PTC® University

Creo 3.0

Curriculum Guide

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Live Classroom Curriculum Guide

- Update to Creo Parametric 3.0 from Creo Parametric 2.0
- Introduction to Creo Parametric 3.0
- Advanced Modeling using Creo Parametric 3.0
- Advanced Assembly Design using Creo Parametric 3.0
- Introduction to Creo Simulate 3.0
- Detailing using Creo Parametric 3.0
- Surfacing using Creo Parametric 3.0
- Sheetmetal using Creo Parametric 3.0
- Milling using Creo Parametric 3.0
- Introduction to Creo Direct 3.0
- Flexible Modeling using Creo Parametric 3.0
- Introduction to Creo Layout 3.0
- Introduction to Creo Options Modeler 3.0
- Piping using Creo Parametric 3.0
- Behavioral Modeling using Creo Parametric 3.0

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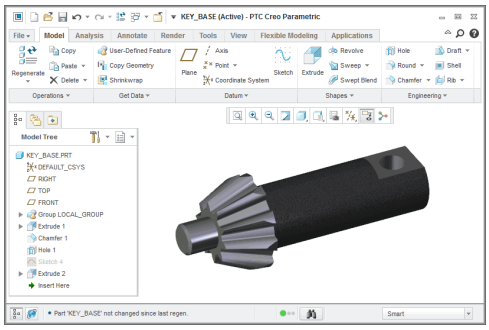
Update to Creo Parametric 3.0 from Creo Parametric 2.0

Overview

Course Code	TRN-4500-T
Course Length	1 Day

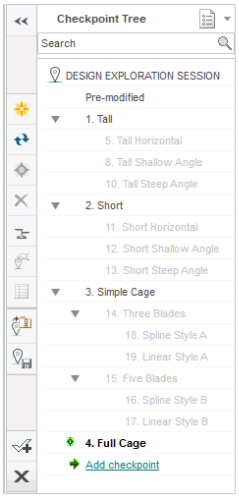


In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric 3.0. You will be introduced to user interface appearance enhancements such as the active window and right-click menu enhancements. You will examine the Part Modeling enhancements to features such as Draft, Pattern, UDF, Rounds, and Spinal Bend. You will experiment with the new Edit References dialog box on part models and study the surfacing enhancements to surface copy, flatten quilt, boundary blends, and the new connection analysis tool. Freestyle surfacing enhancements to the Join tool and aligning to geometry will also be covered. You will also learn about Flexible Modeling enhancements to tangency propagation, flexible patterns, round/chamfer recognition, and the flexible move tool. You will investigate the new Assembly capabilities such as the built-in hardware library and the Intelligent Fastener extension, as well as enhanced Assembly functionality such as heterogeneous (multi-CAD) assembly and the Design Exploration extension. You will examine the new Dimension and Note functionality for 2-D drawings and review various detailing enhancements. Finally, you will learn to use the many updated tools in Sheetmetal mode such as Die Forms, Bend Relief, and Rip, as well as the enhancement for bending in multiple planes.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Use the Interface enhancements in Creo Parametric 3.0.
- Use the Part Modeling enhancements in Creo Parametric 3.0.
- Use the Surfacing enhancements in Creo Parametric 3.0.
- Use the Flexible Modeling enhancements in Creo Parametric 3.0.
- Use the Intelligent Fastener extension in Creo Parametric 3.0.
- Use the Design Exploration extension in Creo Parametric 3.0.
- Use the Drawing enhancements in Creo Parametric 3.0.
- Use the Sheetmetal enhancements in Creo Parametric 3.0.



Design Exploration session : Full Cage
Session name: FAN_DEX

Prerequisites

- Introduction to Creo Parametric 2.0, or equivalent experience with Creo Parametric 2.0

Audience

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

Agenda

Day 1

Module	1	Interface Enhancements
Module	2	Part Modeling Enhancements
Module	3	Surfacing Enhancements
Module	4	Flexible Modeling Enhancements
Module	5	Utilizing Intelligent Fasteners
Module	6	Utilizing Design Exploration
Module	7	Drawing Enhancements
Module	8	Sheetmetal Enhancements

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Introduction to Creo Parametric 3.0

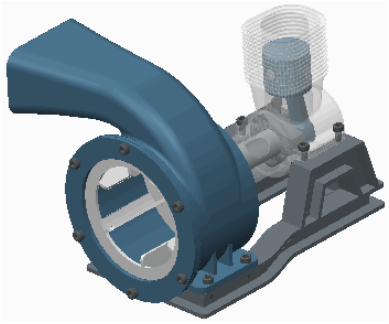
Overview

Course Code	TRN-4501-T
Course Length	5 Days

In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 3.0. 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 Creo Parametric 3.0.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Learn the basic Creo Parametric modeling process
- Understand Creo Parametric concepts
- Learn how to use the Creo Parametric interface
- Select and edit geometry, features, and models
- Sketch geometry and use tools
- Create sketches for features
- Create datum planes and datum axes
- Create extrudes, revolves, and profile ribs
- Utilize internal sketches and embedded datums
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Group, copy, and mirror items
- Create patterns
- Measure and inspect models
- Assemble with constraints
- Assemble with connections
- Explode assemblies
- Lay out drawings and create views
- Create drawing annotations

- Use layers
- Investigate parent/child relationships
- Capture and manage design intent
- Resolve failures and seek 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	Sketcher Workflow
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 3.0

Overview

Course Code	TRN-4502-T
Course Length	3 Days

The Advanced Modeling using Creo Parametric 3.0 training course teaches you how to use advanced part modeling techniques 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 3.0.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- 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 advanced reference management techniques
- Create family tables
- Reuse features
- Learn advanced copy techniques
- Create advanced patterns

Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

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 3.0

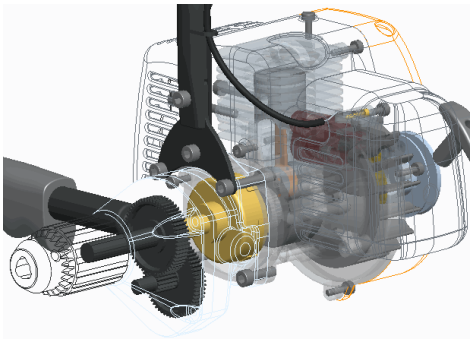
Overview

Course Code	TRN-4503-T
Course Length	3 Days

In this course, you will learn how to use Creo Parametric 3.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain designs, 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, which are both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Use advanced component selection
- Use advanced assembly constraints
- Create and use component interfaces
- Utilize intelligent fasteners
- Create and use flexible components
- Restructure and mirror assemblies
- Use assembly features and shrinkwrap
- Replace components in an assembly
- Understand the basics of simplified reps
- Create cross-sections, display styles, and combined views
- Substitute components by reps, envelopes, and simplified reps
- Understand advanced simplified rep functionality
- Create and use assembly structure and skeletons
- Utilize design exploration

Prerequisites

- Introduction to Creo Parametric 3.0
- Update to Creo Parametric 3.0 from Creo Parametric 2.0

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	Advanced Component Selection
Module	2	Using Advanced Assembly Constraints
Module	3	Creating and Using Component Interfaces
Module	4	Utilizing Intelligent Fasteners
Module	5	Creating and Using Flexible Components
Module	6	Restructuring and Mirroring Assemblies

Day 2

Module	7	Using Assembly Features and Shrinkwrap
Module	8	Replacing Components in an Assembly
Module	9	Understanding the Basics of Simplified Reps
Module	10	Creating Cross-Sections, Display Styles, Layer States, and Combined Views

Day 3

Module	11	Substituting Components using User Defined, Envelopes, and Simplified Reps
Module	12	Understanding Advanced Simplified Rep Functionality
Module	13	Creating and Using Assembly Structure and Skeletons
Module	14	Utilizing Design Exploration
Module	15	Project

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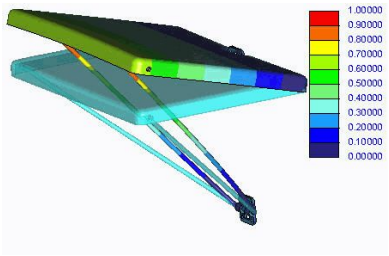
Introduction to Creo Simulate 3.0

Overview

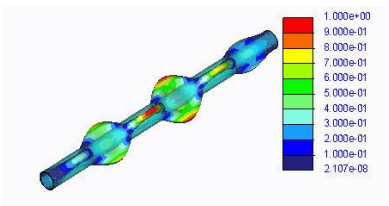
Course Code	TRN-4504-T
Course Length	5 Days



In this course, you will learn how to test, validate, and optimize product designs with the Creo Simulate module. Creo Simulate enables you to simulate structural and thermal loads on product designs. 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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Understand the basic Simulate analysis process
- Understand theory and simulate model topics
- Explore results
- Explore materials and material properties
- Understand and use Simulate idealizations
- Understand and use structural loads
- Understand and use structural constraints
- Run structural analyses
- Understand convergence
- Analyze assemblies with Simulate
- Complete design and sensitivity studies
- Run optimization studies
- Understand the basics of Thermal analysis

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	Simulation Models
Module	4	Materials and Material Properties
Module	5	Structural Constraints
Module	6	Structural Loads

Day 2

Module	7	Meshing
Module	8	Convergence
Module	9	Structural Analysis
Module	10	Introduction to Results Evaluation

Day 3

Module	11	Refining the Design
Module	12	Basic Model Debugging
Module	13	Singularities

Day 4

Module	14	Analyzing Assemblies
Module	15	Shells
Module	16	Idealizations

Day 5

Module	17	Thermal Analysis
Module	18	Advanced Analysis
Module	19	Project

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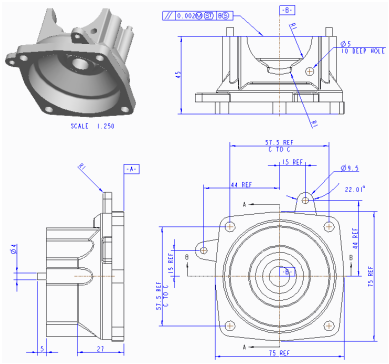
Detailing using Creo Parametric 3.0

Overview

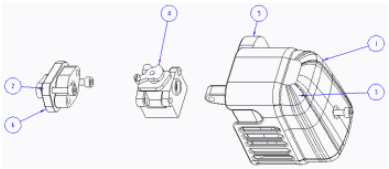
Course Code	TRN-4505-T
Course Length	3 Days



In this course, you will learn how to quickly create detailed drawings using information captured within 3-D design models. You will also learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and associative nature of Creo Parametric 3.0 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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- 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 3.0

Audience

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

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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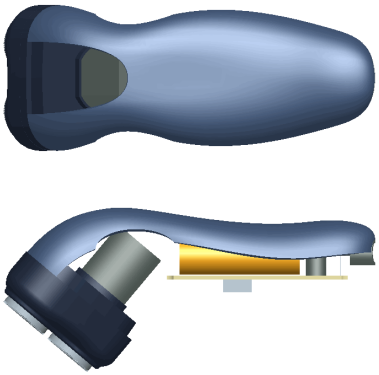
Surfacing using Creo Parametric 3.0

Overview

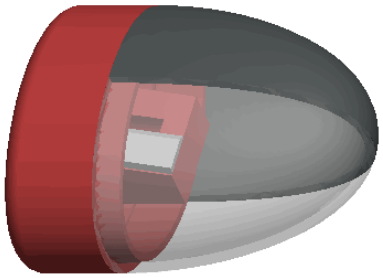
Course Code	TRN-4506-T
Course Length	3 Days



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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



