER Wildfire. This includes learning how to configure pipelines, how to route pipelines, and how to insert pipe fittings such as valves and reducers.

You will also learn how to create specification driven industrial piping designs using Pro/ENGINEER Wildfire. This includes learning how to use schematic diagrams created with Routed Systems Designer to drive 3-D industrial piping designs created within Pro/ENGINEER Wildfire.

You will learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information. You will also learn how to export ISOGEN format files for creating pipeline, spool and systems isometric drawings.

Course Objectives

- Understand the manual piping design process.
- Understand the specification-driven piping design process.
- Create piping assembly structures.
- Configure and route pipelines.
- Move and modify pipelines.
- Create pipe solids and fabricate pipes.
- Configure and insert fittings.
- Create piping report information.
- Create piping drawings.
- Configure a piping specification database.
- Configure project specific data files.
- Create specification-driven pipelines.
- Create schematic driven pipelines.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- This course is intended for engineers, involved in the 3-D routing of mechanical piping systems and industrial piping systems.

Agenda

Day 1

Module	1	Introduction to Piping
Module	2	Creating Piping Assembly Structures
Module	3	Configuring and Routing Pipelines
Module	4	Moving and Modifying Pipelines
Module	5	Configuring and Inserting Fittings

Day 2

Module	6	Creating Solid Pipeline Models
Module	7	Gathering Piping Information
Module	8	Creating Piping Drawings
Module	9	Specification Database Overview
Module	10	Setting Up Specification Databases: Piping
Module	11	Setting Up Specification Databases: Fittings

Day 3

Module	12	General Master Catalog Files
Module	13	Configuring Project Specific Data Files
Module	14	Specification-Driven Routing and Inserting Fittings
Module	15	Using RSD Process and Instrumentation Diagrams Data
Module	16	Schematic Driven Pipeline Modeling
Module	17	Using ISOGEN PCF Data

PTC university

Routed Systems Designer 9.0 for Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	TRN-2247-T
Course Length	3 Days

In this course, you will learn how to use Routed Systems Designer (RSD) to create schematic diagrams for both electrical harness and piping designs. You will learn how to administer the RSD working environment, and how to configure the catalog library. You will learn how to create functional block diagrams and block interconnect diagrams. You will learn how to create electrical circuit diagrams, and wiring diagrams, including wire interconnect diagrams. You will also learn how to create process and instrumentation design (P&ID) diagrams for piping systems. Finally, you will learn how to use wiring diagrams and P&ID diagrams to configure 3-D harness designs and industrial piping designs created within Pro/ENGINEER Wildfire. At the end of each module, you will complete Pro/FICIENCY skills assessment questions. These questions are used to help reinforce your understanding of the course topics and form the basis for review sessions.

Course Objectives

- Understanding the 2-D schematic design processes for electrical harness designs and piping designs.
- Configuring the RSD working environment.
- Creating and configuring the catalog library.
- Creating functional block diagrams, (including block interconnect diagrams).
- Creating circuit diagrams.
- Creating wiring diagrams, (including wire interconnect diagrams).
- Creating process and instrumentation design diagrams.
- Communicating diagram information to Pro/ENGINEER Wildfire.

Prerequisites

- None

Audience

- This course is intended for engineers, involved in the schematic 2-D layout of either electrical diagrams or piping diagrams.

Agenda

Day 1

Module	1	Introduction to Routed Systems Designer
Module	2	Configuring the Working Environment
Module	3	Configuring Designs and Diagram Sheets
Module	4	Configuring Catalog Properties
Module	5	Creating Design Templates and Template Sheets
Module	6	Using RSD Tools

Day 2

Module	7	Creating Catalog Artifacts
Module	8	Creating and Configuring a Central Catalog
Module	9	Reviewing Designs
Module	10	Instancing and Manipulating Objects
Module	11	Creating Block Diagrams

Day 3

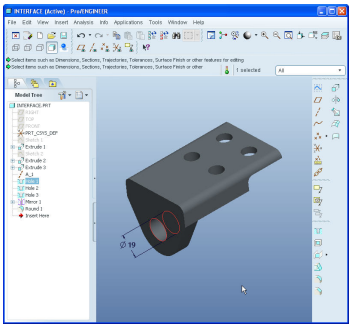
Module	12	Creating Circuit Diagrams
Module	13	Creating Wiring Diagrams
Module	14	Creating Interconnect Diagrams
Module	15	Creating P&ID Diagrams
Module	16	Using Schematic Design Data in Pro/ENGINEER Harness and Piping Designs
Module	17	Using Pro/Diagram Information

PTC university

Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 2.0

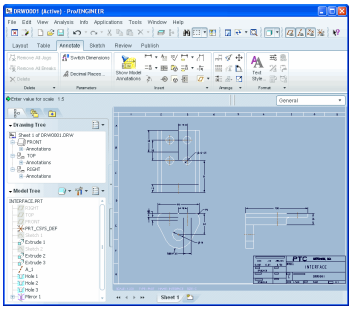
Overview

Course Code	TRN-2248-T
Course Length	3 Days



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0, when upgrading from Pro/ENGINEER Wildfire 2.0. First, changes to the interface will be explored, such as the new skin and screen colors. Next, you will become familiar with the enhancements to Sketcher geometry and tools, such as sketching parallelograms and chamfers, as well as using the sketcher diagnostic tools. Next, you will explore enhancements to Part mode, including creating trajectory ribs, pattern and UDF enhancements, the all-new resolve mode, and auto-round. You will also learn about the Assembly dashboard interface and other assembly enhancements such as the accessory window, the new reference viewer, improved simplified rep functionality, and mechanism enhancements to gears and belts. Next, you will examine the new ribbon interface and drawing tree for drawing mode, as well as numerous detailing enhancements. Finally, you will review Sheetmetal enhancements for flat and flange walls, in addition to patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Geometry Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Tools Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Feature Duplication Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Part Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Assembly Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Sheet and View Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Detailing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 2.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 2.0.

