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creo™ 1.0

Curriculum Guide

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Live Classroom Curriculum Guide

- Update to Creo Parametric from Creo Elements/Pro 5.0
- Update to Creo Parametric from Pro/ENGINEER Wildfire 4.0
- Introduction to Creo Parametric
- Advanced Modeling using Creo Parametric
- Advanced Assembly Design using Creo Parametric
- Detailing using Creo Parametric
- Surfacing using Creo Parametric
- Sheetmetal Design using Creo Parametric
- Introduction to Creo Simulate
- Introduction to Creo Illustrate
- Flexible Modeling using Creo Parametric

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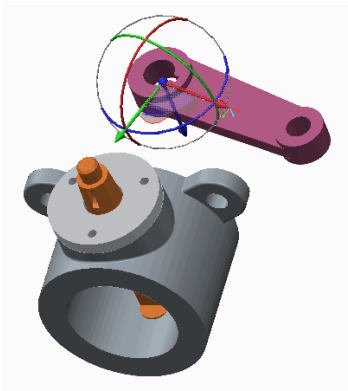
Update to Creo Parametric from Creo Elements/Pro 5.0

Overview

Course Code	TRN-3400-T
Course Length	1 Day

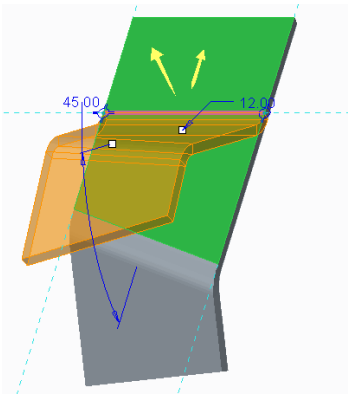


In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 1.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. Next, you will become familiar with the Sketcher workflow and reference enhancements. Part modeling enhancements to features such as Extrude, Corner Chamfer, Sweeps, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities, such as selecting multiple components, the new relationship constraints, and enhancements for dragging components. Next, you will examine the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode, you will learn to use the many updated tools such as Wall, Bend, and Relief, as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

- Utilize the Interface enhancements in Creo Parametric
- Utilize the Sketcher enhancements in Creo Parametric
- Utilize the Modeling enhancements in Creo Parametric
- Utilize the Assembly enhancements in Creo Parametric
- Utilize the Drawing enhancements in Creo Parametric
- Utilize the Sheetmetal enhancements in Creo Parametric



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 5.0, or equivalent experience with Pro/ENGINEER Wildfire 5.0 or Creo Elements/Pro 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

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

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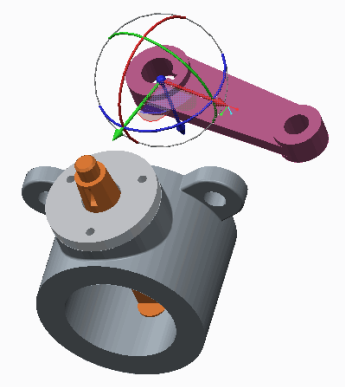
Update to Creo Parametric from Pro/ENGINEER Wildfire 4.0

Overview

Course Code	TRN-3401-T
Course Length	2 Days

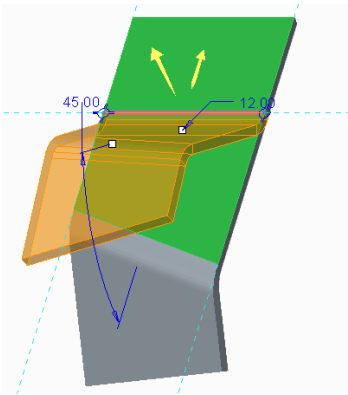


In this course, you will learn how to utilize the core functionality enhancements in Creo Parametric 1.0. First, you will become familiar with using and customizing the new ribbon interface in Creo Parametric. Next, you will study the Sketcher workflow and reference enhancements, as well as Sketcher constraint, geometry, and diagnostics enhancements. Part modeling enhancements to features such as Extrude, Trajectory Rib, Point Pattern, Corner Chamfer, Sweeps, UDFs, and Datum Curves will then be examined. You will also learn about new and enhanced Assembly capabilities such as selecting multiple components, the new relationship constraints, enhancements for dragging components, explode enhancements, simplified rep enhancements, and dynamic gear enhancements. Then you will learn how to identify and resolve part and assembly failures on-the-fly, without accessing Resolve mode. Next, you will examine the new Drawing tree and drawing sheets tabs, as well as showing annotations. Also in Drawing mode, you will learn the new Table and Balloon functionality for 2-D drawings, and review various detailing enhancements. Finally, in Sheetmetal mode you will learn to use the many updated tools such as Wall, Bend, and Relief; as well as the consolidated Flat Pattern tool and configuring Sheetmetal properties.