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

Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

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	Sweep Surfaces with Variable Sections
Module	8	Helical Sweeps
Module	9	Swept Blends
Module	10	Analyzing Surface Curvature
Module	11	Additional Surface Analysis Tools

Day 3

Module	12	Extending and Trimming Surfaces
Module	13	Manipulating Surfaces
Module	14	Creating and Editing Solids using Quilts
Module	15	Master Model Technique
Module	16	Project

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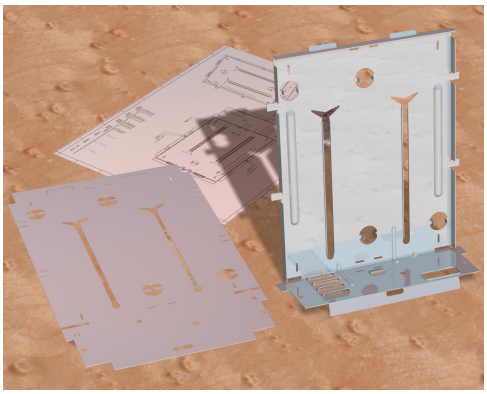
Sheetmetal using Creo Parametric 3.0

Overview

Course Code	TRN-4507-T
Course Length	2 Days

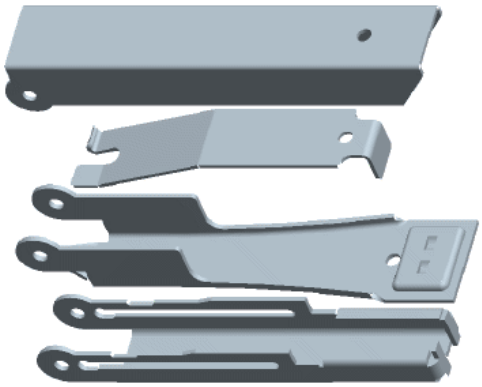


In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 3.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including 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 module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Create, convert, and display the sheetmetal model
- Use methods of developed length calculation
- Use primary and secondary wall features, as well as partial walls
- Use bend relief
- Use unbend and bend back features
- Apply sheetmetal bend features
- Use flat patterns
- Create sheetmetal cuts
- Create forms
- Use notch and punch features
- Utilize the sheetmetal environment setup, sheetmetal design information tools, and sheetmetal design rules
- Detail sheetmetal designs



Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

Agenda

Day 1

Module	1	Introduction to the 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	Bending and Unbending Sheetmetal Models
Module	6	Sheetmetal Form Features
Module	7	Modifying Sheetmetal Models
Module	8	Sheetmetal Setup and Tools
Module	9	Detailing Sheetmetal Designs
Module	10	Design Project

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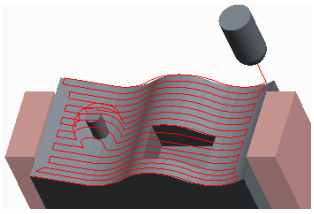
Milling using Creo Parametric 3.0

Overview

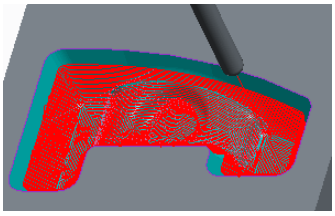
Course Code	TRN-4508-T
Course Length	5 Days



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

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Understand the manufacturing process
- Create and configure manufacturing models
- Configure the manufacturing environment
- Create and modify milling sequences
- Create and modify holemaking sequences
- Use the process manager to create NC sequences
- Post-process cutter location (CL) data

Prerequisites

- Introduction to Creo Parametric – Fundamentals (Web Based Training) or equivalent experience

Audience

- This course is intended for manufacturing engineers and NC machinists

Agenda

Day 1

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

Day 2

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

Day 3

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

Day 4

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

Day 5

Module	20	Creating Trajectory Milling Sequences
Module	21	Creating Holemaking Sequences
Module	22	Creating Engraving Sequences

Module 23 Using the Process Manager

Module 24 Creating and Post-Processing CL Data Files

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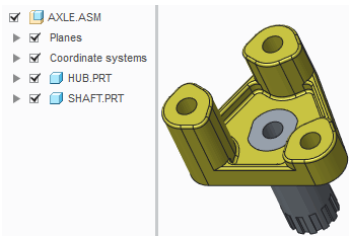
Introduction to Creo Direct 3.0

Overview

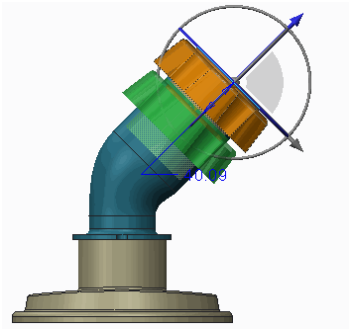
Course Code	TRN-4510-T
Course Length	1 Day



In this course, you will learn direct modeling using Creo Direct 3.0. You will become familiar with Creo Direct's interface and approach to direct modeling, including how to quickly create sketches with precision, transform sketches into 3-D shapes, and directly manipulate existing geometry with ease. You will also learn how to assemble and reposition components in an assembly, as well as use existing geometry in an assembly to create part geometry.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Understand the Direct Modeling approach to creating geometry
- Sketch precise 2-D geometry without constraints
- Create 3-D shapes from the 2-D geometry by extruding, revolving, and sweeping sketches and sketch regions
- Create datum axes and datum planes
- Create engineering geometry, including holes, rounds, chamfers, draft, and shells
- Pattern geometry in one and two directions
- Use measure tools and create part and assembly cross-sections
- Rapidly select and directly modify 3-D shapes in your models
- Understand the direct approach to creating assemblies
- Edit component position in an assembly
- Create new components and design components in the context of an assembly

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 will also benefit from taking this course.