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Sketcher Geometry Enhancements
Module	3	Sketcher Tools Enhancements
Module	4	Part Modeling Enhancements

Day 2

Module	5	Feature Duplication Enhancements
Module	6	Advanced Part Modeling Enhancements
Module	7	Assembly Enhancements
Module	8	Advanced Assembly Enhancements

Day 3

Module	9	Resolving Failures
Module	10	Drawing Sheet and View Enhancements
Module	11	Drawing Detail Enhancements
Module	12	Sheetmetal Enhancements

Web Based Curriculum Guide

- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 4.0
- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 3.0
- Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) - Fundamentals
- Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) - Productivity Tools
- Detailing with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Advanced Assembly Design with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Pro/ENGINEER Mechanica Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Milling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Freeform Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

- Advanced Modeling with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Sheetmetal Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Mold Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Mechanism Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Mechanism Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Behavioral Modeling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Cabling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Piping using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)
- Routed Systems Designer 9.0 for Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

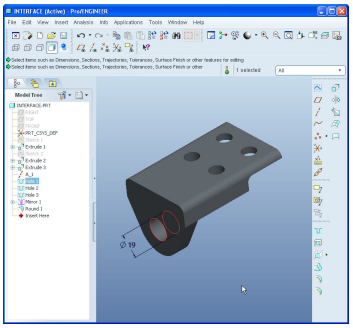
- Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 2.0
- Advanced Turning and Multi-task Machining using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

PTC university

Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 4.0

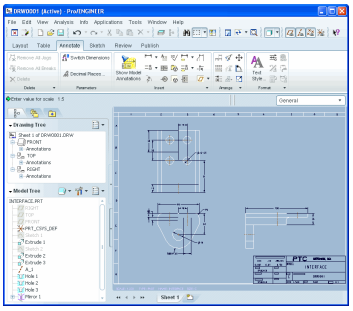
Overview

Course Code	SAB-CEK3004
Course Length	8 Hours



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0. First, you will become familiar with the enhancements to Sketcher, such as sketching datum features, parallelograms, and chamfers. Next, you will explore enhancements to Part mode, including creating trajectory ribs, point patterns, and the all-new resolve mode. You will also learn about new and enhanced Assembly capabilities such as the accessory window, the new explode interface, and enhancements to component repeat and restructure. Finally, you will examine the new ribbon interface for 2–D drawings, and review Sheetmetal enhancements such as patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 4.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 4.0.

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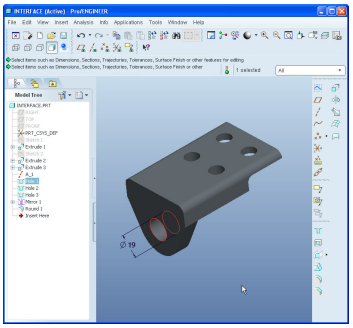
PTC university

Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 3.0

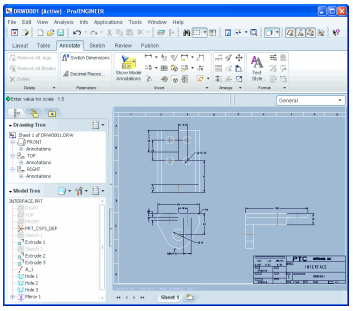
Overview

Course Code	SAB-CEK3005
Course Length	16 Hours



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0, when upgrading from Pro/ENGINEER Wildfire 3.0. First, changes to the interface will be explored, such as the new skin and screen colors. Next, you will become familiar with the enhancements to Sketcher geometry and tools, such as sketching parallelograms and chamfers, as well as using the sketcher diagnostic tools. Next, you will explore enhancements to Part mode, including creating trajectory ribs, pattern and UDF enhancements, the all-new resolve mode, and auto-round. You will also learn about Assembly enhancements such as the accessory window, the new reference viewer, improved simplified rep functionality, and mechanism enhancements to gears and belts. Next, you will examine the new ribbon interface and drawing tree for drawing mode, as well as numerous detailing enhancements. Finally, you will review Sheetmetal enhancements for flat and flange walls, in addition to patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Part Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Assembly Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 3.0.

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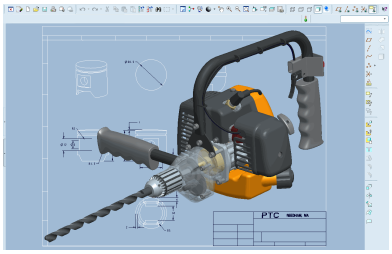
PTC university

Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) - Fundamentals

Overview

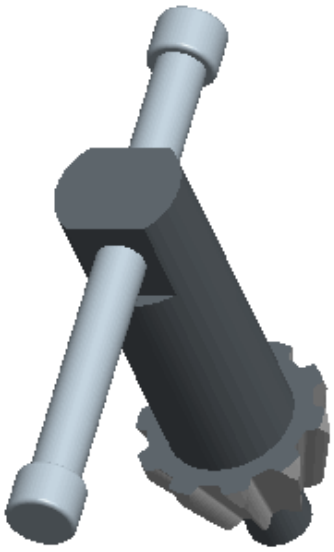
Course Code	SAB-CEK3006
Course Length	20 Hours



This course is designed for new users who want to become proficient with Pro/ENGINEER Wildfire 5.0 as quickly as possible. You will focus on learning core-modeling skills in this comprehensive, hands-on course. Topics include understanding the interface and basic Pro/ENGINEER concepts, selecting and editing, sketching and sketcher tools, and basic feature creation. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing the course you will be well prepared to work effectively on product design projects using Pro/ENGINEER Wildfire.

