NOTE: For a graphical depiction of the curriculum based on job role, please visit this page:
http://www.ptc.com/services/edserv/learning/path...
Live Classroom Curriculum Guide

- Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0
- Advanced Component Operations using ProENGINEER Wildfire 3.0
- Creating Style and Warp Features using ProENGINEER Wildfire 3.0
- Analyzing and Optimizing Design Models with ProENGINEER Wildfire 3.0
- Creating Geometry from Surfaces and Curves using ProENGINEER Wildfire 3.0
- Using Component Interfaces and Flexible Components in ProENGINEER Wildfire 3.0
Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0

Course Code   TRN-2104-T
Course Length  1 Day

Overview

After completing this two hour session, you should be able to create Pro/ENGINEER models with embedded information like dimensions, notes, GD&T, and surface finishes. In addition, you learn how to add information related to manufacturing tools, and manufacturing processes. You can use the 3-D Drawings for planning the downstream process map, layouts, and tools at an early stage of the product development cycle. At the end of the session, you will be able to create 3-D drawings compliant to the ASME Y14.41 standard, manage the information added, display and use this information in traditional 2-D drawings if needed.

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 create 3-D drawings using Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

• Introduction to Pro/ENGINEER Wildfire 3.0.
• Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

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.

Topics

• Describe the Digital Product Definition.
• Create 3-D drawings using Annotation features.
• Use Annotation features to add notes, dimensions, geometric tolerances, surface finish and manufacturing information.
• Create 2-D drawings using Annotation elements.
• Group and manage the information in 3-D Drawings
## Agenda

### Day 1

| Module 1 | Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0 |
Advanced Component Operations using Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to perform advanced operations on design models. You will learn how to add and remove material in components by using the merge operation. You will also learn how to create new components using mirror and intersect operations. Additionally, you will learn how to create features such as holes and cuts at the assembly level.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to perform advanced operations to create and modify components in Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

**Audience**

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

**Topics**

- Add and remove material from components by using the general merge operation.
- Create new components by using intersect and mirror operations.
- Create assembly level features.
## Agenda

### Day 1

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Creating Style and Warp Features using Pro/ENGINEER Wildfire 3.0

Course Code | TRN-1860-T
Course Length | 1 Day

Overview

In this course, you will learn how to create sketch-based features with aesthetic and organic shapes by using styled surface features. You will also learn how to alter the form and shape of solids, quilts, facets, and curves by using warp features.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create style and warp features using Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

Audience

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

Topics

- Create style curves, surfaces and features
- Create warp features to alter design model geometry
## Agenda

### Day 1

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</table>
Analyzing and Optimizing Design Models with Pro/ENGINEER Wildfire 3.0

Overview

In this course, you will learn how to create various types of analyses including measuring distances, calculating mass properties and interferences, and examining surface curvature and continuity in design models. You will be able to save these analyses as features that update along with changes to the 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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create analysis features and create feasibility and optimization studies using Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

Audience

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

Topics

- Create various types of design model analyses
- Create analysis features
- Create feasibility and optimization studies
# Agenda

## Day 1

| Module 1 | Analyzing and Optimizing Design Models |
Creating Geometry from Surfaces and Curves using Pro/ENGINEER Wildfire 3.0

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Overview

In this course, you will learn how to use a variety of surface and datum curve tools to duplicate geometry, manipulate existing features, or create new features. You will learn how to duplicate surfaces using copy, offset, move, and mirror tools, and how to edit existing surfaces using the merge, intersect, trim, and extend tools. You will also learn how to create datum curves using the project and wrap tools, and use those datum curves and edges 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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create geometry from surfaces and curves using Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

Audience

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

Topics

- Duplicate, manipulate, and create surface features.
- Create solid geometry from existing surface features.
Agenda

Day 1

| Module 1 | Creating Geometry from Surfaces and Curves |
Using Component Interfaces and Flexible Components in Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to create component interfaces to quickly assemble components within an assembly, both manually and automatically. Flexible components will enable you to assemble the same component multiple times while varying dimensions as each component is added.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create component interfaces and flexible components using Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

**Audience**

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

**Topics**

- Assemble components using component interfaces
- Assemble flexible components
- Describe how parent/child relationships affect assembly components
## Agenda

### Day 1

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Web Based Curriculum Guide

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- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0
- Advanced Assembly Management with Pro/ENGINEER Wildfire 3.0
- Creating Production Drawings with Pro/ENGINEER Wildfire 3.0
- Advanced Part Modeling with Pro/ENGINEER Wildfire 3.0
- Surface Modeling with Pro/ENGINEER Wildfire 3.0
- Sheetmetal Design using Pro/ENGINEER Wildfire 3.0
- Freeform Surface Modeling using Pro/ENGINEER Wildfire 3.0
- Mechanism Design using Pro/ENGINEER Wildfire 3.0
- Mechanism Simulation using Pro/ENGINEER Wildfire 3.0
- Mold Design using Pro/ENGINEER Wildfire 3.0
- Introduction to Pro/ENGINEER Wildfire 3.0 - Fundamentals
- Introduction to Pro/ENGINEER Wildfire 3.0 - Productivity Tools
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER 2001
- Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0
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- Milling using Pro/ENGINEER Wildfire 3.0
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- Creating Style and Warp Features using ProENGINEER Wildfire 3.0
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- ProENGINEER Mechanica Simulation using ProENGINEER Wildfire 3.0- Optimization
- Milling using ProENGINEER Wildfire 3.0- Setting Up
- Milling using ProENGINEER Wildfire 3.0- Creating NC Data
- Milling using ProENGINEER Wildfire 3.0- Managing NC Data
- Using Component Interfaces and Flexible Components in ProENGINEER Wildfire 3.0
- Reverse Engineering using Pro/ENGINEER Wildfire 3.0
- Introduction to Routed Systems Designer 6.0 for Pro/ENGINEER Wildfire 3.0
Introduction to Pro/ENGINEER Wildfire 3.0