Agenda

Day 1

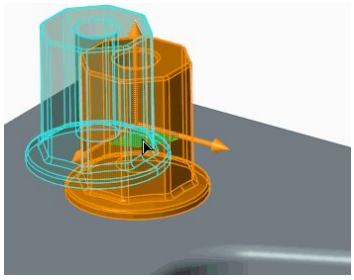
Module	1	Introduction to Creo Direct
Module	2	Creating Sketches in 2-D Mode
Module	3	Creating Geometry Shapes
Module	4	Creating Datums
Module	5	Creating Engineering Geometry
Module	6	Measurements and Sections
Module	7	Editing 3-D Geometry
Module	8	Working with Assemblies

PTC® University

Flexible Modeling using Creo Parametric 3.0

Overview

Course Code	TRN-4511-T
Course Length	1 Day

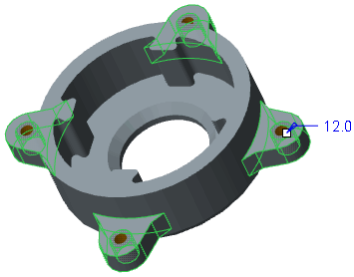


In this course, you will learn how to use Flexible Modeling tools to edit existing geometry on parametric models. The Flexible Modeling process typically involves initially selecting model surfaces, then refining the selected surface set using smart selection tools, and finally modifying the selected geometry by applying transformation tools, patterning tools, or symmetry tools.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Understand Flexible Modeling basics
- Apply shape surface selection
- Perform flexible move operations on geometry
- Perform flexible transform operations on geometry
- Use the various transform options
- Attach and remove geometry
- Understand and recognize how the system handles rounds and chamfers
- Create and recognize patterns and symmetry, and propagate changes



Prerequisites

- Introduction to Creo Parametric 3.0 or equivalent experience

Audience

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

Agenda

Day 1

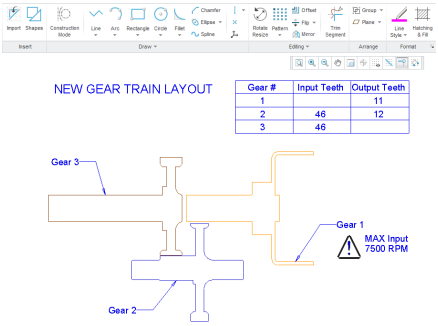
Module	1	Introduction to Flexible Modeling
Module	2	Shape Surface Selection
Module	3	Flexible Move
Module	4	Flexible Transforms
Module	5	Transform Options
Module	6	Attaching and Removing Geometry
Module	7	Rounds and Chamfers
Module	8	Patterns and Symmetry

PTC® University

Introduction to Creo Layout 3.0

Overview

Course Code	TRN-4512-T
Course Length	1 Day

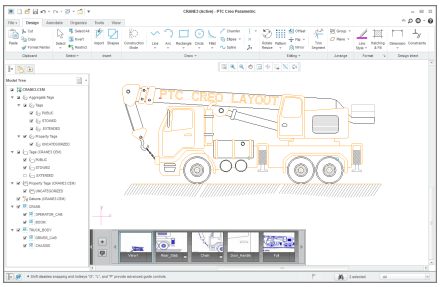


In this course, you will learn about Layout, a Creo module that is used for creating unconstrained 2-D designs that can be leveraged in 3-D models. You will investigate the concepts behind Creo Layout, a typical workflow, and the user interface. You will also learn how Layout utilizes precision panels and sketching guides to intelligently sketch a variety of 2-D geometry, as well as learn how to manipulate, organize, and import layout geometry. Finally, you will learn how to leverage 2-D Layout designs in 3-D models.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Understand Layout sketching methodology
- Sketch Layout geometry
- Manipulate Layout geometry
- Organize Layout geometry
- Import Layout geometry
- Annotate Layouts
- Utilize Layouts in 3-D models



Prerequisites

- Introduction to Creo Parametric 3.0 (optional)

Audience

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

Agenda

Day 1

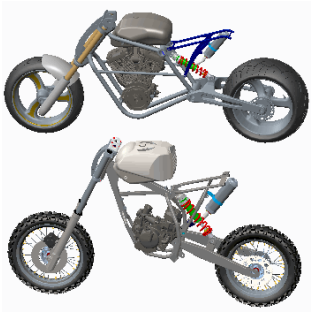
Module	1	Introduction to Layout
Module	2	Layout Sketching Methodology
Module	3	Sketching Layout Geometry
Module	4	Manipulating Layout Geometry
Module	5	Organizing Layout Geometry
Module	6	Importing Layout Geometry
Module	7	Annotating Layouts
Module	8	Utilizing Layouts in 3-D Models

PTC® University

Introduction to Creo Options Modeler 3.0

Overview

Course Code	TRN-4513-T
Course Length	1 Day

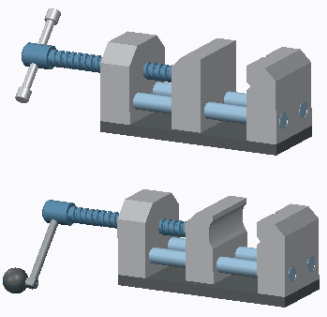


In this course, you will learn how to use Creo Options Modeler to create configurable assemblies. You will begin by exploring a completed configurable assembly that has multiple variations created. You will then learn how to establish interchangeability between components, and how to create and assemble configurable products, configurable modules, and module variants. Then, you will become familiar with the creation of options and choices, which will enable you to explore usage of the Variant Builder to configure assembly variants. Finally, you learn how to leverage an existing design assembly and develop a configurable product assembly containing multiple design variations.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Understand and use Creo Options Modeler
- Create interchange assemblies
- Create configurable modules and products
- Define options and choices
- Utilize configurable assemblies
- Understand how to manage large Options Modeler assemblies with and without Windchill



Prerequisites

- Introduction to Creo Parametric 3.0
- Advanced Assembly Design using Creo Parametric 3.0