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

- Utilize the Interface enhancements in Creo Parametric
- Utilize the Sketcher enhancements in Creo Parametric
- Utilize the Modeling enhancements in Creo Parametric
- Utilize the Assembly enhancements in Creo Parametric
- Utilize the Drawing enhancements in Creo Parametric
- Utilize the Sheetmetal enhancements in Creo Parametric



Prerequisites

- Introduction to Pro/ENGINEER Wildfire 4.0, or equivalent experience with Pro/ENGINEER Wildfire 4.0 or Creo Elements/Pro 4.0

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

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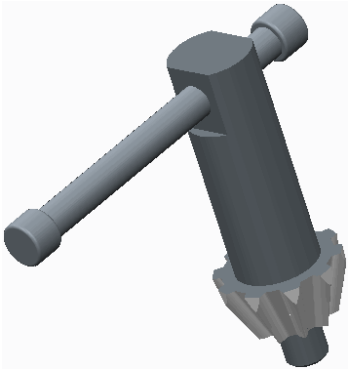
Introduction to Creo Parametric

Overview

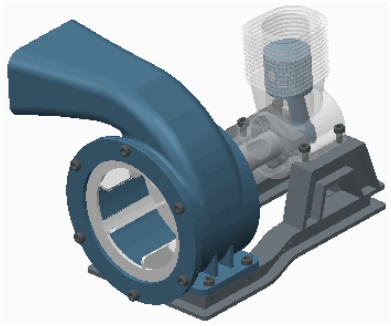
Course Code	TRN-3402–T
Course Length	5 Days



This course is designed for new users who want to become proficient with Creo Parametric 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 Creo Parametric.

Course Objectives



- Learning the basic Creo Parametric modeling process
- Understanding Creo Parametric concepts
- Learning how to use the Creo Parametric interface
- Selecting and editing geometry, features, and models
- Sketching geometry and using tools
- Creating sketches for features
- Creating datum planes and datum axes
- Creating extrudes, revolves, and profile 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 Creo Parametric Basic Modeling Process
Module	2	Understanding Creo Parametric Concepts
Module	3	Using the Creo Parametric Interface
Module	4	Selecting Geometry, Features, and Models
Module	5	Editing Geometry, Features, and Models
Module	6	Creating Sketcher Geometry

Day 2

Module	7	Using Sketcher Tools
Module	8	Creating Sketches for Features
Module	9	Creating Datum Features: Planes and Axes
Module	10	Creating Extrudes, Revolves, and Ribs
Module	11	Utilizing Internal Sketches and Embedded Datums
Module	12	Creating Sweeps and Blends

Day 3

Module	13	Creating Holes, Shells, and Draft
Module	14	Creating Rounds and Chamfers
Module	15	Project I
Module	16	Group, Copy, and Mirror Tools
Module	17	Creating Patterns
Module	18	Measuring and Inspecting Models

Day 4

Module	19	Assembling with Constraints
Module	20	Assembling with Connections
Module	21	Exploding Assemblies
Module	22	Drawing Layout and Views
Module	23	Creating Drawing Annotations
Module	24	Using Layers

Day 5

Module	25	Investigating Parent/Child Relationships
Module	26	Capturing and Managing Design Intent
Module	27	Resolving Failures and Seeking Help
Module	28	Project II

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Advanced Modeling using Creo Parametric

Overview

Course Code	TRN-3403–T
Course Length	3 Days

The Advanced Part Modeling using Creo Parametric training course teaches you how to use advanced part modeling techniques in Creo Parametric 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. After completing this course, you will be well prepared to work efficiently with complex product designs using Creo Parametric.

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 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 sweeps with variable sections
- 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 Creo Parametric
- Update to Creo Parametric from Creo Elements/Pro 5.0

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	Sweeps with Variable Sections
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

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Advanced Assembly Design using Creo Parametric

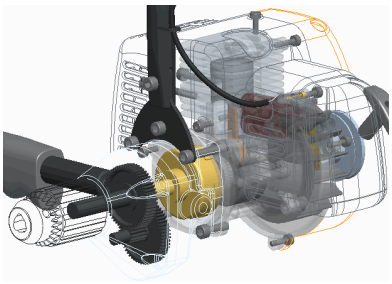
Overview

Course Code	TRN-3404–T
Course Length	3 Days

In this course, you will learn how to use Creo Parametric 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 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