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Overview

This course is designed for new users who want to become proficient with Pro/ENGINEER Wildfire 3.0 as quickly as possible. You will focus on learning core-modeling skills in this comprehensive, hands-on course. 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. Both in the middle and 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.

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.

Topics

- Capturing Sketched Design Intent
- Creating Sketched-based Features
- Creating Part Models
- Assembling Components with Constraints
- Assembling Components with Connections
- Creating Drawings
- Editing Models
- Creating Reference Geometry
- Creating Direct Features
- Managing Parent/Child Relationships
- Resolving Regeneration Failures
- Managing Layers
- Managing Assemblies
- Duplicating Design Model Features
• Analyzing Design Models
• Comprehensive Design Project
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Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

Course Code | TRN-WBT1827-S
Course Length | 16 Hours

Overview

In this course you will learn how to utilize many of the enhancements to core functionality in Pro/ENGINEER Wildfire 3.0. This includes learning how to use new sketcher functionality including how to copy and paste items in sketcher and how to insert predefined shapes such as polygons and common engineering sections, such as I-beams. You learn new methods for creating features such as swept blends, and new methods for placing user defined features. You review new capabilities for feature manipulation, such as enhanced patterning capabilities. You review assembly enhancements, including the new user interface for assembling components. You use new drawing functionality, including how to create shaded drawing views. You also learn about sheetmetal enhancements including how to create multiple walls in a single operation.

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.

Prerequisites

- Completion of T1707-340 Pro/ENGINEER Wildfire 2.0 Update or equivalent experience.

Audience

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

Topics

- Sketcher Enhancements
  - Can now cut or copy, and then paste sketch geometry within a sketch or from feature to feature.
  - Locked dimensions maintained outside of sketcher. Strong dimensions can now be automatically locked.
  - Sketcher Orientation: New workflows when using sketcher, dynamic preview when specifying orientation.
  - Sketcher Palette: Predefined sections such as polygons and common engineering sections, e.g. I-beams can be configured and retrieved from a palette.
- Feature and Interface Enhancements
  - Swept Blend: New user interface, including interactive preview capability.
  - Warp Feature: Improved highlighting during selection and collection.
User Defined Features: New user interface when placing features including updated preview.

Embedded Datum Features: Datum features created during the configuration of other features are now embedded within the primary feature and hidden in the model tree.

Partial Shell: Exclude portions of model from shell operation.

Patterning Enhancements. Many enhancements, including patterning along a curve, patterning an existing pattern, and moving and mirroring of patterns.

Assembly Enhancements
- Dragging components in assembly mode and real time collision detection when dragging components.
- Undo and redo now available for common component operations.

Drawing Enhancements
- Drawing Views. Can now create and plot shaded drawing views, this includes displaying clipped graphical views.
- Drawing Standards. Additional options for display dimensions, geometric tolerances and other drawing entities

Sheetmetal Enhancements
- Flange Wall Tool Enhancements: Single operation to create multiple walls and mitered cuts.
- Unattached Walls: New user interface for creating unattached flat and extruded walls.
- Sheetmetal Cut: Consolidated solid and sheetmetal cut tool, now uses new user interface.
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Advanced Assembly Management with Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1828-S
Course Length | 24 Hours

Overview

Pro/ENGINEER Wildfire 3.0 enables you to create complex assemblies using a top-down design process.

In this course, you will learn how to use Pro/ENGINEER Wildfire 3.0 to create and manage complex assemblies using top-down design techniques. In the top-down design process, you start an assembly design by creating a layout. The layout contains specifications and parameters that are used to control the entire assembly design. Then you create a preliminary assembly structure. This structure contains the components and their hierarchy within the assembly. Next, you use skeletons to define critical component dimensions and mounting locations, space requirements, and the motion between the assembly components. Finally, you create component geometry by referencing the skeletons and sharing design information within the assembly.

In addition, you will also learn how to reduce file size and regeneration time in complex parts, assemblies and drawings by using simplified representations. Simplified representations enable you to exclude components that are not necessary to the current design task, thereby reducing geometry detail and improving system response times.

You will apply the concepts and techniques that you learnt in the course to complete a top-down design project.

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 the course, you will have a better understanding of the top-down design process and how to work efficiently with large, complex product designs using Pro/ENGINEER Wildfire 3.0.