Course Objectives

- Learning the basic Pro/ENGINEER Design Process
- Understanding Pro/ENGINEER concepts
- Learning how to use the Pro/ENGINEER interface
- Selecting and editing items
- Sketching geometry and using tools
- Creating sketches for features
- Creating datum planes and datum axes
- Creating extrudes, revolves, and ribs
- Utilizing internal sketches and embedded datums
- Creating sweeps and blends
- Creating holes, shells, and drafts
- Creating rounds and chamfers
- Comprehensive Design Project



Prerequisites

- None

Audience

- This course is intended for product designers, drafters, industrial/conceptual designers, and routed systems designers. People in related roles will also benefit from taking this course.

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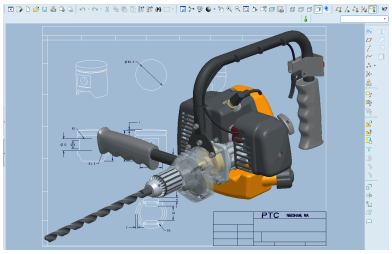
PTC university

Introduction to Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) - Productivity Tools

Overview

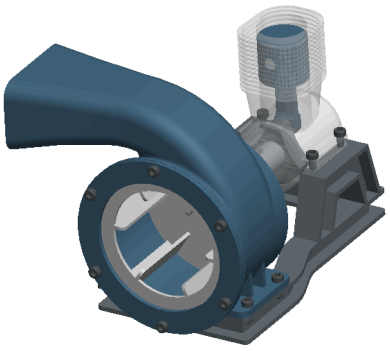
Course Code	SAB-CEK3007
Course Length	20 Hours



This course is designed for existing Pro/ENGINEER Wildfire 5.0 users who want to increase their productivity. You will focus on learning advanced modeling skills in this comprehensive, hands-on course. Topics include patterning; measuring and inspecting models; group, copy, and mirror tools; assembly creation; explode states and explode line creation; drawing creation; using layers; investigating parent/child relationships; capturing design intent; and resolving failures and seeking help. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts, assemblies, and drawings. After completing the course you will be well prepared to work effectively on product design projects using Pro/ENGINEER Wildfire.

Course Objectives

- Grouping, copying, and mirroring items
- Creating patterns
- Measuring and inspecting models
- Assembling with constraints
- Assembling with connections
- Exploding assemblies
- Laying out drawings and creating views
- Creating drawing annotations
- Using layers
- Investigating parent/child relationships
- Capturing and managing design intent
- Resolving failures and seeking help
- Comprehensive Design Project



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0 - Productivity Tools

Audience

- This course is intended for product designers, drafters, industrial/conceptual designers, and routed systems designers. People in related roles will also benefit from taking this course.

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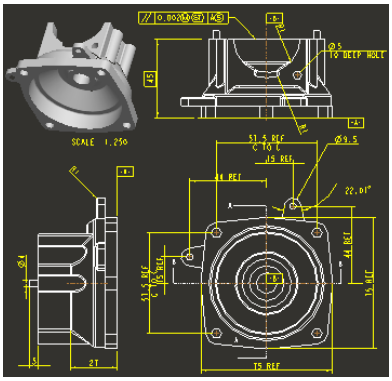
PTC university

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

Overview

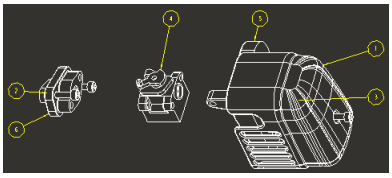
Course Code	WBT-2233-0
Course Length	24 Hours



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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PTC university

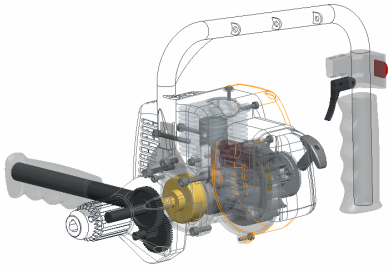
Advanced Assembly Design with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	WBT-2234-0
Course Length	24 Hours

In this course, you will learn how to use Pro/ENGINEER Wildfire 5.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain design, increase your efficiency, and increase system performance when working with large assemblies. In addition, you will learn the basics of using and creating predefined assembly structures and skeletons, both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model.At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Using Advanced Assembly Constraints
- Creating and Using Component Interfaces
- Creating and Using Flexible Components
- Restructuring and Mirroring Assemblies
- Using Assembly Features and Shrinkwrap
- Replacing Components in an Assembly
- Understanding the Basics of Simplified Reps
- Creating Cross-Sections, Display Styles, Layer States, and Combined Views
- Substituting Components using User Defined, Envelopes, and Simplified Reps
- Understanding Advanced Simplified Rep Functionality
- Creating and Using Assembly Structure and Skeletons
- Project

Prerequisites

- Fast Track to Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers, mechanical designers, and related roles.

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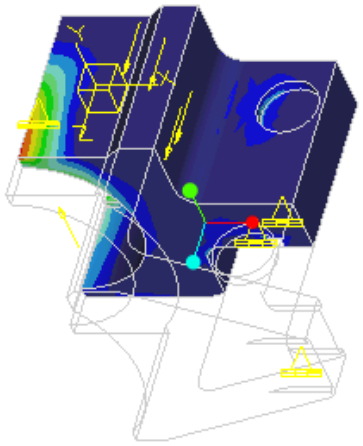
PTC university

Pro/ENGINEER Mechanica Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

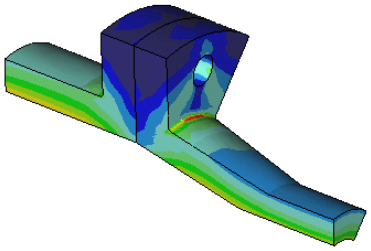
Course Code	SAB-CEK3010
Course Length	40 Hours



This course is designed for new users who want to test, validate, and optimize product designs with the Pro/ENGINEER Wildfire 5.0 Mechanica module. Mechanica enables you to simulate structural and thermal loads on product designs. In this course, you will complete comprehensive, hands-on lab exercises that simulate realistic analysis and design optimization activities. You will also learn about advanced topics such as dynamic analyses, combined mechanical and thermal analyses, and Fatigue Studies. A module on Mechanica Best Practices is also included to help users avoid some of the more common problems that new users encounter. After completing the course, you will be able to run engineering analyses and optimizations on your product design models. At the end of each module, you will find a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole. After completing the course you will be well prepared to complete Pro/MECHANICA analyses on product design projects in Pro/ENGINEER Wildfire 5.0.