- 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

Prerequisites

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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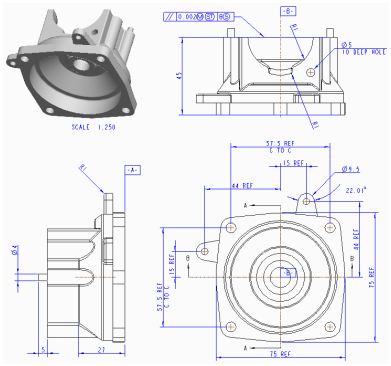
Detailing using Creo Parametric

Overview

Course Code	TRN-3405–T
Course Length	3 Days

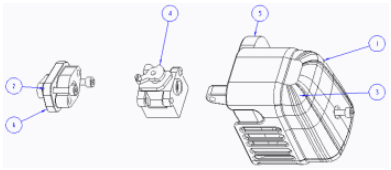


Detailing with Creo Parametric is a comprehensive training course that teaches you how to quickly create detailed drawings using information captured within 3-D design models. In this course, you will learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and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

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

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



- 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 Creo Parametric

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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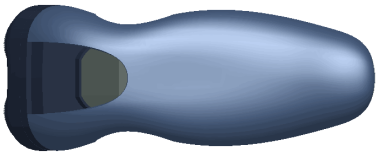
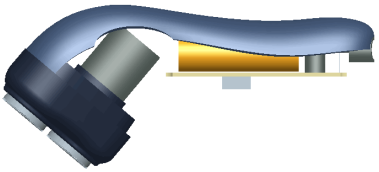
Surfacing using Creo Parametric

Overview

Course Code	TRN-3406-T
Course Length	3 Days

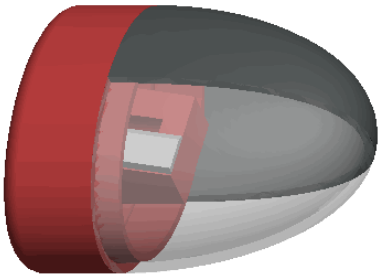




In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how manipulate surfaces using editing tools, and analyze surfaces for quality and desired characteristics. In addition, you will learn how to create solid features using the surfaces as references. After completing this course, you will be well prepared to create complex shaped models using surfaces in Creo Parametric.

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 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 toCreo Parametric
- Update to Creo Parametric from Creo Elements/Pro 5.0

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	Sweep Surfaces with Variable Sections
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

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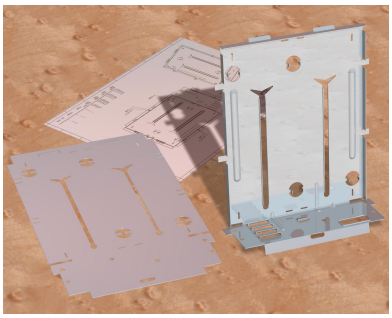
Sheetmetal Design using Creo Parametric

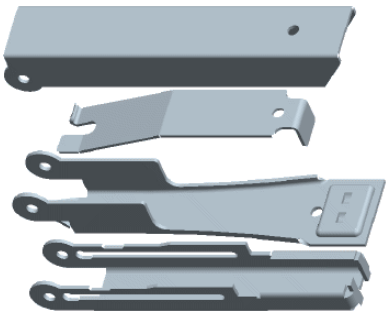
Overview

Course Code	TRN-3407-T
Course Length	2 days



Sheetmetal Design using Creo Parametric is a comprehensive training course that teaches you how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 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 Creo Parametric

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 Creo Parametric Sheetmetal Design Process
Module	2	Sheetmetal Model Fundamentals
Module	3	Creating Primary Sheetmetal Wall Features
Module	4	Creating Secondary Sheetmetal 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

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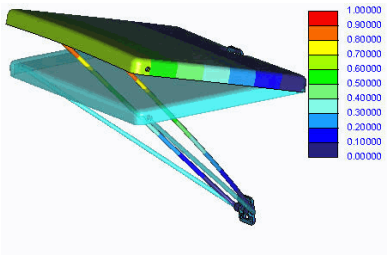
Introduction to Creo Simulate

Overview

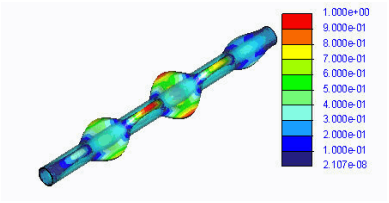
Course Code	TRN-3411–T
Course Length	5 Days