Duration

3 days

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

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.
Topics

- Create design layouts and assembly structures.
- Creating design frameworks using skeletons.
- Creating design geometry using data sharing features.
- Managing external references in design models.
- Modifying assembly structures using layouts and skeletons.
- Reducing regeneration times in complex parts and drawings.
- Creating simplified representations of complex assemblies, using various selection techniques.
- Replacing and substituting components in assemblies.
- Modifying components in simplified representations using on-demand simplified representations.
- Updating simplified representations using definition rules.
- Creating simplified representations of complex drawings.
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Creating Production Drawings with Pro/ENGINEER Wildfire 3.0

Overview
Creating Production Drawings with Pro/ENGINEER Wildfire 3.0 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 will learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and associative nature of Pro/ENGINEER Wildfire 3.0 when manipulating drawings. You will also learn system administration information relating to drawings. Upon completion of this course, you will be able to create complete production drawings suitable for manufacturing.

At the end of each topic, you use the Pro/FICIENCY skills assessments to reinforce your understanding of the course topics. Your instructor utilizes the results from the anonymous skills assessments as the basis for daily review sessions.

Duration
3 days

Prerequisites
- Introduction to Pro/ENGINEER Wildfire 3.0.

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.

Topics
- Creating views of parts, assemblies, and multiple models on drawings.
- Creating dimensions and notes.
- Controlling display options using layers.
- Applying linear and geometric tolerances in drawings.
- Adding 2-D drafting and symbols to drawings.
- Creating drawing tables and Bill of Materials.
- Documenting family tables within drawings to create parts catalogs.
- Creating drawing formats, templates and standards.
- Managing large drawings.
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Advanced Part Modeling with Pro/ENGINEER Wildfire 3.0

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<td>Course Length</td>
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**Overview**

The Advanced Part Modeling with Pro/ENGINEER Wildfire 3.0 training course teaches you how to use advanced part modeling techniques in Pro/ENGINEER Wildfire 3.0 to improve your product designs. 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 3.0.

**Duration**

2 days

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

**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.

**Topics**

- Creating sketch-based features using advanced sketching techniques such as conics, splines, and elliptical fillets.
- Creating finishing features such as rounds, chamfers, drafts and shells using advanced options.
- Creating sweeps and blend features using advanced options.
- Duplicating features and components using advanced patterning techniques.
- Duplicating features and components by using pattern tables.
- Reusing design geometry by using inheritance and user-defined features (UDF).
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Surface Modeling with Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1832-S
Course Length | 24 hours

Overview

In Pro/ENGINEER Wildfire 3.0, you can use surface modeling to create design models with shapes that are too complex for solid features. In this course, you learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You can then create solids using the surfaces as references. You will also learn how to analyze surfaces for quality as well as manipulate surfaces using the various editing tools available in Pro/ENGINEER Wildfire 3.0.

After completing this course, you will be well prepared to create complexly shaped design models using surfaces in Pro/ENGINEER Wildfire 3.0.

At the end of each day, you use the Pro/FICIENCY skills assessments to reinforce your understanding of the course topics. Your instructor utilizes the results from the anonymous skills assessments as the basis for daily review sessions.

Prerequisites

- Pro/ENGINEER Wildfire 3.0 Update training.
- Introduction to Pro/ENGINEER Wildfire 3.0.

Audience

Design engineers, mechanical designers, and industrial designers

Topics

- Creating basic surfaces using techniques such as Extrude, Revolve, Sweeps and Blends.
- Understanding the surface modeling workflow.
- Creating a network of curves using various techniques such as through points and sketches.
- Creating surfaces using a network of curves as boundaries.
- Creating surfaces using variable section sweep and swept blend techniques.
- Creating surfaces using advanced blend tools such as Section to Surface, Surface to Surface and Tangent to Surface.
- Using analysis tools to check for surface quality and potential problems.
- Manipulating surfaces using various editing tools such as Extend, Merge, Trim and Offset.
- Creating and editing solids using surfaces and quilts.
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Sheetmetal Design using Pro/ENGINEER Wildfire 3.0

Course Code: TRN-WBT1848-S
Course Length: 16 hours

Overview

Sheetmetal Design using Pro/ENGINEER Wildfire 3.0 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 T1803 - Introduction to Pro/ENGINEER Wildfire 3.0 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. Your instructor utilizes the results from the anonymous skills assessments as the basis for daily review sessions.

Prerequisites

- Successful completion of T1803 - Introduction to Pro/ENGINEER Wildfire 3.0 or equivalent experience.

Audience

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

Topics

- Sheetmetal Part Construction Philosophy
- Sheetmetal Construction Features
- Setting up the Sheetmetal Design Environment
- Bend Tables, Start Parts, and Templates
- Documenting Bend Order Sequences with Bend Order Tables
- Generating Flat State Models for Manufacturing
- Sheetmetal Drawings
- Converting Solid Parts to Sheetmetal Parts
- Application of User-Defined Features in Sheetmetal Design
- Sheetmetal Information Tools
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Freeform Surface Modeling using Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1851-S
Course Length | 16 Hours

Overview

In Pro/ENGINEER Wildfire 3.0 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 learn how to use the Style tool to create and manipulate freeform curves, freeform surfaces, freeform surface details, and advanced freeform surface models. You also learn how to integrate style features with other parametric features in 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 design complex-shaped freeform surface models in Pro/ENGINEER Wildfire 3.0.