Audience

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

Agenda

Day 1

Module	1	Introduction to Creo Options Modeler
Module	2	Creating Interchange Assemblies
Module	3	Creating Configurable Modules and Products
Module	4	Defining Options and Choices
Module	5	Utilizing Configurable Assemblies
Module	6	The Options Modeler with Very Large Assemblies and Windchill
Module	7	Project

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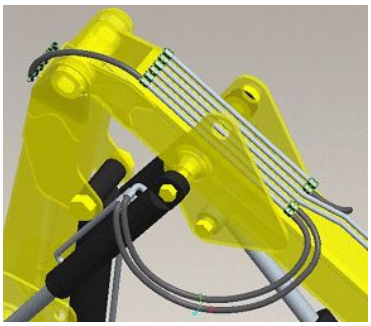
Piping using Creo Parametric 3.0

Overview

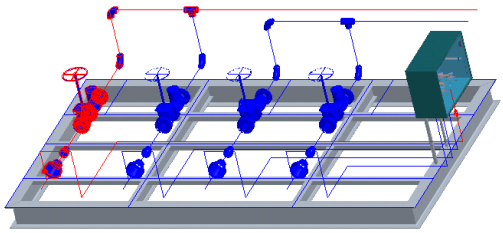
Course Code	TRN-4516-T
Course Length	3 Days



In this course, you will learn how to manually create (non-specification-driven) mechanical piping designs using Creo Parametric 3.0. This includes learning how to configure pipelines, route pipelines, and insert pipe fittings such as valves and reducers. You will also learn how to create specification-driven industrial piping designs using Creo Parametric 3.0. This includes learning how to use schematic diagrams created with Creo Schematics to drive 3-D industrial piping designs created within Creo Parametric 3.0. Finally, you learn how to document piping designs by creating drawings that include BOM tables, pipe bend tables, and engineering information, as well as how to export ISOGEN format files for creating pipeline, spool, and systems isometric drawings.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



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

Prerequisites

- Introduction to Creo Parametric 3.0 or equivalent experience

Audience

- This course is intended for engineers who are involved in the 3-D routing of mechanical piping systems and industrial piping systems. People in related roles will also benefit from taking this course.

Agenda

Day 1

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

Day 2

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

Day 3

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

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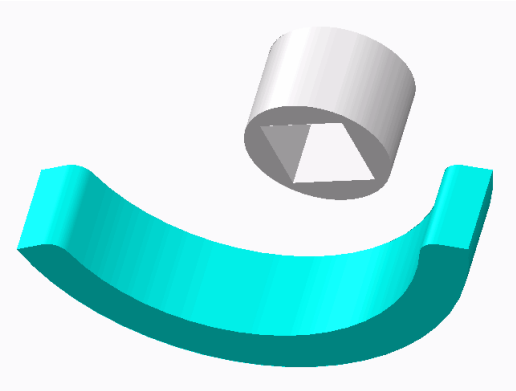
Behavioral Modeling Using Creo Parametric 3.0

Overview

Course Code	TRN-4520-T
Course Length	1 Day



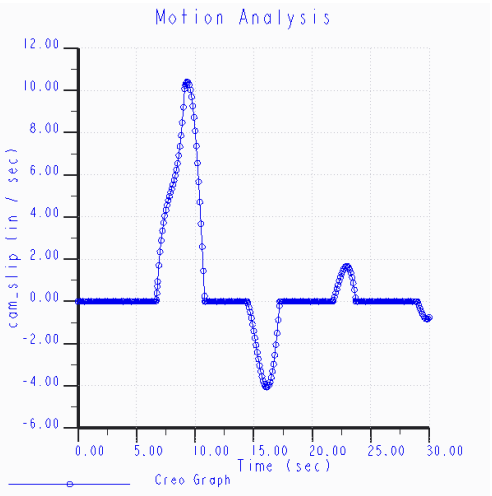
This course is designed for experienced users who want to add additional features to meet or exceed the design specifications of their products.

In this course, you will learn advanced analysis skills unrelated to structural or thermal analysis. You will learn how to analyze your models and create analysis features that can enforce your design intent. You will also learn how to create sensitivity and feasibility studies that help you determine how to reach your design goals. Furthermore, you will learn how to create optimization design studies that enable you to configure the dimensions and parameters that Creo Parametric can change to meet your design specifications. After completing this course, you will be prepared to work on critical component designs using Creo Parametric Behavioral Modeling.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Apply the behavioral modeling process and concepts to your designs
- Create measurement analysis features
- Create relation, motion, Creo Simulate, and MS Excel analysis features
- Create user-defined analysis features
- Conduct sensitivity analyses
- Conduct feasibility and optimization studies



Prerequisites

- Introduction to Creo Parametric or equivalent experience
- Experience with MS Excel, Mechanism Design, Creo Simulate, and Creo Mechanism Dynamics Extension is useful but not required

Audience

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

Agenda

Day 1

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

PTC® University

Web Based Curriculum Guide

- Update to Creo Parametric 3.0 from Creo Parametric 2.0
- Introduction to Creo Parametric 3.0 – Fundamentals
- Introduction to Creo Parametric 3.0 – Productivity Tools
- Advanced Modeling using Creo Parametric 3.0
- Advanced Assembly Design using Creo Parametric 3.0
- Introduction to Creo Simulate 3.0
- Detailing using Creo Parametric 3.0
- Surfacing using Creo Parametric 3.0
- Sheetmetal using Creo Parametric 3.0
- Behavioral Modeling Using Creo Parametric 3.0

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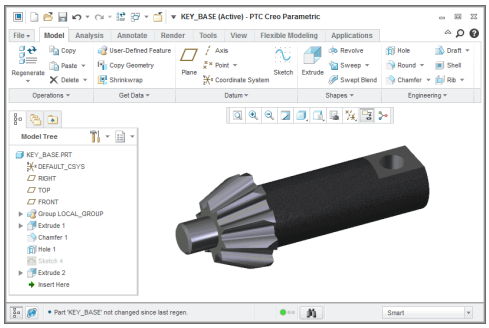
Update to Creo Parametric 3.0 from Creo Parametric 2.0

