PTC® University

Creo 3.0

Curriculum Guide

PTC® University

Live Classroom Curriculum Guide

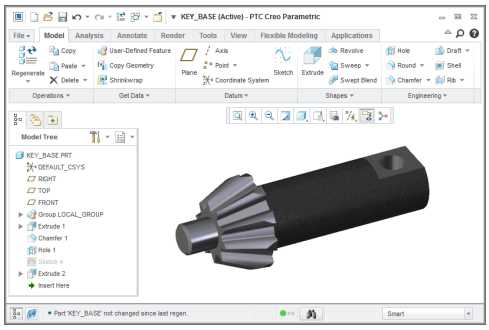
- Update to Creo Parametric 3.0 from Creo Parametric 2.0
- Introduction to Creo Parametric 3.0
- Advanced Modeling using Creo Parametric 3.0
- Advanced Assembly Design using Creo Parametric 3.0
- Introduction to Creo Simulate 3.0
- Detailing using Creo Parametric 3.0
- Surfacing using Creo Parametric 3.0
- Sheetmetal using Creo Parametric 3.0
- Milling using Creo Parametric 3.0
- Introduction to Creo Direct 3.0
- Introduction to Creo Illustrate 3.0
- Behavioral Modeling using Creo Parametric 3.0

PTC® University

Update to Creo Parametric 3.0 from Creo Parametric 2.0

Overview

Course Code	TRN-4500-T
Course Length	1 Day

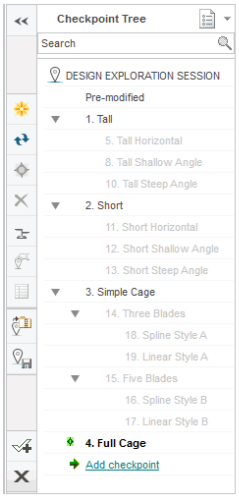


In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric 3.0. You will be introduced to user interface appearance enhancements such as the active window and right-click menu enhancements. You will examine the Part Modeling enhancements to features such 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

Course Content

Module 1. Interface Enhancements

i. Interface Appearance Enhancements
ii. File Save Enhancements
iii. Active Window Enhancements
iv. Graphics Enhancements
v. Applying Realistic Appearances
vi. Model Orientation Enhancement
vii. Right-Click Menu Enhancements
viii. Active Component Enhancements

Knowledge Check Questions

Module 2. Part Modeling Enhancements

i. Draft Enhancements
ii. Pattern Enhancements
iii. UDF and Paste Special Enhancements
iv. Collapse Feature Enhancements
v. Feature Operation Enhancements
vi. Creating Constant Width Rounds
vii. Creating Spinal Bends
viii. Creating Toroidal Bends
ix. Editing Feature References
x. Replacing Feature References

Knowledge Check Questions

Module 3. Surfacing Enhancements

i. Boundary Blend Enhancements
ii. Untrimming Surface Copies
iii. Flattening Quilts
iv. Analyzing Connections
v. Previewing Style Features
vi. Reusing Creo Sketch Curves
vii. Creating Curves from Isolines
viii. Joining Freestyle Geometry
ix. Aligning Freestyle Geometry

Knowledge Check Questions

Module 4. Flexible Modeling Enhancements

i. Managing Tangency
ii. Attaching Moved Geometry
iii. Recognizing Rounds and Chamfers
iv. Editing Chamfers

v. Editing Non-Circular Rounds
vi. Creating Flexible Patterns
vii. Recognizing Patterns and Propagating Changes

Knowledge Check Questions

Module 5. Utilizing Intelligent Fasteners

i. Understanding the Intelligent Fastener Extension
ii. Assembling Intelligent Fasteners
iii. Manipulating Intelligent Fasteners
iv. Assembling Intelligent Fasteners Using Advanced Options
v. Manipulating Intelligent Fasteners Using Advanced Options

Knowledge Check Questions

Module 6. Utilizing Design Exploration

i. Understanding Design Exploration
ii. Exploring Part and Assembly Designs
iii. Creating Design Exploration Branches
iv. Opening and Saving Design Exploration Sessions
v. Using Design Exploration Options
vi. Utilizing Update Control with Copy Geometry Features

Knowledge Check Questions

Module 7. Drawing Enhancements

i. Drawing Standards Enhancements
ii. Printing Enhancements
iii. Dimension Creation Enhancements
iv. Note Creation Enhancements

Knowledge Check Questions

Module 8. Sheetmetal Enhancements

i. Rip Enhancements
ii. Flatten Form Enhancements
iii. Bending in Multiple Planes
iv. Creating Multiple Bend Reliefs
v. Bend Line Relief Placement
vi. Creating Die Forms
vii. Creating Die Forms Using Annotations

Knowledge Check Questions

PTC® University

Introduction to Creo Parametric 3.0

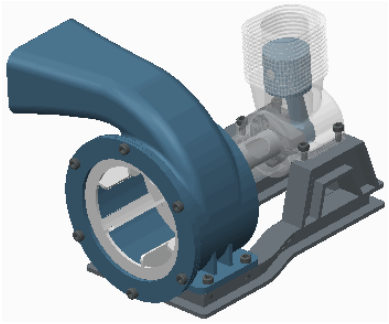
Overview

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

Course Content

Module 1. Introduction to the Creo Parametric Basic Modeling Process

i. Creo Parametric Basic Modeling Process

Module 2. Understanding Creo Parametric Concepts

i. Understanding Solid Modeling Concepts
ii. Understanding Feature-Based Concepts
iii. Understanding Parametric Concepts
iv. Understanding Associative Concepts
v. Understanding Model-Centric Concepts
vi. Recognizing File Extensions

Knowledge Check Questions

Module 3. Using the Creo Parametric Interface

i. Understanding the Main Interface
ii. Understanding the Folder Browser
iii. Understanding the Web Browser
iv. Setting the Working Directory and Opening and Saving Files
v. Understanding the Ribbon Interface
vi. Working with Multiple Windows
vii. Managing Files in Creo Parametric
viii. Understanding Datum Display Options
ix. Understanding Display Style Options
x. Analyzing Basic 3-D Orientation
xi. Understanding the View Manager
xii. Creating and Managing View Orientations
xiii. Managing and Editing Appearances
xiv. Setting Up New Part Models

Knowledge Check Questions

Module 4. Selecting Geometry, Features, and Models

i. Understanding Creo Parametric Basic Controls
ii. Using Drag Handles and Dimension Draggers
iii. Understanding the Model Tree
iv. Understanding Model Tree Filters
v. Selecting Items using Direct Selection
vi. Selecting Items using Query Selection
vii. Using the Search Tool
viii. Using the Smart Selection Filter
ix. Understanding Selection Filters

Knowledge Check Questions

Module 5. Editing Geometry, Features, and Models

i. Renaming Objects
ii. Utilizing Undo and Redo Operations
iii. Understanding Regeneration and Auto Regeneration
iv. Editing Features
v. Editing Features using Edit Definition
vi. Activating and Editing Models
vii. Deleting and Suppressing Items
viii. Editing Feature and Component Visibility

Knowledge Check Questions

Module 6. Creating Sketcher Geometry

i. Reviewing Sketcher Theory
ii. Understanding Design Intent
iii. Modifying the Sketcher Display
iv. Utilizing Constraints
v. Sketching with On-the-Fly Constraints
vi. Sketching Lines
vii. Sketching Centerlines
viii. Sketching Rectangles and Parallelograms
ix. Sketching Circles
x. Sketching Arcs
xi. Sketching Circular Fillets
xii. Sketching Chamfers

Knowledge Check Questions

Module 7. Using Sketcher Tools

i. Understanding Construction Geometry Theory
ii. Sketching Points
iii. Using Geometry Tools within Sketcher
iv. Manipulating Sketches within Sketcher
v. Dimensioning Entities within Sketcher
vi. Modifying Dimensions within Sketcher
vii. Sketcher Conflicts
viii. Creating New Sketch Files
ix. Placing Sections into Sketcher