Course Objectives

- Learning the basic Pro/MECHANICA Analysis Process
- Theory and Mechanica Model Topics
- Exploring Results
- Materials and Material Properties
- Understanding and Using Pro/MECHANICA idealizations
- Understanding and Using Structural Loads
- Understanding and Using Structural Constraints
- Running Structural Analyses
- Running Thermal Analyses
- Convergence
- Analyzing Assemblies with Pro/MECHANICA
- Completing Design and Sensitivity Studies
- Running Optimization Studies
- Advanced Topics
- Analysis Best Practices
- Analysis Projects



Prerequisites

- Three months of Pro/ENGINEER Wildfire 5.0 experience

Audience

- This course is intended for design engineers and mechanical designers. People in related roles will also benefit from taking this course.

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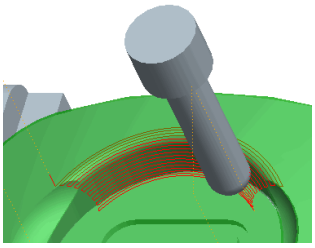
PTC university

Milling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

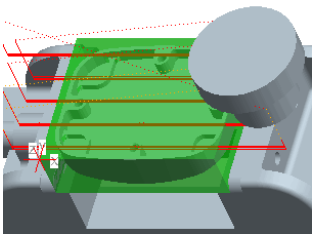
Overview

Course Code	SAB-CEK3011
Course Length	40 Hours



In this course, you will learn how to machine products using Pro/ENGINEER Wildfire 5.0 manufacturing tools. This course covers creating tool paths for 3 axis milling machines. During the course, you will learn how to complete each phase of the manufacturing process. You will start by creating manufacturing models and configuring the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will then learn how to create milling sequences and holemaking sequences, and post-process cutter location (CL) data to create machine code.After completing the course, you will be able to create numerical control (NC) programs for milling machines and post-process cutter location (CL) data to create machine specific code. 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



- Understanding the manufacturing process.
- Creating and configuring manufacturing models.
- Configuring the manufacturing environment.
- Creating and modifying milling sequences.
- Creating and modifying holemaking sequences.
- Using the process manager to create NC sequences.
- Post-processing cutter location (CL) data.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

- This course is intended for manufacturing engineers and NC machinists.

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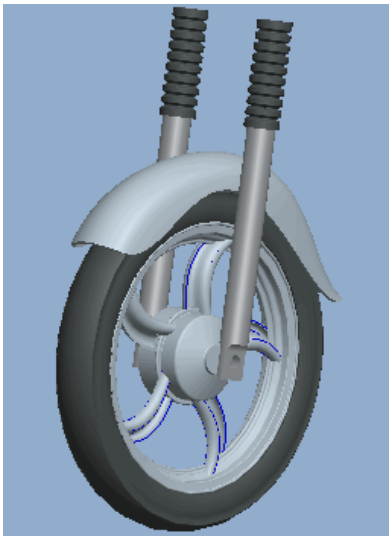
PTC university

Freeform Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

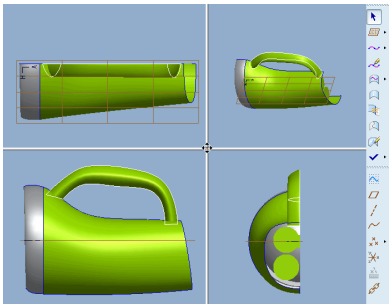
Course Code	WBT-2237-0
Course Length	16 Hours



In Pro/ENGINEER Wildfire 5.0 you can create freeform surface models using the interactive surface design extension (ISDX) modeling environment, often called Freeform Surfacing or Style surfacing. The Style tool is a spline-based freeform modeler that enables you to combine the parametric feature-based modeling approach with the unconstrained freeform surface modeling approach. This gives you the flexibility to design complex-shaped products in a single modeling environment. In this course, you will learn how to use the Style tool to create and manipulate freeform curves, freeform surfaces, freeform surface details, and advanced freeform surface models. You will also learn how to integrate style features with other parametric features in design models. After completing this course, you will be well prepared to design complex-shaped freeform surface models in Pro/ENGINEER Wildfire 5.0. At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. Your instructor will discuss these with the class. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Introduction to the Freeform Surface Modeling Process
- Understanding Freeform Surface Modeling Concepts
- Creating Initial Freeform Curves
- Developing Freeform Surface Models
- Advanced Tools and Techniques for Defining Freeform Shapes
- Creating Smooth Freeform Surface Models
- Integrating Style and Parametric Features
- Techniques for Creating Common Detailed Shapes
- Creating Complex, High Quality Freeform Models



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0.

Audience

- Design engineers, mechanical designers, who have a need to create styled surface geometry.