Prerequisites

- Pro/ENGINEER Wildfire 3.0 Update training.

Audience

Design engineers, mechanical designers, industrial designers, and related roles. The topics in this course are also available as Web-based training courses.

Topics

- Describing the Style modeling environment.
- Creating and editing 2-D, 3-D, and planar style curves.
- Analyzing and modifying curve shapes.
- Creating style geometry using images, design references, and imported data.
- Creating different types of style surfaces such as boundary, loft and blend surfaces.
- Creating curves using techniques such as sketching, dropping curves on surfaces, and using the Curve from Surface option.
- Manipulating curves and surfaces in complex models.
- Creating continuous curves and surfaces.
• Integrating freeform style features with other parametric features.
• Creating features such as scoops, bulges and recesses using Style.
• Overbuilding surfaces using four boundaries.
• Creating surfaces using the Create Boundaries technique.
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Mechanism Design using Pro/ENGINEER Wildfire 3.0

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Overview

This course is designed for experienced users who want to add motion to their products. You will focus on learning advanced assembly skills in this comprehensive, hands-on course. Topics include 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. Furthermore, you learn about how you can use Design Animation to animate your product, how you can use Mechanism Dynamics to simulate reaction forces between components, and how you can use Behavioral Modeling to optimize your mechanism.

After completing this course, you will be prepared to work on mechanism designs using Pro/ENGINEER Wildfire Mechanism Design.

Pro/FIICIENCY 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.

Prerequisites

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

Audience

This course is intended for product designers. People in related roles will also benefit from taking this course.

Topics

- Identifying Differences between Mechanism Design Extension and Mechanism Dynamics Option
- Creating Bodies
- Creating Mechanism Connections in Pro/ENGINEER
- Creating Cam-Follower and Gear Pair Connections
- Applying Servo Motors to Assemblies
- Creating Position Analyses
- Creating Kinematic Analyses
- Evaluating Results
- Creating Trace Curves and Motion Envelopes
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Mechanism Simulation using Pro/ENGINEER Wildfire 3.0

Course Code  TRN-WBT1855-S
Course Length  8 Hours

Overview

This course is designed for experienced users who want to add motion to their products and analyze dynamic reactions of moving components. 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.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0 or equivalent experience.
- Mechanism Design using Pro/ENGINEER Wildfire 3.0 or equivalent experience with creating mechanism joint and cam connections.

Audience

This course is intended for product designers. People in related roles will also benefit from taking this course.

Topics

- Identifying Differences between Mechanism Design Extension and Mechanism Dynamics Option
- Applying 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 on Assemblies
- Evaluating Results
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Mold Design using Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1890-S
Course Length | 16 Hours

Overview

Pro/MOLDESIGN provides the tools to create a mold model from start to finish by using the mold design process within Pro/ENGINEER Wildfire 3.0.

In this course, you 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 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.

You apply the concepts and techniques that you learn in the course to complete a mold design project.

There is an optional appendix where you use mold layout to assemble a vendor mold base and create ejector pins.

Pro/FICIENCY assessments are provided for you to assess your understanding of the course materials. The assessment results 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 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.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Basic understanding of industry standard Mold design terminology and processes.

Audience

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

Topics

- Introduction to Mold Design
- Analyzing Design Models
- Creating Mold Models
- Creating Sliders
- Creating Parting Surfaces
- Creating Mold Components
- Creating Mold Features
- Filling and Opening the Mold
- Using Mold Layout
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Introduction to Pro/ENGINEER Wildfire 3.0 - Fundamentals

Overview

This course is designed for new users who want to become proficient with Pro/ENGINEER Wildfire 3.0 as quickly as possible. You will focus on learning core-modeling skills in this comprehensive, hands-on course. 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.

After completing the course you will be well prepared to work effectively on product design projects using Pro/ENGINEER Wildfire.

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.

Topics

- Capturing Sketched Design Intent
- Creating Sketched-based Features
- Creating Part Models
- Assembling Components with Constraints
- Editing Models
- Creating Reference Geometry
- Creating Direct Features
- Managing Parent/Child Relationships
- Comprehensive Design Project
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Introduction to Pro/ENGINEER Wildfire 3.0 - Productivity Tools

Course Code | TRN-WBT1894-S
Course Length | 24 Hours

Overview

This course is designed for existing Pro/ENGINEER Wildfire 3.0 users who want to increase their productivity. You will focus on learning advanced modeling skills in this comprehensive, hands-on course. Topics include resolving regeneration errors; sketching advanced features; creating family tables, Simplified Representations, and explode states; and analyzing part and assembly models. 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.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0 - Fundamentals

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.

Topics

- Assembling Components with Connections
- Creating Drawings
- Resolving Regeneration Failures
- Managing Layers
- Managing Assemblies
- Duplicating Design Model Features
- Analyzing Design Models
- Comprehensive Design Project
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Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER 2001

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Overview

With the release of Pro/ENGINEER Wildfire 3.0, there have been many productivity and functionality improvements from Pro/ENGINEER 2001. These include a new user interface, a consolidated set of feature tools, and the ability to interact directly with models and features.