Knowledge Check Questions

Module 8. Creating Sketches for Features

i. Creating Sketches ('Sketch' Feature)
ii. Specifying and Manipulating the Sketch Setup
iii. Utilizing Sketch References
iv. Using Entity from Edge within Sketcher

Knowledge Check Questions

Module 9. Creating Datum Features: Planes and Axes

i. Creating Datum Features Theory
ii. Creating Datum Axes
iii. Creating Datum Planes

Knowledge Check Questions

Module 10. Creating Extrudes, Revolves, and Ribs

i. Creating Solid Extrude Features
ii. Adding Taper to Extrude Features
iii. Common Dashboard Options: Extrude Depth
iv. Common Dashboard Options: Feature Direction
v. Common Dashboard Options: Thicken Sketch
vi. Creating Solid Revolve Features
vii. Common Dashboard Options: Revolve Angle
viii. Creating Profile Rib Features

Knowledge Check Questions

Module 11. Sketcher Workflow

i. Analyzing Open and Closed Sections
ii. Creating Internal Sketches
iii. Analyzing Sketcher Workflow
iv. Creating Embedded Datum Features

Knowledge Check Questions

Module 12. Creating Sweeps and Blends

i. Creating Sweeps with Open Trajectories
ii. Creating Sweeps with Closed Trajectories
iii. Analyzing Sweep Feature Attributes
iv. Creating Blends by Selecting Parallel Sections
v. Creating Blends by Sketching Sections
vi. Analyzing Blend Options

Knowledge Check Questions

Module 13. Creating Holes, Shells, and Draft

i. Common Dashboard Options: Hole Depth
ii. Creating Coaxial Holes
iii. Creating Linear Holes
iv. Creating Radial and Diameter Holes
v. Exploring Hole Profile Options
vi. Creating Shell Features
vii. Creating Draft Features
viii. Creating Basic Split Drafts

Knowledge Check Questions

Module 14. Creating Rounds and Chamfers

i. Creating Rounds Theory
ii. Creating Rounds by Selecting Edges
iii. Creating Rounds by Selecting a Surface and Edge
iv. Creating Rounds by Selecting Two Surfaces
v. Creating Full Rounds
vi. Creating Round Sets
vii. Creating Chamfers by Selecting Edges
viii. Analyzing Basic Chamfer Dimensioning Schemes
ix. Creating Chamfer Sets

Knowledge Check Questions

Module 15. Project I

i. The Air Circulator
ii. Piston Assembly Components
iii. Crankshaft, Engine Block, Impeller, and Impeller Housing
iv. The Frame and Bolt

Module 16. Group, Copy, and Mirror Tools

i. Creating Local Groups
ii. Copying and Pasting Features
iii. Moving and Rotating Copied Features
iv. Mirroring Selected Features
v. Mirroring All Features
vi. Creating Mirrored Parts

Knowledge Check Questions

Module 17. Creating Patterns

i. Direction Patterning in the First Direction
ii. Direction Patterning in the Second Direction
iii. Axis Patterning in the First Direction
iv. Axis Patterning in the Second Direction
v. Direction Patterning with Multiple Direction Types
vi. Creating Reference Patterns of Features
vii. Creating Reference Patterns of Components
viii. Deleting Patterns or Pattern Members

Knowledge Check Questions

Module 18. Measuring and Inspecting Models

i. Viewing and Editing Model Properties
ii. Investigating Model Units
iii. Analyzing Mass Properties
iv. Using the Measure Tools
v. Using the Measure Summary Tool

vi. Creating Planar Part Cross-Sections
vii. Measuring Global Interference

Knowledge Check Questions

Module 19. Assembling with Constraints

i. Understanding Assembly Theory
ii. Creating New Assembly Models
iii. Understanding Constraint Theory
iv. Understanding Assembly Constraint Status
v. Assembling Components using the Default Constraint
vi. Orienting Components
vii. Creating Coincident Constraints using Geometry
viii. Creating Coincident Constraints using Datum Features
ix. Creating Distance Constraints
x. Creating Parallel, Normal, and Angle Constraints
xi. Assembling using Automatic
xii. Utilizing the Accessory Window

Knowledge Check Questions

Module 20. Assembling with Connections

i. Understanding Connection Theory
ii. Dragging Connected Components
iii. Assembling Components using the Slider Connection
iv. Assembling Components using the Pin Connection
v. Assembling Components using the Cylinder Connection
vi. Analyzing Collision Detection Settings

Knowledge Check Questions

Module 21. Exploding Assemblies

i. Creating and Managing Explode States
ii. Creating Explode Lines
iii. Animating Explode States

Knowledge Check Questions

Module 22. Drawing Layout and Views

i. Analyzing Drawing Concepts and Theory
ii. Analyzing Basic 2-D Orientation
iii. Utilizing the Drawing Tree
iv. Creating New Drawings and Applying Formats
v. Creating and Orienting General Views
vi. Managing Drawing Sheets
vii. Adding Drawing Models
viii. Creating Projection Views
ix. Creating Cross-Section Views

x. Creating Detailed Views
xi. Creating Auxiliary Views
xii. Creating Assembly and Exploded Views
xiii. Modifying Drawing Views
xiv. Creating New Drawings using Drawing Templates

Knowledge Check Questions

Module 23. Creating Drawing Annotations

i. Analyzing Annotation Concepts and Types
ii. Creating Tables from File
iii. Creating BOM Balloons
iv. Showing, Erasing, and Deleting Annotations
v. Cleaning Up Dimensions
vi. Manipulating Dimensions
vii. Creating Driven Dimensions
viii. Inserting Notes
ix. Analyzing Drawing Associativity
x. Publishing Drawings

Knowledge Check Questions

Module 24. Using Layers

i. Understanding Layers
ii. Creating and Managing Layers
iii. Utilizing Layers in Part Models
iv. Utilizing Layers in Assembly Models

Knowledge Check Questions

Module 25. Investigating Parent/Child Relationships

i. Understanding Parent Child Relationships
ii. Viewing Part Parent Child Information
iii. Viewing Assembly Parent Child Information
iv. Viewing Model, Feature, and Component Information

Knowledge Check Questions

Module 26. Capturing and Managing Design Intent

i. Handling Children of Deleted and Suppressed Items
ii. Reordering Features
iii. Inserting Features
iv. Redefining Features and Sketches
v. Capturing Design Intent in Sketches
vi. Capturing Design Intent in Features
vii. Capturing Design Intent in Parts
viii. Capturing Design Intent in Assemblies

Knowledge Check Questions

Module 27. Resolving Failures and Seeking Help

i. Understanding and Identifying Failures
ii. Analyzing Geometry Failures
iii. Analyzing Open Section Failures
iv. Analyzing Missing Part Reference Failures
v. Analyzing Missing Component Failures
vi. Analyzing Missing Component Reference Failures
vii. Analyzing Invalid Assembly Constraint Failures
viii. Recovering Models
ix. Using Creo Parametric Help

Knowledge Check Questions

Module 28. Project II

i. The Air Circulator
ii. Piston Assembly
iii. Engine Block and Drawing
iv. Blower Assembly
v. Engine Blower Assembly
vi. Completing the Design

PTC® University

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

Course Content

Module 1. Advanced Selection

i. Advanced Chain Selection
ii. Advanced Surface Selection
iii. Using the Search Tool

Knowledge Check Questions

Module 2. Advanced Datum Features

i. Creating Datum Graphs
ii. Creating Datum Coordinate Systems
iii. Creating Points On or Offset from Entities
iv. Creating Points at Intersections
v. Creating Points using an Offset Coordinate System
vi. Sketching Geometry Datums
vii. Creating Curves Through a Point or Vertex
viii. Creating a Curve Through a Point Array
ix. Creating a Curve from a Cross-Section
x. Creating a Curve From Equation
xi. Creating Composite Curves
xii. Creating a Curve from Curve Intersections
xiii. Creating a Curve at Surface Intersection
xiv. Projecting and Wrapping Curves
xv. Trimming Curves
xvi. Creating Offset Curves
xvii. Creating Cosmetic Sketches