Overview

Course Code	WBT-4500-0
Course Length	8 Hours

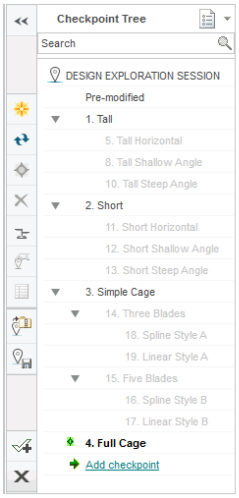


In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric 3.0. You will be introduced to user interface appearance enhancements such as the active window and right-click menu enhancements. You will examine the Part Modeling enhancements to features such as Draft, Pattern, UDF, Rounds, and Spinal Bend. You will experiment with the new Edit References dialog box on part models and study the surfacing enhancements to surface copy, flatten quilt, boundary blends, and the new connection analysis tool. Freestyle surfacing enhancements to the Join tool and aligning to geometry will also be covered. You will also learn about Flexible Modeling enhancements to tangency propagation, flexible patterns, round/chamfer recognition, and the flexible move tool. You will investigate the new Assembly capabilities such as the built-in hardware library and the Intelligent Fastener extension, as well as enhanced Assembly functionality such as heterogeneous (multi-CAD) assembly and the Design Exploration extension. You will examine the new Dimension and Note functionality for 2-D drawings and review various detailing enhancements. Finally, you will learn to use the many updated tools in Sheetmetal mode such as Die Forms, Bend Relief, and Rip, as well as the enhancement for bending in multiple planes.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Use the Interface enhancements in Creo Parametric 3.0
- Use the Part Modeling enhancements in Creo Parametric 3.0
- Use the Surfacing enhancements in Creo Parametric 3.0
- Use the Flexible Modeling enhancements in Creo Parametric 3.0
- Use the Intelligent Fastener extension in Creo Parametric 3.0
- Use the Design Exploration extension in Creo Parametric 3.0
- Use the Drawing enhancements in Creo Parametric 3.0
- Use the Sheetmetal enhancements in Creo Parametric 3.0



Design Exploration session : Full Cage
Session name: FAN_DEX

Prerequisites

- Introduction to Creo Parametric 2.0, or equivalent experience with Creo Parametric 2.0

Audience

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

Table of Contents

Module	1	Interface Enhancements
Module	2	Part Modeling Enhancements
Module	3	Surfacing Enhancements
Module	4	Flexible Modeling Enhancements
Module	5	Utilizing Intelligent Fasteners
Module	6	Utilizing Design Exploration
Module	7	Drawing Enhancements
Module	8	Sheetmetal Enhancements

PTC® University

Introduction to Creo Parametric 3.0 – Fundamentals

Overview

Course Code	WBT-4501-A
Course Length	20 Hours

In this course, you will focus on learning core modeling skills. Topics include understanding the interface and basic Creo Parametric concepts, selecting and editing, sketching and sketcher tools, and basic feature creation. This course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts. After completing this course, you will be well-prepared to work effectively on product design projects using Creo Parametric.

You will complete Pro/FICIENCY skills assessment questions for each topic. These questions are used to help reinforce your understanding of the course topics.

Course Objectives

- Learn the basic Creo Parametric modeling process
- Understand Creo Parametric concepts
- Learn how to use the Creo Parametric interface
- Select and edit geometry, features, and models
- Sketch geometry and use tools
- Create sketches for features
- Create datum planes and datum axes
- Create extrudes, revolves, and profile ribs
- Utilize internal sketches and embedded datums
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Complete the comprehensive design project

Prerequisites

- None

Audience

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

Table of Contents

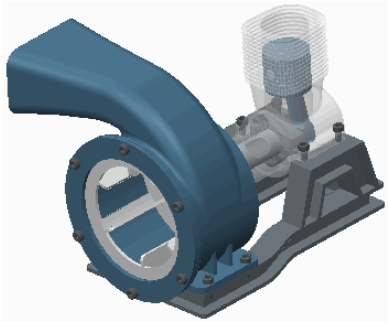
Module	1	Introduction to the Creo Parametric Basic Modeling Process
Module	2	Understanding Creo Parametric Concepts
Module	3	Using the Creo Parametric Interface
Module	4	Selecting Geometry, Features, and Models
Module	5	Editing Geometry, Features, and Models
Module	6	Creating Sketcher Geometry
Module	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	Sketcher Workflow
Module	12	Creating Sweeps and Blends
Module	13	Creating Holes, Shells, and Draft
Module	14	Creating Rounds and Chamfers
Module	15	Project I
Module	16	Appendix

PTC® University

Introduction to Creo Parametric 3.0 – Productivity Tools

Overview

Course Code	WBT-4501-B
Course Length	20 Hours



In this course, you will focus on learning advanced modeling skills. Topics include patterning; measuring and inspecting models; group, copy, and mirror tools; assembly creation; explode states and explode line creation; drawing creation; using layers; investigating parent/child relationships; capturing design intent; and resolving failures and seeking help. This course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts, assemblies, and drawings. After completing this course, you will be well prepared to work effectively on product design projects using Creo Parametric.

You will complete Pro/FICIENCY skills assessment questions for each topic. These questions are used to help reinforce your understanding of the course topics.

Course Objectives

- Group, copy, and mirror items
- Create patterns
- Measure and inspect models
- Assemble with constraints
- Assemble with connections
- Explode assemblies
- Lay out drawings and create views
- Create drawing annotations
- Use layers
- Investigate parent/child relationships
- Capture and manage design intent
- Resolve failures and seek help
- Complete the comprehensive design project

Prerequisites

- Introduction to Creo Parametric – 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.