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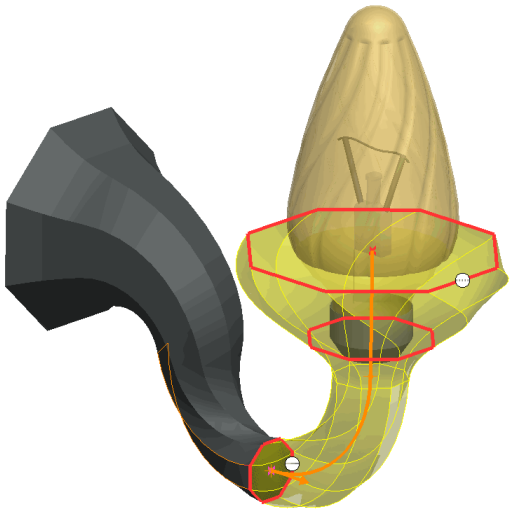
PTC university

Advanced Modeling with Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	WBT-2238-0
Course Length	24 Hours



The Advanced Part Modeling with Pro/ENGINEER Wildfire 5.0 training course teaches you how to use advanced part modeling techniques in Pro/ENGINEER Wildfire 5.0 to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse existing design geometry when creating new design models. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment via your PTC University account, or your instructor will provide training on how to do this after the class. After completing this course, you will be well prepared to work efficiently with complex product designs using Pro/ENGINEER Wildfire 5.0.

Course Objectives

- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Create advanced holes
- Create advanced drafts and ribs
- Create advanced shells
- Create advanced rounds and chamfers
- Use relations and parameters
- Create advanced blends
- Create variable section sweeps
- Create helical sweeps
- Create swept blends
- Learn advanced layer techniques
- Learn how to use different advanced reference management techniques
- Create family tables
- Reuse features
- Learn advanced copy techniques
- Create advanced patterns

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Pro/ENGINEER Wildfire 5.0 Update from Pro/ENGINEER Wildfire 4.0.

Audience

- This course is intended for mechanical designers, design engineers and related roles. The topics in this course are also available as Web-based training courses.

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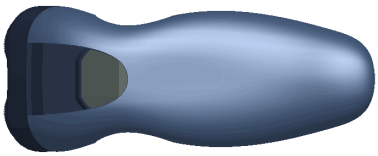
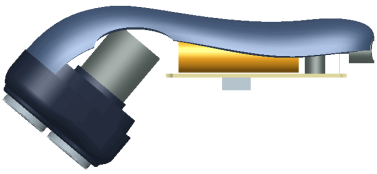
PTC university

Surfacing using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

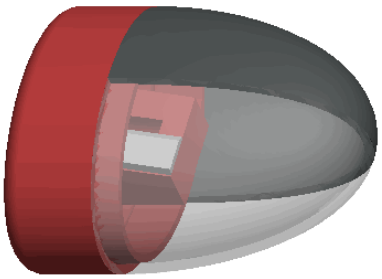
Course Code	WBT-2239-0
Course Length	24 Hours





In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how manipulate surfaces using editing tools, and analyze surfaces for quality and desired characteristics. In addition, you will learn how to create solid features using the surfaces as references. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment by using your PTC University account, or your instructor will provide training on how to do this after the class. After completing this course, you will be well prepared to create complex shaped models using surfaces in Pro/ENGINEER Wildfire 5.0.

Course Objectives



- Describe surface modeling and its terminology
- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Learn basic surfacing tools
- Create various boundary surfaces
- Create variable section sweep surfaces
- Create helical sweep surfaces
- Create swept blend surfaces
- Utilize surface analysis tools
- Extend and trim surfaces
- Manipulate surfaces
- Create and edit solid models using surface quilts
- Utilize the master model technique

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Pro/ENGINEER Wildfire 5.0 Update from Pro/ENGINEER Wildfire 4.0.

Audience

- This course is intended for mechanical designers, design engineers, industrial designers, and related roles. The topics in this course are also available as Web-based training courses.

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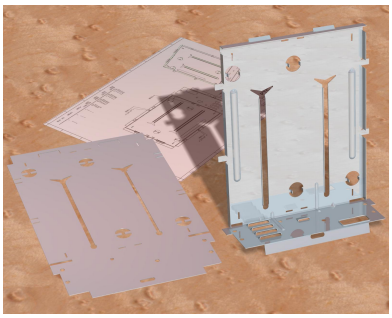
PTC university

Sheetmetal Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

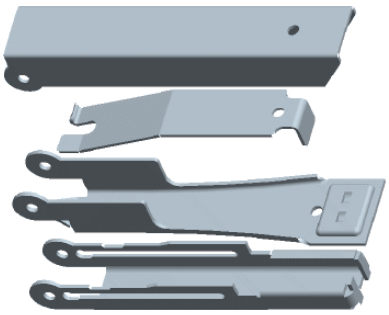
Overview

Course Code	WBT-2240-0
Course Length	16 Hours



Sheetmetal Design using Pro/ENGINEER Wildfire 5.0 is a comprehensive training course that teaches you how to create sheetmetal parts in Pro/ENGINEER. The course builds upon the basic lessons you learned in Introduction to Pro/ENGINEER Wildfire 5.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including sheetmetal production drawings. All the functions needed to create sheetmetal parts, drawings, and assemblies are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model, and document both in production drawings. At the end of each day, you use the Pro/FICIENCY skills assessments to reinforce your understanding of the course topics.

Course Objectives



- The Sheetmetal Design Process
- Sheetmetal Model Creation, Conversion, and Display
- Methods of Developed Length Calculation
- Primary Wall Features
- Secondary Wall Features
- Partial Walls
- Bend Relief
- Unbend and Bend Back Features
- Sheetmetal Bend Features
- Flat Patterns
- Sheetmetal Cuts
- Forms
- Notch and Punch Features
- Sheetmetal Environment Setup
- Sheetmetal Design Information Tools
- Sheetmetal Design Rules
- Detailing Sheetmetal Designs
- Sheetmetal Design Project

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- This course is intended for design engineers, mechanical designers, and industrial designers. People in related roles can also benefit from taking this course.