Knowledge Check Questions

Module 3. Advanced Sketching

i. Using Sketched Curves
ii. Sketching Ellipses
iii. Sketching Elliptical Fillets
iv. Sketching Splines
v. Modifying Splines — Basic Operations
vi. Modifying Splines — Advanced Operations
vii. Importing and Exporting Spline Points
viii. Sketching Conics
ix. Sketching Text
x. Thickening Edges
xi. Analyzing Sketcher Convert Options
xii. Locking Sketcher Entities
xiii. Analyzing Sketcher Dimension Options
xiv. Sketcher Diagnostic Tools

Knowledge Check Questions

Module 4. Advanced Hole Creation

i. Creating Standard Holes
ii. Lightweight Hole Display
iii. Creating Sketched Holes
iv. Creating On Point Holes
v. Creating Cosmetic Threads

Knowledge Check Questions

Module 5. Advanced Drafts and Ribs

i. Drafting Intent Surfaces
ii. Analyzing Draft Hinges and Pull Direction
iii. Creating Drafts with Multiple Angles
iv. Using the Extend Intersect Surfaces Draft Option
v. Creating Drafts Split at Sketch
vi. Creating Drafts Split at Curve
vii. Creating Drafts Split at Surface
viii. Creating Drafts with Variable Pull Direction
ix. Creating Trajectory Ribs

Knowledge Check Questions

Module 6. Advanced Shells

i. Analyzing Shell References and Thickness Options
ii. Excluding Surfaces from Shells
iii. Extending Shell Surfaces
iv. Analyzing Shell Corner Options

Knowledge Check Questions

Module 7. Advanced Rounds and Chamfers

i. Analyzing Round Profile
ii. Analyzing Round Creation Methods
iii. Creating Rounds Through Curve
iv. Creating Variable Radius Rounds
v. Auto Round
vi. Creating Rounds by Reference
vii. Analyzing Round References and Pieces
viii. Using Intent Edges for Rounds
ix. Using Round Transitions
x. Creating Constant Width Rounds
xi. Analyzing Additional Chamfer Types
xii. Analyzing Advanced Chamfer Dimensioning Schemes
xiii. Analyzing Chamfer Creation Methods
xiv. Creating Corner Chamfers

xv. Creating Chamfers by Reference
xvi. Analyzing Chamfer References and Pieces
xvii. Using Intent Edges for Chamfers
xviii. Using Chamfer Transitions

Knowledge Check Questions

Module 8. Relations and Parameters

i. Understanding Relation Theory
ii. Understanding Relation Types
iii. Understanding Basic Relation Operators and Functions
iv. Understanding Advanced Relation Operators and Functions
v. Exact Relation
vi. Creating Parameters
vii. Understanding Advanced Parameter Options
viii. Creating Relations
ix. Creating Relations for Patterns
x. Creating Section Relations
xi. Using the Evalgraph Function
xii. Using Simultaneous Equations

Knowledge Check Questions

Module 9. Advanced Blends

i. Creating Blends by Selecting Non-Parallel Sections
ii. Analyzing Blend Section Tools
iii. Analyzing Blend Tangency
iv. Creating Rotational Blends by Selecting Sections
v. Creating Rotational Blends by Sketching Sections
vi. Analyzing Rotational Blend Options
vii. Analyzing Rotational Blend Tangency

Knowledge Check Questions

Module 10. Sweeps with Variable Sections

i. Understanding Sweeps with Variable Sections Theory
ii. Creating Sweeps using a Constant Section
iii. Creating Sweeps Normal to Trajectory
iv. Creating Sweeps Using Constant Normal Direction
v. Creating Sweeps with Variable Sections Normal to Projection
vi. Analyzing Horizontal and Vertical Control in Sweeps
vii. Creating Sweeps with Variable Sections Utilizing Multiple Trajectories
viii. Creating Sweeps with Variable Sections using Tangent Trajectories
ix. Analyzing Sweeps with Variable Sections Trajectory Options and Rules
x. Using Trajpar with Solid Features
xi. Using Trajpar and Datum Graphs with Solid Features

Knowledge Check Questions

Module 11. Helical Sweeps

i. Understanding Helical Sweeps Theory
ii. Creating Helical Sweeps for Springs
iii. Creating Helical Sweeps for Threads
iv. Analyzing Helical Sweep Profile and Pitch Variations
v. Utilizing Variable Sections in Helical Sweeps

Knowledge Check Questions

Module 12. Swept Blends

i. Understanding Swept Blend Theory
ii. Creating Swept Blends by Selecting Sections
iii. Creating Swept Blends by Sketching Sections
iv. Analyzing Swept Blend Section Options
v. Analyzing Swept Blend Section Plane Control
vi. Analyzing Horizontal and Vertical Control in a Swept Blend
vii. Analyzing Swept Blend Tangency
viii. Analyzing Swept Blend Options
ix. Analyzing Swept Blend Rules
x. Creating Spinal Bends
xi. Creating Toroidal Bends

Knowledge Check Questions

Module 13. Advanced Layers

i. Understanding Layers
ii. Creating and Managing Layers
iii. Creating Layer States
iv. Creating Layer Rules
v. Creating Layers in Assemblies

Knowledge Check Questions

Module 14. Advanced Reference Management

i. Editing Feature References
ii. Replacing Feature References
iii. Replacing Sketcher References
iv. Replacing Sketcher Geometry

Knowledge Check Questions

Module 15. Family Tables

i. Understanding Family Table Theory
ii. Creating a Family Table
iii. Patternizing Family Table Instances
iv. Creating a Multi-Level Family Table

v. Editing Family Table Members

Knowledge Check Questions

Module 16. Reusing Features

i. Creating UDFs
ii. Placing UDFs
iii. Creating UDFs Using On-Surface Coordinate Systems
iv. Creating Inheritance Features
v. Using External Merge to Add Material
vi. Using External Merge to Remove Material

Knowledge Check Questions

Module 17. Advanced Copy

i. Configuring Independency
ii. Analyzing Advanced Reference Configuration
iii. Copying Features Fully Dependent with Options to Vary

Knowledge Check Questions

Module 18. Advanced Patterns

i. Understanding Pattern Regeneration Options
ii. Creating Dimension Patterns in One Direction
iii. Creating Dimension Patterns in Two Directions
iv. Creating Rotational Dimension Patterns
v. Creating Geometry Patterns
vi. Creating Fill Patterns
vii. Specifying Fill Pattern Settings
viii. Creating Pattern Tables
ix. Applying Pattern Tables
x. Creating Curve Patterns
xi. Creating Point Patterns
xii. Unpatterning Group Patterns
xiii. Creating Patterns of Patterns
xiv. Moving/Mirroring Patterns

Knowledge Check Questions

PTC® University

Advanced Assembly Design using Creo Parametric 3.0

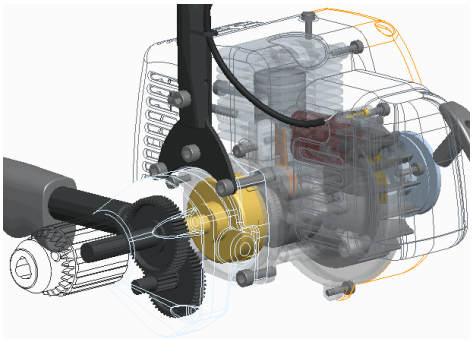
Overview

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

Course Content

Module 1. Advanced Component Selection

i. Locating Components in the Model Tree
ii. Using the Assembly Model Tree Search Field
iii. Selecting Multiple Components

Knowledge Check Questions

Module 2. Using Advanced Assembly Constraints

i. Constraining Components using Fix
ii. Constraining Two Coordinate Systems
iii. Constraining a Point on a Line
iv. Constraining a Point on a Surface
v. Constraining an Edge on a Surface
vi. Constraining a Point on a Point
vii. Creating a Tangent Constraint
viii. Configuring Constraint Sets with Parameters