The Pro/ENGINEER Wildfire 3.0 user interface includes integrated web and folder browsers, providing easy access to design information and model data. Pro/ENGINEER feature tools utilize a dashboard interface, providing simple and intuitive access to options for creating and editing features. Creating sketches has been made easier through use of new simplified workflows. Modifying models is made easier through the use of drag handles that enable dynamic modifying of model geometry, and the new undo/redo tool enables reversing of operations such as deleting and editing features. There are many enhancements to feature editing tools, including several new patterning options, and new tools for quickly and dynamically grouping, copying and pasting features. Creating assemblies has been made easier by utilizing a dashboard interface for assembling components. Creating drawings has been made easier using a single new drawing view dialog box that consolidates all drawing view options. Creating and editing of sheetmetal parts is now quicker through the use of new dashboard tools that consolidate several wall types, enabling easy editing and previewing of wall geometry and the ability to create multiple walls in a single operation.

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.

Prerequisites

- Completion of T1779-320 Introduction to Pro/ENGINEER or equivalent experience.

Audience

This course is intended for people who will transition directly from Pro/ENGINEER 2001 to Pro/ENGINEER Wildfire 3.0.

Topics

- User Interface.
  - Using the new flexible workflow.
  - Wildfire User Interface: Navigator, Folder Browser, and Integrated Web Browser.
  - Spin / Pan / Zoom Operations and Orient Mode.
• Selecting and Editing Design Models.
  • Methods for selecting components, features and geometry.
  • Editing features and model dimensions.
  • Modifying model geometry dynamically with drag handles.
  • Changing design intent with undo and redo.
• Creating Direct and Datum Features.
  • Feature tools: Holes, Rounds, Chamfers, Shells, Drafts.
  • Datum tools: Axes, Planes, Points, Coordinate systems, Reference Features.
  • Datum curves: Project, Wrap, Intersect.
• Sketcher Enhancements
  • Sketcher Orientation: New workflows when using sketcher.
  • Sketcher Palette: Predefined sections such as polygons and common engineering sections, from a palette.
  • Sketcher: Axis of Revolution, Sketched Text enhancements.
• Feature Enhancements
  • Feature tools: Extrude, Revolve, Rib, Variable Section Sweep, Boundary Blend, Swept Blend, and Warp Feature.
  • Embedded Datum Features: Datum features created during the configuration of other features are now embedded within the primary feature and hidden in the model tree.
  • Partial Shell: Exclude portions of model from shell operation.
• Editing Features
  • Patterns: Fill Patterns, Linear Directional and Rotational Axis Patterns. patterning along a curve, patterning an existing pattern, and moving and mirroring of patterns
  • Groups: Creating and modifying groups in the model tree.
  • Duplicating Features: Copy, Paste, Paste Special, Mirror.
  • Surface and Curve Tools: Fill, Trim, Merge, Extend, Intersect, Offset, Thicken, Solidify
• Managing Design Models.
  • Creating Cross-sections.
  • Relations and Parameters interface.
  • Layers and the layer tree.
  • Using the new search tool.
  • Managing parent/child relationships.
  • Web-based feature and model information tools.
  • New Troubleshooter tool.
• Creating and Modifying Assemblies
  • New Component Placement User Interface. New methods for placing components using connections.
  • Dragging components in assembly mode and real time collision detection when dragging components.
  • Undo and redo now available for common component operations.
  • Assembly layers and assembly features.
  • View Manager: Simplified Reps, Style Reps, Symbolic Reps, Explode States, Orientations, and Cross-Sections.
  • Flexible components and component interfaces
• Creating and Modifying Drawings
  • Drawing Views. Control creation and modification of drawing views using a single new drawing view dialog box.
  • Creating and plotting shaded drawing views, this includes displaying clipped graphical views.
  • Enhancements to BOM Balloons and Repeat Regions.
• Undo/Redo, Selection Mechanisms, New Pull-Down menu options.
• Drawing Standards: Additional options for display dimensions, geometric tolerances and other drawing entities.
• Sheetmetal Enhancements
  • New Flat Wall Tool. Four default wall shapes, plus all wall creation options in one new dashboard.
  • New Flange Wall Tool. Swept, extruded, and hem wall types consolidated into one dashboard; also includes eight default wall shapes and all other wall options. Single operation to create multiple walls and mitered cuts.
  • Unattached Walls: New user interface for creating unattached flat and extruded walls.
  • Sheetmetal Cut: Consolidated solid and sheetmetal cut tool, now uses new user interface.
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Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0

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**Overview**

After completing this two hour session, you should be able to create Pro/ENGINEER models with embedded information like dimensions, notes, GD&T, and surface finishes. In addition, you learn how to add information related to manufacturing tools, and manufacturing processes. You can use the 3-D Drawings for planning the downstream process map, layouts, and tools at an early stage of the product development cycle. At the end of the session, you will be able to create 3-D drawings compliant to the ASME Y14.41 standard, manage the information added, display and use this information in traditional 2-D drawings if needed.