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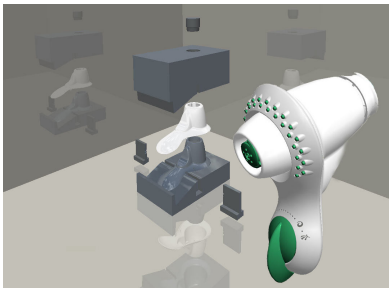
PTC university

Mold Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

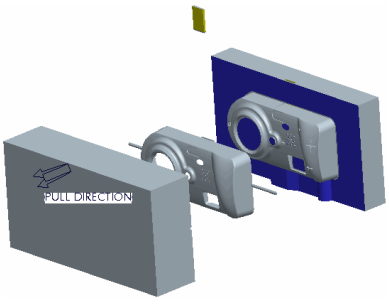
Course Code	TRN-2241–T
Course Length	2 days



Pro/MOLDESIGN provides the tools to create a mold model from start to finish by using the mold design process within Pro/ENGINEER Wildfire 5.0. In this course, you will learn how to create, modify, and analyze mold components and assemblies. Any changes made to the design model automatically propagate to the mold components and assemblies. You will learn how to create final extract components that reflect the geometry of the design model, along with shrinkage considerations, adequate drafting, mold features, and cooling systems. After completing the course, you will have a better understanding of the mold design process and how to create molded products by using the mold design process. 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

- Learn the basic mold process
- Prepare design models for the mold process
- Analyze design models to ensure their readiness for molding
- Create mold models
- Apply shrinkage to the reference model
- Create and assemble workpieces into the mold model
- Create mold volumes
- Create parting lines and parting surfaces
- Split mold volumes
- Extract mold components
- Create mold features
- Learn how to fill and open the mold



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Basic understanding of industry standard Mold design terminology and processes.
- Knowledge of Pro/ENGINEER surfacing techniques a plus.

Audience

- This course is intended for designers, machinists, and manufacturing engineers. The topics in this course are also available as a Web-based training course.

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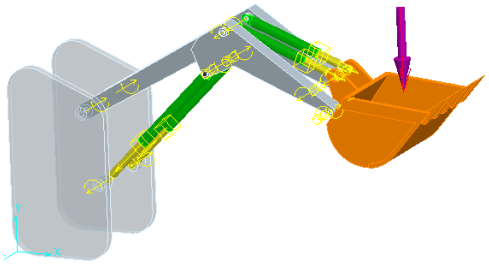
PTC university

Mechanism Design using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

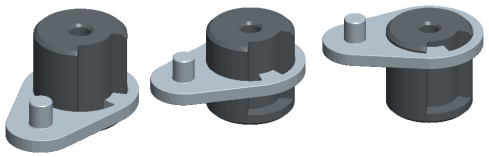
Course Code	WBT-2242-0
Course Length	8 Hours



Mechanism Design using Pro/ENGINEER Wildfire 5.0 is designed for experienced users who want to add motion to their models by creating mechanism connections and servo motors. In Pro/ENGINEER Wildfire 5.0 you can add motion to your models using the standard mechanism functionality, often referred to as the Mechanism Design Extension (MDX). In this course, you will learn about creating mechanism connections, configuring the mechanism model, creating a kinematic analysis, and evaluating results. These topics will enable you to simulate the range of motion between components in your moving assemblies, create gear connections that simulate the gear ratios, create Cam connections that enable Pro/ENGINEER parts to "push" other parts they come into contact with, and check for collisions between moving components. After completing this course, you will be prepared to work on mechanism designs using Pro/ENGINEER Wildfire Mechanism Design. Pro/FICIENCY assessments will be provided in order for you to assess your understanding of the course materials. The assessment results will also identify the class topics that require further review. At the end of the class, you will either take an assessment via your PTC University account, or your instructor will provide training on how to do this after the class.

Course Objectives

- Introduction to the mechanism design process
- Creating mechanism connections
- Configuring motion and analysis
- Evaluating analysis results



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers and mechanical designers who need to add and evaluate the motion of their assemblies.

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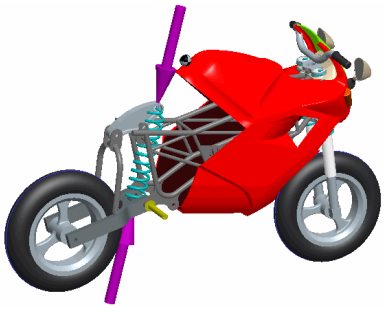
PTC university

Mechanism Simulation using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

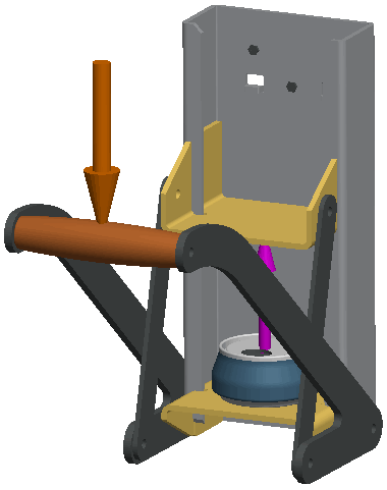
Course Code	WBT-2243-0
Course Length	8 Hours



This course is designed for experienced users who want to add motion to their products and analyze dynamic reactions of moving components. In this course, you will focus on learning advanced modeling and analysis skills in this comprehensive, hands-on course. Topics include developing the 3-D model, analyzing the mechanism model, and evaluating results. These topics will enable you to measure dynamic reactions of components, measure the force required to keep a mechanism balanced, and determine the resting state of a mechanism. After completing this course, you will be prepared to work on mechanism designs using Pro/ENGINEER Wildfire Mechanism Dynamics Option (MDO). 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

- Understanding the mechanism dynamics option
- Applying force motors, springs, and dampers to assemblies
- Applying forces, torques, and gravity to assemblies
- Creating dynamic analyses
- Creating force balance analyses
- Creating static analyses
- Measuring forces, velocities, accelerations, and other reactions
- Evaluating results



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0
- Mechanism Design using Pro/ENGINEER Wildfire 5.0

Audience

- Design engineers and mechanical designers who need to add and evaluate the motion of their assemblies.