Knowledge Check Questions

Module 3. Creating and Using Component Interfaces

i. Understanding Component Interfaces
ii. Using a Placing Component Interface
iii. Using a Receiving Component Interface
iv. Creating a Component Interface using the Save as Interface Dialog Box
v. Auto Placing Components
vi. Copying and Pasting Components
vii. Repeating Component Placement

Knowledge Check Questions

Module 4. Utilizing Intelligent Fasteners

i. Understanding the Intelligent Fastener Extension
ii. Assembling Intelligent Fasteners
iii. Manipulating Intelligent Fasteners
iv. Assembling Intelligent Fasteners Using Advanced Options
v. Manipulating Intelligent Fasteners Using Advanced Options

Knowledge Check Questions

Module 5. Creating and Using Flexible Components

i. Adding Flexibility to a Component
ii. Placing Flexible Components in an Assembly
iii. Adding Flexibility to Already Placed Components
iv. Using Flexible Parameters

Knowledge Check Questions

Module 6. Restructuring and Mirroring Assemblies

i. Restructuring and Reordering Assembly Components
ii. Creating Mirrored Assemblies
iii. Creating Mirrored Components
iv. Creating Mirrored Sub-Assemblies

Knowledge Check Questions

Module 7. Using Assembly Features and Shrinkwrap

i. Understanding Assembly Features
ii. Understanding Assembly Feature Intersections
iii. Creating an Assembly Cut
iv. Creating Assembly Holes
v. Creating a Shrinkwrap Feature
vi. Creating a Shrinkwrap Model
vii. Summarizing Shrinkwrap Features and Models

Knowledge Check Questions

Module 8. Replacing Components in an Assembly

i. Understanding Component Replace
ii. Replacing Components using Family Table
iii. Replacing Components using Reference Model
iv. Replacing Components using By Copy
v. Replacing Unrelated Components
vi. Understanding Interchange Assemblies
vii. Replacing using a Functional Interchange Assembly

Knowledge Check Questions

Module 9. Understanding the Basics of Simplified Reps

i. Retrieving Assembly Subsets
ii. Understanding Standard Simplified Reps
iii. Understanding Custom Simplified Reps
iv. Lightweight Graphics Representations
v. Using Graphics Simplified Reps
vi. Using Geometry Simplified Reps
vii. Excluding Components using Simplified Reps
viii. Defining Simplified Reps Using the Component Chooser
ix. Creating a Default Envelope Simplified Rep
x. Creating Part Simplified Reps
xi. Opening Simplified Reps

Knowledge Check Questions

Module 10. Creating Cross-Sections, Display Styles, Layer States, and Combined Views

i. Understanding Assembly Cross-Sections
ii. Creating Assembly Cross-Sections
iii. Creating Offset Assembly Cross-Sections

iv. Creating Zone Assembly Cross-Sections
v. Creating Display Styles
vi. Creating Layer States in an Assembly
vii. Creating Combination Views

Knowledge Check Questions

Module 11. Substituting Components using User Defined, Envelopes, and Simplified Reps

i. Understanding Envelopes
ii. Creating and using a Surface Subset Shrinkwrap Envelope
iii. Creating and Using a Faceted Shrinkwrap Envelope
iv. Creating and Using an All Solid Surfaces Shrinkwrap Envelope
v. Creating and Using a Create Features Envelope
vi. Creating and Using an Envelope Copied from an Existing Part
vii. Substituting Components using User Defined
viii. Substituting by Interchange and Family Table

Knowledge Check Questions

Module 12. Understanding Advanced Simplified Rep Functionality

i. Searching for Components for Simplified Reps
ii. Creating Simplified Reps by Size
iii. Creating Simplified Reps using Zones
iv. Creating Simplified Reps by Distance
v. Creating Simplified Reps using Exterior Components
vi. Defining Simplified Reps using Rules
vii. Using On-Demand Simplified Reps
viii. Creating External Simplified Reps

Knowledge Check Questions

Module 13. Creating and Using Assembly Structure and Skeletons

i. Understanding Skeletons
ii. Creating an Assembly Structure
iii. Creating Skeletons for Space Claims
iv. Creating Skeletons for Placement References
v. Copying a Model to a Skeleton
vi. Creating Multiple Skeletons
vii. Sharing Skeleton Geometry
viii. Creating and Placing Models using Skeleton References
ix. Creating a Motion Skeleton
x. Sketching a Motion Skeleton
xi. Creating Bodies for a Motion Skeleton
xii. Assigning Connections for a Motion Skeleton
xiii. Creating Solid Models from a Motion Skeleton

Knowledge Check Questions

Module 14. Utilizing Design Exploration

i. Understanding Design Exploration
ii. Exploring Part and Assembly Designs
iii. Creating Design Exploration Branches
iv. Opening and Saving Design Exploration Sessions
v. Using Design Exploration Options
vi. Utilizing Update Control with Copy Geometry Features

Knowledge Check Questions

Module 15. Project

i. The Table Fan
ii. Skeleton Models
iii. The Shaft and Arm Parts
iv. Components to Assemblies
v. Editing the Design

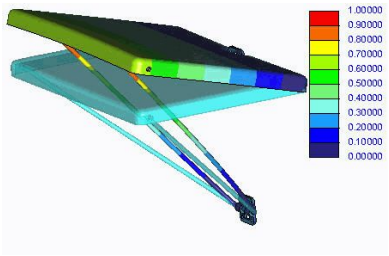
Knowledge Check Questions

PTC® University

Introduction to Creo Simulate 3.0

Overview

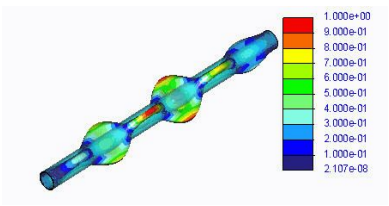
Course Code	TRN-4504-T
Course Length	5 Days



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

Course Content

Module 1. Introduction to Creo Simulate

i. Simulate Analysis Functionality
ii. Simulate Model Functionality
iii. The Simulate Application
iv. The Simulate User Interface Functionality
v. The Typical Simulation Process

Knowledge Check Questions

Module 2. Theoretical Foundations

i. The Finite Element Method
ii. The h- and p-Versions of Finite Elements
iii. The p-Method
iv. Structural Mechanics — Stress Definitions and Hooke's Law
v. Structural Mechanics — Strain Energy and Failure Theories

Knowledge Check Questions

Module 3. Simulation Models

i. Preparing a CAD Model
ii. Using Inheritance and Remove Features
iii. Managing Units
iv. Understanding Model Types
v. Element Types Overview
vi. Defining Simulate Model Geometry
vii. Using Simulate Coordinate Systems
viii. Using Surface Regions
ix. Using Volume Regions
x. Controlling the Display of Simulation Entities
xi. Using Measures

Knowledge Check Questions

Module 4. Materials and Material Properties

i. Understanding Material Properties
ii. Defining Linear Elastic Materials
iii. Understanding Failure Criteria
iv. Creating Materials
v. Using 3-D Material Orientation
vi. Using 2-D Material Orientation
vii. Understanding Material Libraries

Knowledge Check Questions

Module 5. Structural Constraints

i. Defining Constraints

ii. Understanding Displacement Constraints
iii. Understanding Planar, Pin, and Ball Constraints
iv. Understanding Mirror Symmetry Constraints
v. Understanding Cyclic Symmetry Constraints

Knowledge Check Questions

Module 6. Structural Loads

i. Understanding Structural Loads
ii. Defining Global Loads
iii. Defining Forces, Moments, and Pressure
iv. Defining Loads as Functions

Knowledge Check Questions

Module 7. Meshing

i. Understanding Meshes
ii. Understanding Mesh Options
iii. Using AutoGEM Settings

Knowledge Check Questions

Module 8. Convergence

i. Convergence Methods
ii. Error Norms
iii. Comparing Convergence Methods
iv. Selecting a Convergence Method
v. Understanding P-Level Plots
vi. Recommendations for Memory Allocation