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 create 3-D drawings using Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

**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.

**Topics**

- Describe the Digital Product Definition.
- Create 3-D drawings using Annotation features.
- Use Annotation features to add notes, dimensions, geometric tolerances, surface finish and manufacturing information.
- Create 2-D drawings using Annotation elements.
- Group and manage the information in 3-D Drawings.
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| Module 1 | Creating 3-D Drawings using Pro/ENGINEER Wildfire 3.0 |
Pro/ENGINEER Wildfire 3.0 has a wide range of tools that support the top-down design process. Simply using these tools in your design does not constitute a top-down design. Top-down design is a process in which a product design is planned, structured and executed on from the top level.

In this course, you will first learn to use tools within Pro/ENGINEER that support top-down design process. Layouts, Skeletons, Publish Geometry and Copy Geometry features are fundamental ingredients to any top-down design process.

After learning how to use these tools, you will begin a top-down design project in which you are part of a project team tasked with developing a home security camera. Using the top-down design plan as your guide, you will complete the design by taking on the roll of various project team members and completing their assigned design activities.

At the end of each day, 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 daily review sessions.

After completing the course, you will have a better understanding of the top-down design process and how it can be implemented using Pro/ENGINEER Wildfire 3.0.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

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.

Topics

- Using top-down design fundamentals.
- Creating and using design layouts.
- Creating and using skeleton models.
- Creating and using copy geometry features.
- Creating and using publish geometry features.
- Creating a top-down assembly structure.
- Using reference controls and investigation tools.
- Planning and documenting a top-down design.
- Distributing design tasks for concurrent development.
- Importing and exporting ECAD data.
- Using display styles.
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Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0

Course Code  | TRN-WBT1833-S
Course Length  | 40 Hours

**Overview**

This course is designed for new users who want to test, validate, and optimize product designs with Pro/ENGINEER Wildfire 3.0’s Mechanica module. Mechanica enables you to simulate structural and thermal loads on product designs. You will also complete comprehensive, hands-on lab exercises that simulate realistic analysis and design optimization activities. Advanced topics such as combined mechanical and thermal analysis techniques are also covered.

After completing the course, you will be able to run engineering analyses and optimizations on your product 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.

**Prerequisites**

- 3 months of Pro/ENGINEER Wildfire 3.0 experience or
- CADTrain Pro/ENGINEER Wildfire 3.0 Mechanica Primer

**Audience**

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

**Topics**

- Simulating Mechanical and Thermal Loads in Machine Parts
- Introduction to the Mechanica Simulation Process
- Modeling Geometry, Material Properties, Loads, and Constraints
- Running Static, Modal, and Thermal Analyses
- Reviewing and Interpreting Results
- Assigning Design Variables
- Running Design Sensitivity and Optimization Analyses
- Best Practices for Simulating Mechanical and Thermal Loads in Machine Parts
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Cabling using Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1849-S
Course Length | 24 Hours

Overview

In this course, you will learn how to create 3-D electrical harnesses using Pro/ENGINEER Wildfire 3.0. This includes using schematic diagrams created with Routed Systems Designer 6.0 to pass information into 3-D harness designs created within Pro/ENGINEER Wildfire 3.0.

You will also learn how to route electrical harnesses, create flattened harnesses for manufacturing, and document harness designs by creating flattened harness drawings that include customized BOM tables and wire list information.

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 3.0.

It is strongly recommended that cabling engineers attend this course and the Creating 2-D Schematics with RSD 6.0 course. This will enable a full understanding of the complete process and an understanding of how the configuration of deliverables from RSD such as wiring diagrams provide essential input for creating electrical harness assemblies in Pro/ENGINEER Wildfire 3.0.

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Creating Schematics using RSD 6.0 (Recommended)

Audience

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

Topics

- Diagram and Harness Development Process Overview
- Creating Wiring Diagrams in Routed Systems Designer (RSD)
- Transferring RSD Information into Harness Designs
- Creating Electrical Harness Assembly Structures
- Routing Wires and Cables in Harness Designs
- Creating Flat Harnesses for Manufacturing
- Creating Harness Drawings
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Milling using Pro/ENGINEER Wildfire 3.0

Course Code | TRN-WBT1853-S
---|---
Course Length | 40 Hours

Overview

In this training course you will learn how to machine products using Pro/ENGINEER Wildfire 3.0 manufacturing tools. This course covers creating tool paths for milling machines. During the course you will learn how to complete each phase of the manufacturing process. You will start by creating the manufacturing model and setting up the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will also create milling sequences and post-process cutter location (CL) data to create machine code.

After successfully 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 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.

Prerequisites

- Completion of TRN-WBT1893-S Introduction to Pro/ENGINEER Wildfire 3.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

This course is intended for manufacturing engineers and NC machinists.