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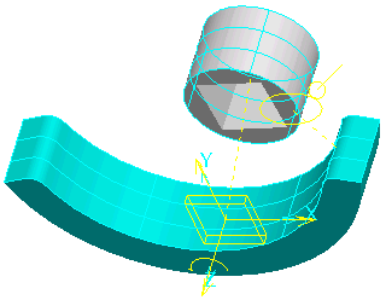
PTC university

Behavioral Modeling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

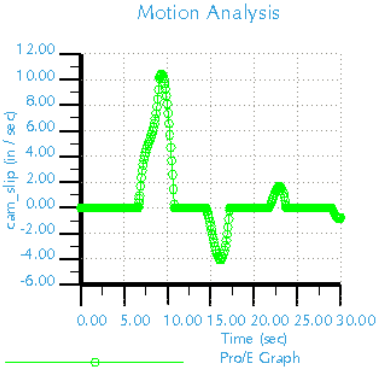
Course Code	WBT-2244-0
Course Length	8 Hours



In this course, you will focus on learning advanced analysis skills unrelated to structural or thermal analysis. Also, you will learn how to analyze your models and create analysis features that can enforce your design intent. You will also learn how to create sensitivity and feasibility studies that aid you in determining how to reach your design goals. Furthermore, you will learn how to create optimization design studies that enable you to configure the dimensions and parameters that Pro/ENGINEER can change in order to meet your design specifications. This course is designed for experienced users who want to add additional features that enable you to meet or exceed the design specifications of your products. At the end of each module, you will find a set of review questions to reinforce critical topics from that module. After completing this course, you will be prepared to work on critical component designs using Pro/ENGINEER Wildfire Behavioral Modeling. At the end of the course, you will find a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Applying the Behavioral Modeling process and concepts to your designs
- Creating measurement analysis features
- Creating relation, motion, Mechanica, and MS Excel analysis features
- Creating user-defined analysis features
- Conducting sensitivity analyses
- Conducting feasibility and optimization studies
- Design Project



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0 or equivalent experience.
- Experience with MS Excel, Mechanica, Mechanism Design, and Mechanism Dynamics is useful but not required.

Audience

- This course is intended for product designers and engineers. Related roles will also benefit from taking this course.

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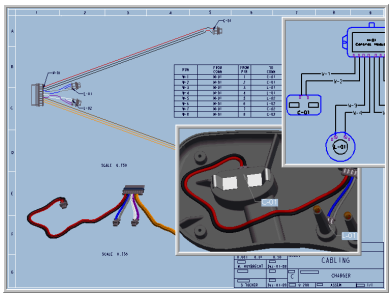
PTC university

Cabling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	WBT-2245-0
Course Length	24 Hours



In this course, you will learn how to create 3-D electrical harnesses using Pro/ENGINEER Wildfire 5.0. This includes using Routed Systems Designer 9.0 to pass schematic diagram information into the 3-D harness designs created within Pro/ENGINEER Wildfire 5.0.

You will learn how to route electrical harnesses both with and without schematic diagram information, create flattened harnesses for manufacturing, and document harness designs by creating flattened harness drawings that include customized BOM tables and wire list information.

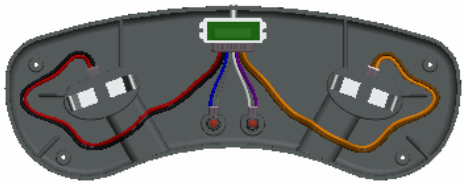
A significant portion of the course is devoted to a cabling design project, where students create a full wiring harness with minimal 'picks and clicks' to solidify techniques learned previously in the course.

At the end of each module, you will complete on-line learning assessments that reinforce your understanding of the course topics. There is also a wrap-up assessment at the end of the course.

After successfully completing the course, you will be able to create 3-D electrical harnesses and associated manufacturing deliverables using Pro/ENGINEER Wildfire 5.0.

Optionally, students may wish to attend the Routed Systems Designer 9.0 for Pro/ENGINEER Wildfire 5.0 course. This will enable a full understanding of the schematic design process used to provide schematic data for the creation of electrical harness assemblies in Pro/ENGINEER Wildfire 5.0.

Course Objectives



- Learning the basic Pro/ENGINEER Cabling Process
- Creating harness assembly structures
- Setting up for cabling
- Routing wires and cables
- Modifying wire routings
- Routing and utilizing networks
- Establishing logical references
- Routing wires and cables using logical data
- Creating harness components and cosmetics
- Creating flat harnesses
- Documenting harness designs
- Comprehensive Design Project

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0.
- Routed Systems Designer 9.0 for Pro/ENGINEER Wildfire 5.0 (Optional).

Audience

- This course is intended for engineers involved in the 3-D routing and documenting of electrical wiring and cabling harnesses. The topics in this course are also available as Web-based training courses.

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PTC university

Piping using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	WBT-2246-0
Course Length	8 Hours

In this course, you will learn how to manually create (non-specification driven) mechanical piping designs using Pro/ENGINEER Wildfire. This includes learning how to configure pipelines, how to route pipelines, and how to insert pipe fittings such as valves and reducers.

You will also learn how to create specification driven industrial piping designs using Pro/ENGINEER Wildfire. This includes learning how to use schematic diagrams created with Routed Systems Designer to drive 3-D industrial piping designs created within Pro/ENGINEER Wildfire.

You will learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information. You will also learn how to export ISOGEN format files for creating pipeline, spool and systems isometric drawings.