Knowledge Check Questions

Module 9. Structural Analysis

i. Fundamentals of a Linear Static Analysis
ii. Defining a Linear Static Analysis
iii. Understanding Modal Analysis
iv. Defining Fatigue Studies and Properties
v. Setting Up the Simulate Solver
vi. Starting, Stopping, and Monitoring the Simulate Solver
vii. Understanding the Batch Process

Knowledge Check Questions

Module 10. Introduction to Results Evaluation

i. Ensuring Result Quality
ii. The Simulate Result Directory Structure
iii. Using the Postprocessor
iv. Reviewing the Results Window
v. Inserting Results

vi. Formatting Results
vii. Performing Basic View Operations
viii. Hiding and Unhiding Results
ix. Editing, Copying, Deleting, Swapping, and Reordering Results Windows
x. Using Results Templates
xi. Using Annotations
xii. Creating Fringe Results
xiii. Creating Vector Results
xiv. Creating Graph Results
xv. Creating a Graph Preference File
xvi. Creating Model Results
xvii. Using Cutting and Capping Surfaces
xviii. Using Results Mode Info and Query
xix. Tying and Untying Results
xx. Controlling Animations
xxi. Exporting Results

Knowledge Check Questions

Module 11. Refining the Design

i. Understanding Design Variables
ii. Defining Design Studies
iii. Understanding Standard Design Studies
iv. Understanding Local Sensitivity Design Studies
v. Understanding Global Sensitivity Design Studies
vi. Understanding Optimization Design Studies
vii. Understanding Design Study Options

Knowledge Check Questions

Module 12. Basic Model Debugging

i. The Diagnostic Tool
ii. Debugging a Model

Knowledge Check Questions

Module 13. Singularities

i. Understanding Singularities
ii. Treating Singularities

Knowledge Check Questions

Module 14. Analyzing Assemblies

i. Using Interfaces
ii. Reviewing Interfaces
iii. Understanding Connections
iv. Using End Welds
v. Using Perimeter Welds

vi. Using Spot Welds
vii. Using Fasteners

Knowledge Check Questions

Module 15. Shells

i. Understanding Shells
ii. Using Shells on Quilts or Volume Surfaces
iii. Using Shell Pairs for Midsurface Models
iv. Using Connection Tools to Join Shell Midsurface Assemblies

Knowledge Check Questions

Module 16. Idealizations

i. Creating Discrete Masses
ii. Creating Rigid Links
iii. Creating Weighted Links
iv. Creating Springs
v. Defining a Beam
vi. Understanding Beam Results

Knowledge Check Questions

Module 17. Thermal Analysis

i. Understanding Thermal Analysis
ii. Creating Heat Loads
iii. Creating Prescribed Temperature Boundary Conditions
iv. Applying Traveling Heat Loads
v. Applying Temperature Loads to a Simulate Structure Model

Knowledge Check Questions

Module 18. Advanced Analysis

i. Understanding Static Analysis with Prestress
ii. Understanding Modal Analysis with Prestress
iii. Understanding Dynamic Analysis
iv. Understanding Linear Buckling Analysis
v. Understanding Nonlinear Stability Analysis: Snap-through
vi. Understanding Contact Analysis
vii. Understanding 2-D Plane Stress and Strain
viii. Understanding Symmetry

Knowledge Check Questions

Module 19. Project

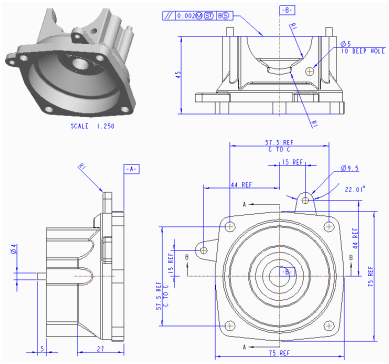
i. The Journeyman's Piece

PTC® University

Detailing using Creo Parametric 3.0

Overview

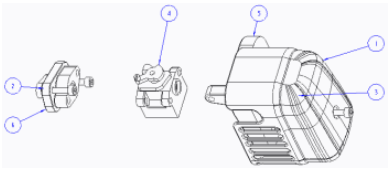
Course Code	TRN-4505-T
Course Length	3 Days



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

Course Content

Module 1. Introduction to Drawings

i. Understanding Drawing Concepts
ii. Drawing Development Process
iii. Understanding the Drawing Ribbon User Interface
iv. Exploring Drawing Ribbon Commands

Knowledge Check Questions

Module 2. Creating New Drawings

i. Creating Drawings Using Formats and Sheets
ii. Creating Drawing Templates
iii. Creating Drawings Using Drawing Templates

Knowledge Check Questions

Module 3. Creating Drawing Views

i. Configuring Drawing Models
ii. Configuring Drawing Sheets
iii. Adding General Views
iv. Adding Projection Views
v. Editing Drawing Views
vi. Editing Visible View Area
vii. Adding Detailed Views
viii. Adding Auxiliary Views
ix. Understanding Cross-Section Concepts and View Types
x. Adding 2-D Cross-Section Views
xi. Modifying Cross Hatching Display
xii. Adding Assembly Exploded Views

Knowledge Check Questions

Module 4. Adding Model Details to Drawings

i. Understanding Annotations in Drawings
ii. Showing, Erasing, and Deleting Annotations
iii. Inserting Driven Dimensions
iv. Inserting Ordinate Dimensions
v. Adjusting Dimensions and Detail Items
vi. Changing Dimension Display

Knowledge Check Questions

Module 5. Adding Notes to Drawings

i. Adding and Editing Notes
ii. Using Parametric Information and Special Characters in Notes

Knowledge Check Questions

Module 6. Adding Tolerance Information

i. Understanding Dimensional Tolerances
ii. Configuring Dimensional Tolerances
iii. Understanding Geometric Tolerances
iv. Setting Up Geometric Tolerance References
v. Applying Geometric Tolerances

Knowledge Check Questions

Module 7. Adding Draft Geometry and Symbols

i. Creating and Editing Draft Geometry
ii. Understanding Drawing Symbols
iii. Using Surface Finish Symbols
iv. Using the Symbol Palette and Custom Symbols
v. Creating Symbols

Knowledge Check Questions

Module 8. Using Layers in Drawings

i. Understanding Layers in Drawings
ii. Using Layers in Drawings

Knowledge Check Questions

Module 9. Creating and Using Tables in Drawings

i. Inserting Tables
ii. Editing Table Properties
iii. Creating Tables from File
iv. Creating Hole Tables

Knowledge Check Questions

Module 10. Using Report Information in Drawings

i. Creating Report Tables
ii. Editing Report Tables
iii. Creating BOM Balloons
iv. Creating Part Catalog Drawings

Knowledge Check Questions

Module 11. Creating Drawing Formats

i. Creating Drawing Formats

Knowledge Check Questions

Module 12. Configuring the Drawing Environment

i. Configuring the Drawing Environment

Knowledge Check Questions

Module 13. Managing Large Drawings

i. Understanding Drawing Regeneration
ii. Managing Large Drawings

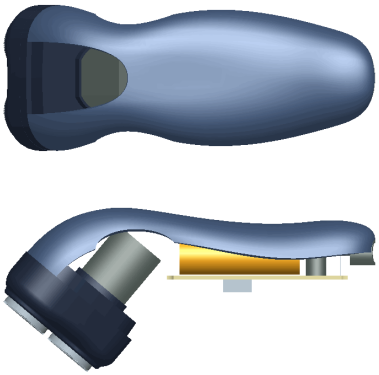
Knowledge Check Questions

PTC® University

Surfacing using Creo Parametric 3.0

Overview

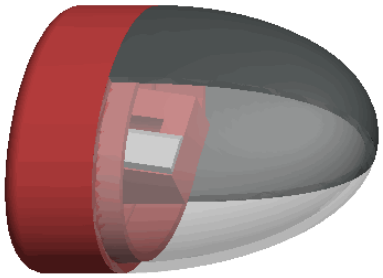
Course Code	TRN-4506-T
Course Length	3 Days