This course is designed for new users who want to test, validate, and optimize product designs with the Creo Simulate module. Simulate 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 be introduced to advanced topics such as dynamic analyses, combined mechanical and thermal analyses, and Optimization Studies. 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 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



- Learning the basic Simulate analysis process
- Learning theory and simulate model topics
- Exploring results
- Learning about materials and material properties
- Understanding and using Simulate idealizations
- Understanding and using structural loads
- Understanding and using structural constraints
- Running structural analyses
- Understanding convergence
- Analyzing assemblies with Simulate
- Completing design and sensitivity studies
- Running optimization studies

Prerequisites

- Three months of Pro/ENGINEER Wildfire 5.0, or Creo Parametric 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	Introduction to Creo Simulate
Module	2	Theoretical Foundations
Module	3	Model Preparation
Module	4	Analysis Definition Basics
Module	5	Introduction to Results Evaluation

Day 2

Module	6	Materials and Simulate Geometry Features
Module	7	Loads and Constraints
Module	8	Interfaces, Assemblies, and Measures

Day 3

Module	9	Meshing
Module	10	More Analysis Types
Module	11	Singularities
Module	12	Basic Model Debugging
Module	13	Project

Day 4

Module	14	Model Types
Module	15	Shells
Module	16	Idealizations

Day 5

Module	17	Advanced Analysis
Module	18	Sensitivity and Optimization

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Introduction to Creo Illustrate

Overview

Course Code	TRN-3414–T
Course Length	1 Day

In this course, you will learn about Creo Illustrate and its role as a purpose-built, role-based solution for creating 3-D technical illustrations. You will learn how to automatically create technical illustrations from existing 3-D CAD data. You will also learn how to map existing eBOM data to populate an illustration specific sBOM. In addition, you will learn how to manipulate imported 3-D viewables to create figures and animations to create service information content including:

- Service Procedures
- Parts Identification
- Training Materials
- Product Assembly and Disassembly

Finally, you will learn how to create markup and annotations with figures and animations.

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

- List and describe the primary features and benefits of Creo Illustrate
- Demonstrate fundamental Creo Illustrate illustration and figure creation steps to produce an illustration-specific sBOM from imported 3-D CAD data
- List and describe methods and tools used to create exploded views, create section cuts, and remove graphic objects from an illustration
- Modify illustrations by adding rendering styles, color, and work with the Creo Illustrate 3-D symbols library
- Create and manage a parts list from the Creo Illustrate sBOM
- Create parts list callouts and annotations in illustration figures
- Use the Creo Illustrate Animator tools to create animated illustrations
- Save, export, and publish illustrations

Prerequisites

- Students should be familiar with Windows-based file systems and mouse operations
- Students should have some familiarity with creating 2-D and 3-D illustrations from CAD data sources

Audience

- Technical publications illustrators
- Technical publications authors
- Training authors
- Manufacturing instructors
- Users responsible for parts definition, technical marketing, and service planning

Agenda

Day 1

Module	1	Introduction to the Creo Illustrate User Interface
Module	2	Create an Illustration and Work with Figures
Module	3	Deconstruct a Figure
Module	4	Creo Illustrate Animations
Module	5	Create and Manage Annotations, Sub-Assemblies, and Parts Lists
Module	6	Publishing and Exporting 3-D Illustrations

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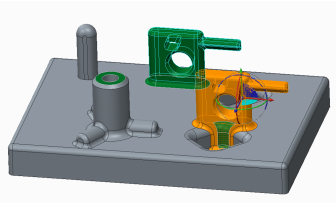
Flexible Modeling using Creo Parametric

Overview

Course Code	TRN-3415–T
Course Length	1 Day

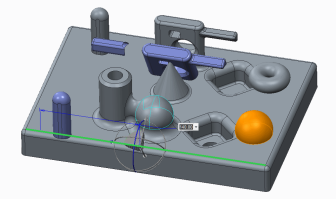


In this course, you will learn how to use flexible modeling tools to edit existing geometry on parametric models. The flexible modeling process typically involves initially selecting model surfaces, then refining the selected surface set using smart selection tools, and finally modifying the selected geometry by applying transformation tools, patterning tools, or symmetry tools. Each stage of the process is described in detail and supported by step-by-step exercises.

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 flexible modeling basics
- Applying selection and tools
- Utilizing editing and transformations
- Working with recognition
- Using propagation and other editing features



Prerequisites

- Introduction to Creo Parametric 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.

Agenda

Module	1	Introduction to Flexible Modeling
Module	2	Selection and Tools
Module	3	Editing and Transformations
Module	4	Recognition
Module	5	Propagation and Other Editing Features