Course Objectives

- Understand the manual piping design process.
- Understand the specification-driven piping design process.
- Create piping assembly structures.
- Configure and route pipelines.
- Move and modify pipelines.
- Create pipe solids and fabricate pipes.
- Configure and insert fittings.
- Create piping report information.
- Create piping drawings.
- Configure a piping specification database.
- Configure project specific data files.
- Create specification-driven pipelines.
- Create schematic driven pipelines.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0

Audience

- This course is intended for engineers, involved in the 3-D routing of mechanical piping systems and industrial piping systems.

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PTC university

Routed Systems Designer 9.0 for Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	SAB-CEK 4585
Course Length	24 Hours

In this course, you will learn how to use Routed Systems Designer (RSD) to create schematic diagrams for both electrical harness and piping designs. You will learn how to administer the RSD working environment, and how to configure the catalog library. You will learn how to create functional block diagrams and block interconnect diagrams. You will learn how to create electrical circuit diagrams, and wiring diagrams, including wire interconnect diagrams. You will also learn how to create process and instrumentation design (P&ID) diagrams for piping systems. Finally, you will learn how to use wiring diagrams and P&ID diagrams to configure 3-D harness designs and industrial piping designs created within Pro/ENGINEER Wildfire. At the end of each module, you will complete Pro/FICIENCY skills assessment questions. These questions are used to help reinforce your understanding of the course topics and form the basis for review sessions.

Course Objectives

- Understanding the 2-D schematic design processes for electrical harness designs and piping designs.
- Configuring the RSD working environment.
- Creating and configuring the catalog library.
- Creating functional block diagrams, (including block interconnect diagrams).
- Creating circuit diagrams.
- Creating wiring diagrams, (including wire interconnect diagrams).
- Creating process and instrumentation design diagrams.
- Communicating diagram information to Pro/ENGINEER Wildfire.

Prerequisites

- None

Audience

- This course is intended for engineers, involved in the schematic 2-D layout of either electrical diagrams or piping diagrams.

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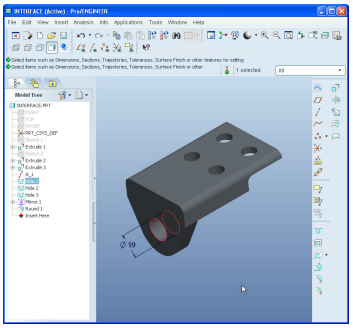
PTC university

Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) Update from Pro/ENGINEER Wildfire 2.0

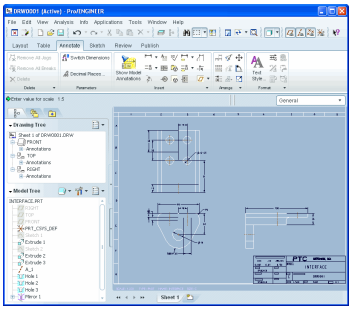
Overview

Course Code	SAB-CEK3076
Course Length	24 Hours



In this course, you will learn how to utilize the core functionality enhancements in Pro/ENGINEER Wildfire 5.0, when upgrading from Pro/ENGINEER Wildfire 2.0. First, changes to the interface will be explored, such as the new skin and screen colors. Next, you will become familiar with the enhancements to Sketcher geometry and tools, such as sketching parallelograms and chamfers, as well as using the sketcher diagnostic tools. Next, you will explore enhancements to Part mode, including creating trajectory ribs, pattern and UDF enhancements, the all-new resolve mode, and auto-round. You will also learn about the Assembly dashboard interface and other assembly enhancements such as the accessory window, the new reference viewer, improved simplified rep functionality, and mechanism enhancements to gears and belts. Next, you will examine the new ribbon interface and drawing tree for drawing mode, as well as numerous detailing enhancements. Finally, you will review Sheetmetal enhancements for flat and flange walls, in addition to patterning and mirroring wall features. 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



- Describe the Interface Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Geometry Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sketcher Tools Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Part Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Feature Duplication Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Part Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Assembly Modeling Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Advanced Assembly Enhancements for Pro/ENGINEER Wildfire 5.0
- Resolve model failures in Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Sheet and View Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Drawing Detailing Enhancements for Pro/ENGINEER Wildfire 5.0
- Utilize the Sheetmetal Enhancements for Pro/ENGINEER Wildfire 5.0

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 2.0 or equivalent experience.

Audience

- This course is intended for people who have already upgraded to Pro/ENGINEER Wildfire 5.0 from Pro/ENGINEER Wildfire 2.0.

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Advanced Turning and Multi-task Machining using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0)

Overview

Course Code	WBT-5171-0
Course Length	16 Hours

This course provides a number of tutorial style topics that describe and demonstrate how to machine components using mill-turn machines, twin-turret machines, and in-line twin spindle lathes. You will learn how to create turning tool paths and milling tool paths for each machine type where appropriate.

Tutorials will also show you how to post-process mill-turn toolpaths, and how to use the material removal simulation tools NC-Check and Vericut.

Finally, there are a series of tutorials that will show you how to view machine tool simulation's for 3-axis milling, 2-axis turning, and 5-axis mill-turn machines.

The lab files used in the tutorial demonstrations are provided for students. These files are intended to enable students to complete the same steps shown in each tutorial demonstration.

Course Objectives

- Create NC sequences for mill-turn machines.
- Create NC sequences for twin-turret turning machines.
- Create NC sequences for in-line twin spindle lathes.
- Create workcells for mill-turn machines, twin-turret turning machines, and in-line twin spindle lathes.
- Learn how to post-process mill-turn tool paths.
- Learn how to use the material removal simulation tools NC-Check and Vericut.
- Learn how to view machine tool simulation's for 3-axis milling, 2-axis turning, and 5-axis mill-turn machines.

Prerequisites

- Milling using Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5.0) or equivalent experience.

Audience

This course is intended for manufacturing engineers and NC machinists.

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