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

Course Content

Module 1. Surface Modeling Overview

i. Introduction to Surfacing
ii. Surface Modeling Uses
iii. Surface Modeling Paradigms
iv. Blending Surface Modeling Paradigms
v. Surfacing Terms

Knowledge Check Questions

Module 2. Advanced Selection

i. Advanced Chain Selection
ii. Advanced Surface Selection
iii. Using the Search Tool

Knowledge Check Questions

Module 3. Advanced Datum Features

i. Creating Datum Graphs
ii. Creating Datum Coordinate Systems
iii. Creating Points On or Offset from Entities
iv. Creating Points at Intersections
v. Creating Points using an Offset Coordinate System
vi. Sketching Geometry Datums
vii. Creating Curves Through a Point or Vertex
viii. Creating a Curve Through a Point Array
ix. Creating a Curve from a Cross-Section
x. Creating a Curve From Equation
xi. Creating Composite Curves
xii. Creating a Curve from Curve Intersections
xiii. Creating a Curve at Surface Intersection
xiv. Projecting and Wrapping Curves
xv. Trimming Curves
xvi. Creating Offset Curves
xvii. Creating Cosmetic Sketches

Knowledge Check Questions

Module 4. Advanced Sketching

i. Using Sketched Curves
ii. Sketching Ellipses
iii. Sketching Elliptical Fillets
iv. Sketching Splines
v. Modifying Splines — Basic Operations
vi. Modifying Splines — Advanced Operations

vii. Importing and Exporting Spline Points
viii. Sketching Conics
ix. Sketching Text
x. Thickening Edges
xi. Analyzing Sketcher Convert Options
xii. Locking Sketcher Entities
xiii. Analyzing Sketcher Dimension Options
xiv. Sketcher Diagnostic Tools

Knowledge Check Questions

Module 5. Basic Surfacing Tools

i. Creating Surface Extrude Features
ii. Creating Surface Revolve Features
iii. Creating Fill Surfaces
iv. Creating Sweep Surfaces with Open Trajectories
v. Creating Sweep Surfaces with Closed Trajectories
vi. Creating Blend Surfaces by Selecting Parallel Sections
vii. Creating Blend Surfaces by Selecting Non-Parallel Sections
viii. Creating Blend Surfaces by Sketching Sections
ix. Analyzing Blend Surface Section Tools
x. Analyzing Blend Surface Options
xi. Analyzing Blend Surface Tangency
xii. Creating Rotational Blend Surfaces by Selecting Sections
xiii. Creating Rotational Blend Surfaces by Sketching Sections
xiv. Analyzing Rotational Blend Surface Options
xv. Analyzing Rotational Blend Surface Tangency

Knowledge Check Questions

Module 6. Boundary Blend Surfaces

i. Understanding Boundary Curve Concepts
ii. Creating Boundary Blends in One Direction
iii. Creating Boundary Blends in Two Directions
iv. Analyzing Blended Surface Boundary Conditions
v. Analyzing Blended Surface Constraint Options
vi. Analyzing Blended Surface Control Points
vii. Creating Boundary Blends with Influencing Curves
viii. Analyzing Approximate Blended Surface Options
ix. Creating a Blend Tangent to Surfaces

Knowledge Check Questions

Module 7. Sweep Surfaces with Variable Sections

i. Understanding Sweeps with Variable Sections
ii. Creating Sweep Surfaces using a Constant Section

iii. Creating Sweep Surfaces using Normal to Trajectory
iv. Creating Sweep Surfaces using Constant Normal Direction
v. Creating Sweep Surfaces using Normal to Projection
vi. Analyzing Horizontal and Vertical Control in a Sweep Surface
vii. Creating Sweep Surfaces Utilizing Multiple Trajectories
viii. Creating Sweep Surfaces using Tangent Trajectories
ix. Analyzing Sweep Surface Trajectory Options and Rules
x. Using Trajpar with Sweep Surface Features
xi. Using Trajpar and Datum Graphs with Sweep Surface Features

Knowledge Check Questions

Module 8. Helical Sweeps

i. Understanding Helical Sweeps Theory
ii. Utilizing Helical Sweeps for Surfaces
iii. Analyzing Helical Sweep Surface Profile and Pitch Variations
iv. Utilizing Variable Sections in Helical Sweep Surfaces

Knowledge Check Questions

Module 9. Swept Blends

i. Understanding Swept Blend Theory
ii. Creating Swept Blend Surfaces by Selecting Sections
iii. Creating Swept Blend Surfaces by Sketching Sections
iv. Analyzing Swept Blend Surface Section Options
v. Analyzing Swept Blend Surface Section Plane Control
vi. Analyzing Horizontal and Vertical Control in a Swept Blend Surface
vii. Analyzing Swept Blend Surface Tangency
viii. Analyzing Swept Blend Surface Options
ix. Analyzing Swept Blend Rules

Knowledge Check Questions

Module 10. Analyzing Surface Curvature

i. Analyzing Surfaces Theory
ii. Defining Curvature
iii. Defining Curvature Continuity
iv. Analyzing Curvature of Curves
v. Analyzing Curvature of Surfaces
vi. Analyzing Curvature using Sections
vii. Analyzing Curvature using Normals
viii. Using Shaded Curvature Analysis for Surfaces
ix. Using Shaded Section Curvature Analysis
x. Creating Curvature Continuous Surfaces
xi. Analyzing Connections

Knowledge Check Questions

Module 11. Additional Surface Analysis Tools

i. Using the Point Analysis Option
ii. Using the Radius Analysis Option
iii. Using the Dihedral Angle Analysis Option
iv. Using the Offset Analysis Option
v. Using the Draft Analysis Option
vi. Using the Slope Analysis Option
vii. Using the Reflection Analysis Option
viii. Using the Shadow Analysis Option

Knowledge Check Questions

Module 12. Extending and Trimming Surfaces

i. Extending Surfaces
ii. Extending Surfaces Using Measurements
iii. Analyzing Extend Surface Options
iv. Creating a Surface Trim
v. Trimming Surfaces with Geometry
vi. Trimming Surfaces with Quilts Options
vii. Trimming Surfaces with the Silhouette Trim Option
viii. Trimming Surfaces with the Vertex Round Option

Knowledge Check Questions

Module 13. Manipulating Surfaces

i. Copying and Pasting Surfaces
ii. Offsetting Surfaces
iii. Offsetting Surfaces with the Expand Option
iv. Offsetting Surfaces with Draft
v. Moving and Rotating Quilts
vi. Mirroring Quilts
vii. Merging Surfaces
viii. Untrimming Surface Copies
ix. Flattening Quilts

Knowledge Check Questions

Module 14. Creating and Editing Solids using Quilts

i. Thickening Surface Quilts
ii. Solidifying Quilts to Add Material
iii. Solidifying Quilts to Remove Material
iv. Solidifying Quilts to Replace Material
v. Offsetting Surfaces using the Replace Option
vi. Creating Rounds on Surfaces
vii. Converting Solid Rounds to Surfaces

Knowledge Check Questions

Module 15. Master Model Technique

i. Master Model Technique Theory
ii. Creating a Master Model
iii. Creating Framework in the Master
iv. Creating Surfaces in the Master
v. Refining and Completing the Master Model
vi. Sharing Geometry from the Master Model
vii. Completing Body Components

Knowledge Check Questions

Module 16. Project

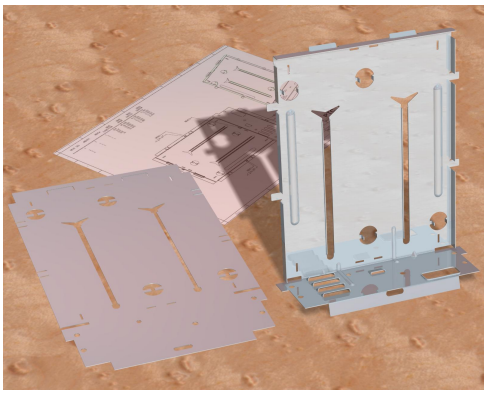
i. The Shaver
ii. Creating the Master Model
iii. Creating Framework in the Master Model
iv. Creating Surfaces in the Master Model
v. Refining and Completing the Master Model
vi. Sharing Geometry from the Master Model
vii. Creating a Body Component