Table of Contents

Module	1	Group, Copy, and Mirror Tools
Module	2	Creating Patterns
Module	3	Measuring and Inspecting Models
Module	4	Assembling with Constraints
Module	5	Assembling with Connections
Module	6	Exploding Assemblies
Module	7	Drawing Layout and Views
Module	8	Creating Drawing Annotations
Module	9	Using Layers
Module	10	Investigating Parent/Child Relationships
Module	11	Capturing and Managing Design Intent
Module	12	Resolving Failures and Seeking Help
Module	13	Project II
Module	14	Appendix

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Advanced Modeling using Creo Parametric 3.0

Overview

Course Code	WBT-4502-0
Course Length	24 Hours

The Advanced Modeling using Creo Parametric 3.0 training course teaches you how to use advanced part modeling techniques 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 3.0.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- 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 advanced reference management techniques
- Create family tables
- Reuse features
- Learn advanced copy techniques
- Create advanced patterns

Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

Table of Contents

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
Module	8	Relations and Parameters
Module	9	Advanced Blends
Module	10	Sweeps with Variable Sections
Module	11	Helical Sweeps
Module	12	Swept Blends
Module	13	Advanced Layers
Module	14	Advanced Reference Management
Module	15	Family Tables
Module	16	Reusing Features
Module	17	Advanced Copy
Module	18	Advanced Patterns

PTC® University

Advanced Assembly Design using Creo Parametric 3.0

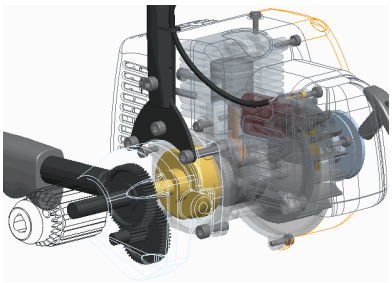
Overview

Course Code	WBT-4503-0
Course Length	24 Hours

In this course, you will learn how to use Creo Parametric 3.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain designs, 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, which are both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Use advanced component selection
- Use advanced assembly constraints
- Create and use component interfaces
- Utilize intelligent fasteners
- Create and use flexible components
- Restructure and mirror assemblies
- Use assembly features and shrinkwrap
- Replace components in an assembly
- Understand the basics of simplified reps
- Create cross-sections, display styles, and combined views
- Substitute components by reps, envelopes, and simplified reps
- Understand advanced simplified rep functionality
- Create and use assembly structure and skeletons
- Utilize design exploration

Prerequisites

- Introduction to Creo Parametric 3.0
- Update to Creo Parametric 3.0 from Creo Parametric 2.0

Audience

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

Table of Contents

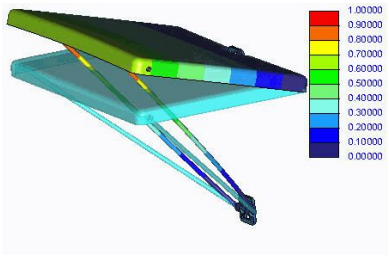
Module	1	Advanced Component Selection
Module	2	Using Advanced Assembly Constraints
Module	3	Creating and Using Component Interfaces
Module	4	Utilizing Intelligent Fasteners
Module	5	Creating and Using Flexible Components
Module	6	Restructuring and Mirroring Assemblies
Module	7	Using Assembly Features and Shrinkwrap
Module	8	Replacing Components in an Assembly
Module	9	Understanding the Basics of Simplified Reps
Module	10	Creating Cross-Sections, Display Styles, Layer States, and Combined Views
Module	11	Substituting Components using User Defined, Envelopes, and Simplified Reps
Module	12	Understanding Advanced Simplified Rep Functionality
Module	13	Creating and Using Assembly Structure and Skeletons
Module	14	Utilizing Design Exploration
Module	15	Project

PTC® University

Introduction to Creo Simulate 3.0

Overview

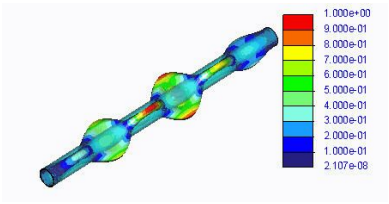
Course Code	WBT-4504-0
Course Length	40 Hours



In this course, you will learn how to test, validate, and optimize product designs with the Creo Simulate module. Creo Simulate enables you to simulate structural and thermal loads on product designs. 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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Understand the basic Simulate analysis process
- Understand theory and simulate model topics
- Explore results
- Explore materials and material properties
- Understand and use Simulate idealizations
- Understand and use structural loads
- Understand and use structural constraints
- Run structural analyses
- Understand convergence
- Analyze assemblies with Simulate
- Complete design and sensitivity studies
- Run optimization studies
- Understand the basics of Thermal analysis

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.

Table of Contents

Module	1	Introduction to Creo Simulate
Module	2	Theoretical Foundations
Module	3	Simulation Models
Module	4	Materials and Material Properties
Module	5	Structural Constraints
Module	6	Structural Loads
Module	7	Meshing
Module	8	Convergence
Module	9	Structural Analysis
Module	10	Introduction to Results Evaluation
Module	11	Refining the Design
Module	12	Basic Model Debugging
Module	13	Singularities
Module	14	Analyzing Assemblies
Module	15	Shells
Module	16	Idealizations
Module	17	Thermal Analysis
Module	18	Advanced Analysis
Module	19	Project

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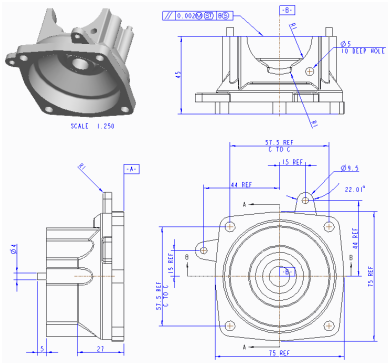
Detailing using Creo Parametric 3.0