Topics

- Manufacturing process overview.
- Setting up the manufacturing environment.
- Creating and modifying milling sequences.
- Creating and modifying holemaking sequences.
- Using the process manager.
- Duplicating NC sequences.
- Correcting NC sequence failures.
- Post-processing cutter location (CL) data.
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Advanced Component Operations using Pro/ENGINEER Wildfire 3.0

Course Code: TRN-WBT1859-S
Course Length: 2 Hours

Overview

In this course, you will learn how to perform advanced operations on design models. You will learn how to add and remove material in components by using the merge operation. You will also learn how to create new components using mirror and intersect operations. Additionally, you will learn how to create features such as holes and cuts at the assembly level.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to perform advanced operations to create and modify components in Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

Audience

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

Topics

- Add and remove material from components by using the general merge operation.
- Create new components by using intersect and mirror operations.
- Create assembly level features.
# Table of Contents

| Module 1 | Advanced Component Operations |
Creating Style and Warp Features using Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to create sketch-based features with aesthetic and organic shapes by using styled surface features. You will also learn how to alter the form and shape of solids, quilts, facets, and curves by using warp features.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create style and warp features using Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

**Audience**

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

**Topics**

- Create style curves, surfaces and features
- Create warp features to alter design model geometry
# Table of Contents

| Module 1 | Creating Style and Warp Features |
Analyzing and Optimizing Design Models with Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to create various types of analyses including measuring distances, calculating mass properties and interferences, and examining surface curvature and continuity in design models. You will be able to save these analyses as features that update along with changes to the 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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create analysis features and create feasibility and optimization studies using Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

**Audience**

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

**Topics**

- Create various types of design model analyses
- Create analysis features
- Create feasibility and optimization studies
## Table of Contents

| Module 1 | Analyzing and Optimizing Design Models |
Creating Geometry from Surfaces and Curves using Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to use a variety of surface and datum curve tools to duplicate geometry, manipulate existing features, or create new features. You will learn how to duplicate surfaces using copy, offset, move, and mirror tools, and how to edit existing surfaces using the merge, intersect, trim, and extend tools. You will also learn how to create datum curves using the project and wrap tools, and use those datum curves and edges 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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create geometry from surfaces and curves using Pro/ENGINEER Wildfire 3.0.

**Duration**

2 hours

**Prerequisites**

- Introduction to Pro/ENGINEER Wildfire 3.0.
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0.

**Audience**

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

**Topics**

- Duplicate, manipulate, and create surface features.
- Create solid geometry from existing surface features.
Module 1  Creating Geometry from Surfaces and Curves
Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0 - Fundamentals

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Overview

This course is designed for new users who want to test and validate product designs with Pro/ENGINEER Wildfire 3.0’s Mechanica module. Mechanica enables you to simulate structural and thermal loads on product designs. You will also complete comprehensive, hands-on lab exercises that simulate realistic analysis activities.

After completing the course, you will be able to run simple engineering analyses on your product 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.

Prerequisites

- 3 months of Pro/ENGINEER Wildfire 3.0 experience or
- CADTrain Pro/ENGINEER Wildfire 3.0 Mechanica Primer

Audience

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

Topics

- Simulating Mechanical and Thermal Loads in Machine Parts
- Introduction to the Mechanica Simulation Process
- Modeling Geometry, Material Properties, Loads, and Constraints
- Preparing models for analysis.
- Running Static, Modal, and Thermal Analyses
- Reviewing and Interpreting Results
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Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0 - Optimization

Overview

This course is designed for new users who want to increase their understanding of Pro/ENGINEER Wildfire 3.0’s Mechanica module and optimize product designs. Advanced topics such as combined mechanical and thermal analysis techniques, contact analyses, modal analyses, large deformation analyses and dynamic analyses are also covered. This course begins where Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0 - Fundamentals ended and provides you with an enhanced understanding of more advanced Mechanica and analysis topics.

After completing the course, you will be able to run advanced and complex engineering analyses and be able to run sensitivity studies and optimizations on your product 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.

Prerequisites

- 3 months of Pro/ENGINEER Wildfire 3.0 experience or
- CADTrain Pro/ENGINEER Wildfire 3.0 Mechanica Primer
- T1895 - Pro/ENGINEER Mechanica Simulation using Pro/ENGINEER Wildfire 3.0 - Fundamentals

Audience

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

Topics

- Running Static, Modal, and Thermal Analyses
- Reviewing and Interpreting Results
- Assigning Design Variables
- Running Design Sensitivity and Optimization Analyses
- Advanced Topics: Transient Thermal, Contact, Large Deformation, Modal and Dynamic (Time, Frequency, Shock and Random) Analyses.
- Best Practices for Simulating Mechanical and Thermal Loads in Machine Parts
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Milling using Pro/ENGINEER Wildfire 3.0 - Setting Up

Overview

In this training course you will learn how to set up for machining products using Pro/ENGINEER Wildfire 3.0 manufacturing tools.

You will start by creating the manufacturing model and setting up the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will also review milling sequences and review cutter location (CL) data.

After successfully completing the course, you will be able to create manufacturing models and set up the manufacturing environment.

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.

Prerequisites

- Completion of TRN-WBT1893-S Introduction to Pro/ENGINEER Wildfire 3.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

This course is intended for manufacturing engineers and NC machinists.

Topics

- Manufacturing process overview.
- Setting up the manufacturing environment.
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Milling using Pro/ENGINEER Wildfire 3.0 - Creating NC Data

Course Code  TRN-WBT1898-S
Course Length  16 Hours

Overview

In this training course, you will learn how to machine products using Pro/ENGINEER Wildfire 3.0 manufacturing tools. This course covers creating tool paths for milling machines.

During the course, you will create face milling sequences, profile milling sequences, volume milling sequences, local milling sequences, surface milling sequences, trajectory milling sequences, roughing sequences, re-roughing sequences, finishing sequences and holemaking sequences.