PTC® University

Sheetmetal using Creo Parametric 3.0

Overview

Course Code	TRN-4507-T
Course Length	2 Days

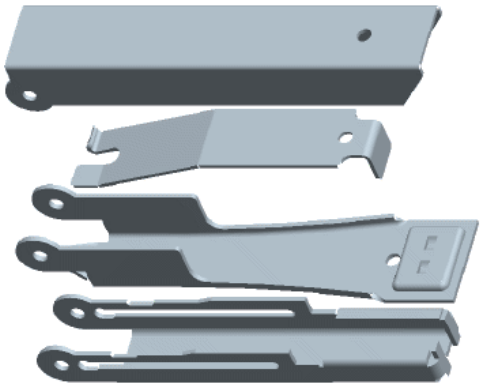


In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 3.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including 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

Course Content

Module 1. Introduction to the Creo Parametric Sheetmetal Design Process

i. Creo Parametric Sheetmetal Design Process

Knowledge Check Questions

Module 2. Sheetmetal Model Fundamentals

i. Sheetmetal Model Fundamentals
ii. Understanding Developed Length
iii. Creating a New Sheetmetal Part in Assembly Mode
iv. Creating a New Sheetmetal Model in Part Mode
v. Converting Solid Models to Sheetmetal

Knowledge Check Questions

Module 3. Creating Primary Sheetmetal Wall Features

i. Understanding Sheetmetal Wall Features
ii. Creating Planar Walls
iii. Extruded Sheetmetal Wall Features
iv. Revolved Sheetmetal Wall Features
v. Blend Sheetmetal Wall Features
vi. Creating Offset Walls
vii. Sheetmetal Wall Sketching Tools
viii. Advanced Primary Walls

Knowledge Check Questions

Module 4. Creating Secondary Sheetmetal Wall Features

i. Understanding Secondary Walls
ii. Creating Secondary Flat Walls
iii. Using Flange Walls
iv. Using Extruded Walls
v. Wall Dashboard Options
vi. Using Partial and Overextended Walls
vii. Understanding Relief
viii. Creating Twist Wall Features
ix. Extending and Trimming Walls
x. Using the Merge Feature

Knowledge Check Questions

Module 5. Bending and Unbending Sheetmetal Models

i. Creating Bend Features
ii. Adding Transition to Bends
iii. Bending in Multiple Planes
iv. Creating Planar Bends
v. Creating Unbend Features

vi. Creating Bend Back Features
vii. Previewing and Creating Flat Patterns
viii. Creating Flat States
ix. Creating Split Area Features

Knowledge Check Questions

Module 6. Sheetmetal Form Features

i. Punch Form Features
ii. Utilizing Punch Model Annotations
iii. Creating Die Forms
iv. Creating Die Forms Using Annotations
v. Creating Sketched Forms
vi. Flattening Forms and Unstamping Edges

Knowledge Check Questions

Module 7. Modifying Sheetmetal Models

i. Sheetmetal Cuts
ii. Notches and Punches
iii. Creating Multiple Bend Reliefs
iv. Bend Line Relief Placement
v. Creating Corner Relief
vi. Creating Rip Features
vii. Creating Edge Bends
viii. Joining Walls
ix. Patterning Walls
x. Mirroring Walls

Knowledge Check Questions

Module 8. Sheetmetal Setup and Tools

i. Bend Line Adjustments
ii. Using Bend Tables for Bend Allowances
iii. Fixed Geometry
iv. Info Tools and Reports
v. Design Rules
vi. Defaults and Parameters
vii. Using Conversion Features

Knowledge Check Questions

Module 9. Detailing Sheetmetal Designs

i. Adding the Flat and Formed States
ii. Auto Ordinate Dimensions
iii. Bend Line Notes
iv. Bend Order Tables

Knowledge Check Questions

Module 10. Design Project

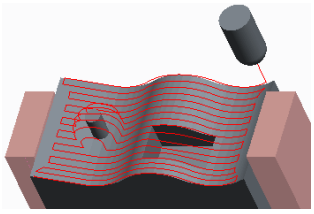
i. Designing a Stapler

PTC® University

Milling using Creo Parametric 3.0

Overview

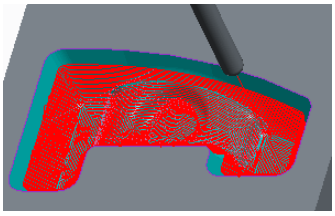
Course Code	TRN-4508-T
Course Length	5 Days



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

Course Content

Module 1. Introduction to Manufacturing

i. Manufacturing Process Overview

Knowledge Check Questions

Module 2. Creating Manufacturing Models

i. Creating Manufacturing Models

Knowledge Check Questions

Module 3. Configuring Operations

i. Configuring Operations

Knowledge Check Questions

Module 4. Using Reference Models

i. Using Reference Models

Knowledge Check Questions

Module 5. Using Workpiece Models

i. Using Workpiece Models

Knowledge Check Questions

Module 6. Creating and Using NC Model Assemblies

i. Creating and Using NC Model Assemblies

Knowledge Check Questions

Module 7. Creating and Configuring a Work Center

i. Creating and Configuring a Work Center

Knowledge Check Questions

Module 8. Creating and Configuring Tools

i. Understanding Milling Tools
ii. Creating Standard Milling Tools
iii. Creating Solid Model Milling Tools
iv. Creating and Using Tool Cutting Data
v. Retrieving Tool Data

Knowledge Check Questions

Module 9. Using Template Manufacturing Models

i. Using Template Manufacturing Models

Knowledge Check Questions

Module 10. Using Manufacturing Parameters

i. Understanding Manufacturing Parameter Concepts
ii. Configuring Parameter Values
iii. Using Site Parameter Files

Knowledge Check Questions

Module 11. Creating Face Milling Sequences

i. Basic Face Milling
ii. Lateral Control Face Milling Parameters
iii. Depth Control Face Milling Parameters
iv. Entry and Exit Face Milling Parameters

Knowledge Check Questions

Module 12. Creating Volume Milling Sequences

i. Basic Volume Milling
ii. Volume Milling with Mill Windows
iii. Scanning Volume Milling Parameters
iv. Depth and Lateral Control Volume Milling Parameters
v. Stock Allowance Volume Milling Parameters
vi. Gathering Mill Volumes
vii. Modifying Volume Milling Toolpaths

Knowledge Check Questions

Module 13. Creating Profile Milling Sequences

i. Basic Profile Milling
ii. Depth and Lateral Control Profile Milling Parameters
iii. Lead In and Lead Out Motions

Knowledge Check Questions

Module 14. Creating Straight Cut Surface Milling Sequences

i. Understanding Surface Milling
ii. Straight Cut Surface Milling
iii. Straight Cut Surface Milling Parameters
iv. Creating Surface Milling Reference Geometry

Knowledge Check Questions

Module 15. Creating From Surface Isolines Surface Milling Sequences

i. From Surface Isolines Surface Milling

Knowledge Check Questions

Module 16. Creating Cut Line Surface Milling Sequences

i. Cut Line Surface Milling

Knowledge Check Questions

Module 17. Advanced Surface Milling Options

i. Advanced Surface Milling Options

Knowledge Check Questions

Module 18. Creating Roughing and Re-roughing Sequences

i. Basic Roughing and Re-roughing
ii. Roughing Scans and Entry and Exit Parameters

iii. Step Depth and Tolerance Control Roughing Parameters
iv. Additional Scallop Height Control Roughing Parameters
v. Roughing Corner Options