Overview

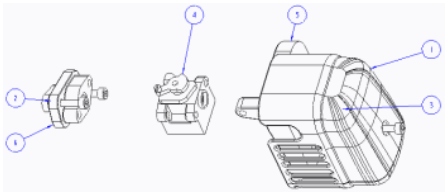
Course Code	WBT-4505-0
Course Length	24 Hours



In this course, you will learn how to quickly create detailed drawings using information captured within 3-D design models. You will also learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and associative nature of Creo Parametric 3.0 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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- 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 3.0

Audience

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

Table of Contents

Module	1	Introduction to Drawings
Module	2	Creating New Drawings
Module	3	Creating Drawing Views
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
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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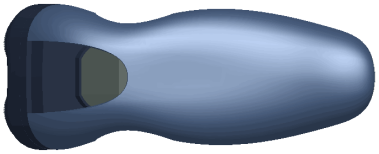
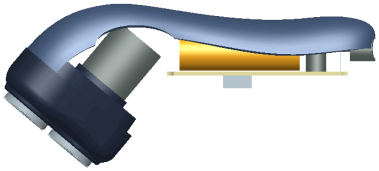
Surfacing using Creo Parametric 3.0

Overview

Course Code	WBT-4506-0
Course Length	24 Hours

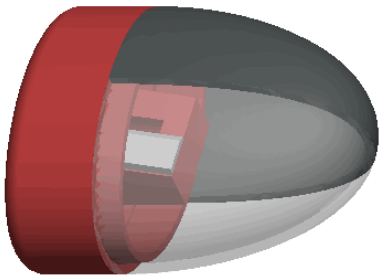




In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how manipulate surfaces using editing tools, and analyze surfaces for quality and desired characteristics. In addition, you will learn how to create solid features using the surfaces as references. 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 set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



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

Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

Table of Contents

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
Module	7	Sweep Surfaces with Variable Sections
Module	8	Helical Sweeps
Module	9	Swept Blends
Module	10	Analyzing Surface Curvature
Module	11	Additional Surface Analysis Tools
Module	12	Extending and Trimming Surfaces
Module	13	Manipulating Surfaces
Module	14	Creating and Editing Solids using Quilts
Module	15	Master Model Technique
Module	16	Project

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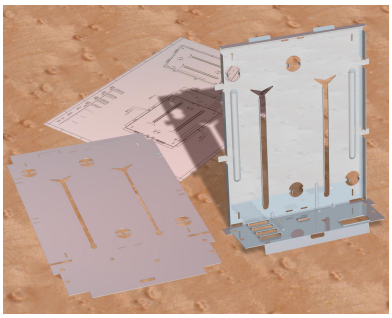
Sheetmetal using Creo Parametric 3.0

Overview

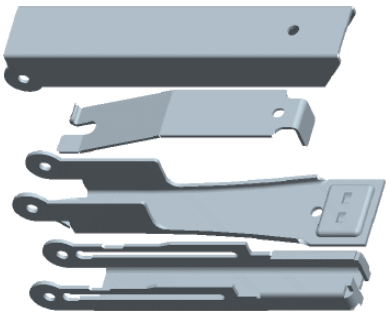
Course Code	WBT-4507-0
Course Length	16 Hours



In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 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 module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives



- Create, convert, and display the sheetmetal model
- Use methods of developed length calculation
- Use primary and secondary wall features, as well as partial walls
- Use bend relief
- Use unbend and bend back features
- Apply sheetmetal bend features
- Use flat patterns
- Create sheetmetal cuts
- Create forms
- Use notch and punch features
- Utilize the sheetmetal environment setup, sheetmetal design information tools, and sheetmetal design rules
- Detail sheetmetal designs

Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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

Table of Contents

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
Module	5	Bending and Unbending Sheetmetal Models
Module	6	Sheetmetal Form Features
Module	7	Modifying Sheetmetal Models
Module	8	Sheetmetal Setup and Tools
Module	9	Detailing Sheetmetal Designs
Module	10	Design Project

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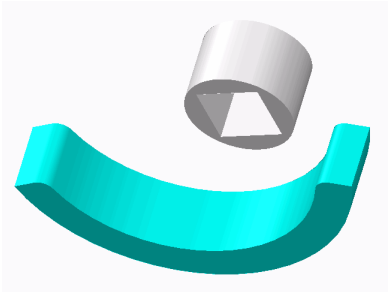
Behavioral Modeling Using Creo Parametric 3.0

Overview

Course Code	WBT-4520-0
Course Length	8 Hours



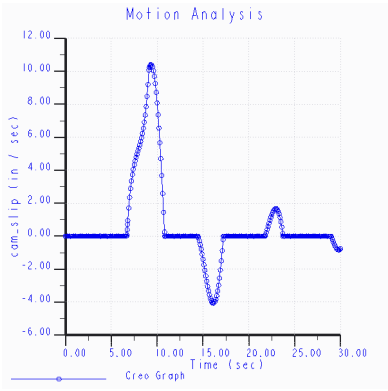
This course is designed for experienced users who want to add additional features to meet or exceed the design specifications of their products.

In this course, you will learn advanced analysis skills unrelated to structural or thermal analysis. You will learn how to analyze your models and create analysis features that can enforce your design intent. You will also learn how to create sensitivity and feasibility studies that help you determine how to reach your design goals. Furthermore, you will learn how to create optimization design studies that enable you to configure the dimensions and parameters that Creo Parametric can change to meet your design specifications. After completing this course, you will be prepared to work on critical component designs using Creo Parametric Behavioral Modeling.

At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

Course Objectives

- Apply the behavioral modeling process and concepts to your designs
- Create measurement analysis features
- Create relation, motion, Creo Simulate, and MS Excel analysis features
- Create user-defined analysis features
- Conduct sensitivity analyses
- Conduct feasibility and optimization studies



Prerequisites

- Introduction to Creo Parametric or equivalent experience
- Experience with MS Excel, Mechanism Design, Creo Simulate, and Creo Mechanism Dynamics Extension is useful but not required

Audience

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

Table of Contents

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