After successfully completing the course, you will be able to create milling sequences and holemaking sequences.

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.

Prerequisites

- Completion of TRN-WBT1893-S Introduction to Pro/ENGINEER Wildfire 3.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

This course is intended for manufacturing engineers and NC machinists.

Topics

- Creating and modifying milling sequences.
- Creating and modifying holemaking sequences
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Milling using Pro/ENGINEER Wildfire 3.0 - Managing NC Data

Course Code | TRN-WBT1899-S
---|---
Course Length | 12 Hours

Overview

In this training course you will learn how to manage NC data created using Pro/ENGINEER Wildfire 3.0 manufacturing tools.

You will learn how to use the process manager to manage NC data. You will learn how to post-process cutter location (CL) data and you will learn how to correct NC sequence failures. You will also learn how to use different tools to duplicate NC sequences.

After successfully completing the course, you will be able to use different tools to manage and configure NC data created using Pro/ENGINEER Wildfire 3.0.

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.

Prerequisites

- Completion of TRN-WBT1893-S Introduction to Pro/ENGINEER Wildfire 3.0 – Fundamentals (Web Based Training) or equivalent experience.

Audience

This course is intended for manufacturing engineers and NC machinists.

Topics

- Using the process manager.
- Duplicating NC sequences.
- Correcting NC sequence failures.
- Post-processing cutter location (CL) data.
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Using Component Interfaces and Flexible Components in Pro/ENGINEER Wildfire 3.0

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Overview

In this course, you will learn how to create component interfaces to quickly assemble components within an assembly, both manually and automatically. Flexible components will enable you to assemble the same component multiple times while varying dimensions as each component is added.

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 take an assessment using your PTC University account.

After completing this course, you will be well-prepared to create component interfaces and flexible components using Pro/ENGINEER Wildfire 3.0.

Duration

2 hours

Prerequisites

- Introduction to Pro/ENGINEER Wildfire 3.0
- Pro/ENGINEER Wildfire 3.0 Update from Pro/ENGINEER Wildfire 2.0

Audience

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

Topics

- Assemble components using component interfaces
- Assemble flexible components
- Describe how parent/child relationships affect assembly components
## Table of Contents

| Module 1 | Using Component Interfaces and Flexible Components |
Reverse Engineering using Pro/ENGINEER Wildfire 3.0

Course Code: TRN-WBT1891-S  
Course Length: 16 Hours

Overview

In this course, you will learn how to use the Reverse Engineering Extension (REX) functions in Pro/ENGINEER Wildfire 3.0. You will learn the capabilities, workflow, and some best practices for undertaking a Reverse Engineering effort with Pro/ENGINEER. You will learn how to import point cloud data and facet data into the Pro/ENGINEER environment and turn this data into a working Pro/ENGINEER model. The course covers the following topics as well as others not listed here: command and menu structure, retrieving and manipulating point data, processing and cleaning data, wrap and facet model creation. You will also learn about the Restyle feature, surface creation on facet models (up to and including G2 surfaces), analysis tools and automatic curve/patch generation.

You will complete Pro/FICIENCY skills assessment questions for each module at the end of the course. These questions are used to help reinforce your understanding of the course topics and form the basis for daily review sessions.

After completing the course, you will be prepared to complete Reverse Engineering tasks and activities at your company.

Prerequisites

- Basic Pro/Engineer Wildfire skills (such as those acquired in Introduction to Pro/ENGINEER Wildfire 3.0).
- Advantageous but not required: surfacing skills in Pro/ENGINEER

Audience

This course is intended for all users with an interest in Reverse Engineering as a general topic and those wishing to use REX in Pro/ENGINEER Wildfire 3.0. The topics in this course are also available as Web-based training courses.

Topics

- REX capabilities in Pro/ENGINEER Wildfire 3.0
- Reverse Engineering workflow in Pro/ENGINEER Wildfire 3.0 (the four phases)
- Command/Menu structure
- Importing/Editing point cloud data
- Wrap model creation
- Facet model creation
- The Restyle feature
- Surface creation on facet models (up to and including G2 surfaces)
- Analysis tools
- Automatic curve/patch generation
- Reverse Engineering best practices
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Introduction to Routed Systems Designer 6.0 for Pro/ENGINEER Wildfire 3.0

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**Overview**

In this course, you will learn how to use Routed Systems Designer (RSD) to create schematic diagrams for both electrical cabling and piping designs. You will learn how to create block diagrams to represent initial system designs. You will also learn how to create electrical circuit and wiring diagrams, and how to create process and instrumentation diagrams (P&ID) for piping systems. Finally, you will learn how to use these diagrams to configure 3-D cabling designs and industrial piping designs created within Pro/ENGINEER Wildfire 3.0.

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.

**Prerequisites**

- None

**Audience**

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

**Topics**

- Overview of the 2-D Schematic Design Process
- Setting up the Design
- Setting up the Catalog Library
- Creating Block Diagrams
- Creating Circuit Diagrams
- Creating Wiring Diagrams
- Creating Process and Instrumentation Diagrams (P&ID)
- Communicating Diagram Information to Pro/ENGINEER Wildfire
- Creating Reports and Other Information
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