Knowledge Check Questions

Module 19. Creating Finishing Sequences

i. Basic Finishing
ii. Editing Finishing Parameters

Knowledge Check Questions

Module 20. Creating Trajectory Milling Sequences

i. Understanding Trajectory Milling
ii. Creating Sketched Milling Tools
iii. Basic 2-Axis Trajectory Milling
iv. 2-Axis Trajectory Milling Depth Control Parameters
v. 2-Axis Trajectory Milling - Cutting Slices Parameters
vi. Basic 3-Axis Trajectory Milling
vii. 3-Axis Trajectory Milling Multi-Step and Multi-Pass Parameters

Knowledge Check Questions

Module 21. Creating Holemaking Sequences

i. Understanding Holemaking
ii. Basic Drilling
iii. Editing Drilling Toolpaths
iv. Creating and Using Drill Groups

Knowledge Check Questions

Module 22. Creating Engraving Sequences

i. Engraving on Flat and Complex Surfaces

Knowledge Check Questions

Module 23. Using the Process Manager

i. Using Process Manager Tools
ii. Editing Process Items
iii. Creating New Items in the Process Manager
iv. Creating and Using Manufacturing Templates

Knowledge Check Questions

Module 24. Creating and Post-Processing CL Data Files

i. Creating and Post-Processing CL Data Files

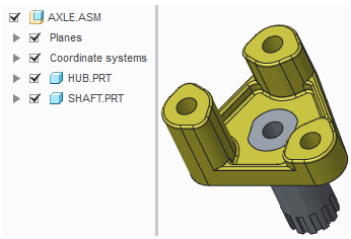
Knowledge Check Questions

PTC® University

Introduction to Creo Direct 3.0

Overview

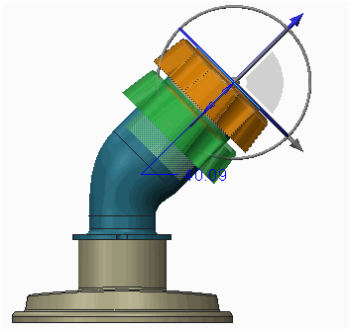
Course Code	TRN-4510-T
Course Length	1 Day



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

Course Content

Module 1. Introduction to Creo Direct

i. Understanding Direct Modeling
ii. Understanding the User Interface
iii. Understanding the Live Toolbar
iv. Opening and Creating a Creo Direct Model
v. Orienting and Positioning the Model
vi. Understanding Selection Behavior
vii. Using Marquee Selection
viii. Understanding Datum and Sketch Display

Knowledge Check Questions

Module 2. Creating Sketches in 2-D Mode

i. Understanding 2-D Mode
ii. Configuring the 2-D Grid
iii. Sketching using Precision Panels
iv. Sketching using Guides
v. Sketching Lines
vi. Sketching Arcs
vii. Sketching Rectangles and Parallelograms
viii. Sketching Circles
ix. Sketching Fillets
x. Sketching Chamfers
xi. Sketching Construction Geometry
xii. Sketching Text
xiii. Sketching Ellipses
xiv. Sketching Splines
xv. Using Sketched Geometry Tools

Knowledge Check Questions

Module 3. Creating Geometry Shapes

i. Understanding Sketches and Sketch Regions
ii. Creating Extrude Geometry
iii. Creating Revolve Geometry
iv. Creating Snapped Geometry Depth
v. Removing Material and Specifying the Side
vi. Projecting 3-D Geometry Into a Sketch
vii. Creating Sweep Geometry

Knowledge Check Questions

Module 4. Creating Datums

i. Creating Datums Theory

ii. Creating Datum Axes
iii. Creating Datum Planes

Knowledge Check Questions

Module 5. Creating Engineering Geometry

i. Creating Holes
ii. Creating and Editing Circular Rounds
iii. Creating and Editing Chamfers
iv. Creating Drafts
v. Shelling Solid Geometry
vi. Patterning Geometry in One Direction
vii. Patterning Geometry in Two Directions

Knowledge Check Questions

Module 6. Measurements and Sections

i. Using the Measure Tools
ii. Creating Part Cross-Sections
iii. Creating Assembly Cross-Sections

Knowledge Check Questions

Module 7. Editing 3-D Geometry

i. Using Shape Selection
ii. Understanding Shape Selection Types
iii. Leveraging Geometry Selection Rules
iv. Understanding the Dragger
v. Moving and Rotating Geometry
vi. Moving Geometry by Dimension
vii. Analyzing Side Surface Options
viii. Offsetting Geometry
ix. Managing Tangency
x. Modifying Analytic Surfaces
xi. Removing Geometry

Knowledge Check Questions

Module 8. Working with Assemblies

i. Analyzing Component Display
ii. Editing Component Position
iii. Creating Assemblies and Inserting Components
iv. Creating Components
v. Designing Components in Assembly

Knowledge Check Questions

Introduction to Creo Illustrate 3.0

Overview

Course Code	TRN-4415-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 dynamically 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 service information content including service procedures, parts identification, training materials, and product assembly and disassembly. Finally, you will learn how to create markup and annotations in figures and animations.

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

- 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 step-wise documentation using the Sequencer module
- Use the Creo Illustrate Animator tools to create animated illustrations
- Create parts list callouts and annotations in illustration figures
- Save, export, and publish illustrations

Prerequisites

- Familiarity with Windows-based file systems and mouse operations
- Familiarity with creating 2-D and 3-D illustrations from CAD data sources

Audience

- This course is intended for technical publications illustrators, technical publications authors, training authors, manufacturing instructors, and users responsible for parts definition, technical marketing, and service planning. People in related roles will also benefit from taking this course.

Agenda

Day 1

Module	1	Introduction to Creo Illustrate
Module	2	Create an Illustration and Work with Figures
Module	3	Exploding Assemblies
Module	4	Authoring Sequenced Process Steps
Module	5	Creo Illustrate Animations
Module	6	Create and Manage Annotations, Sub-Assemblies, and Parts Lists
Module	7	Publishing and Exporting 3-D Illustrations

Course Content

Module 1. Introduction to Creo Illustrate

i. Introducing Creo Illustrate
ii. The Creo Illustrate Process
iii. Understanding the Creo Illustrate User Interface
iv. The Creo Illustrate Figure Viewing Area
v. The Creo Illustrate Ribbon
vi. The Creo Illustrate Ribbon Home Tab
vii. The Creo Illustrate File Menu
viii. The Creo Illustrate Quick Access Toolbar
ix. The Creo Illustrate Primary Panel
x. The Creo Illustrate Upper Data Panel
xi. The Creo Illustrate Lower Data Panel
xii. Creo Illustrate Structure Tree
xiii. The Creo Illustrate Status Bar
xiv. Starting Creo Illustrate
xv. Working with Creo Illustrate

Knowledge Check Questions

Module 2. Create an Illustration and Work with Figures

i. Import 3-D MCAD Data
ii. Working with Figures
iii. Page Setup
iv. sBOM Structure versus Displayed Parts and Sub-Assemblies
v. Figure Orientation
vi. Selecting Parts and Sub-Assemblies
vii. Find Parts and Sub-Assemblies
viii. Changing the Figure Display
ix. Figure Rendering Options
x. Enhanced Lighting
xi. Hide and Unhide Parts and Sub-Assemblies
xii. Inset Views
xiii. Filters

Knowledge Check Questions

Module 3. Exploding Assemblies

i. Explode Figures
ii. Transform Exploded Figures
iii. Free Rotation Exploded Figures
iv. Restore Part and Sub-Assembly Locations
v. Create Explode Lines
vi. Smart Explode

vii. Working with Smart Explode Selection
viii. Setting the Smart Explode Direction
ix. Change Smart Explode Options
x. Save and Exit Smart Explode
xi. Section an Assembly
xii. Orient the Sectioning Plane
xiii. Advanced Sectioning Features
xiv. Create Quarter Cut Sections
xv. Cap the Section
xvi. Set Specific Parts to be Sectioned

Knowledge Check Questions

Module 4. Authoring Sequenced Process Steps

i. The Sequencer Process
ii. Creating Sequence Steps
iii. Editing Sequence Steps
iv. Adding Tagged Symbols and Callouts
v. Reordering Steps within the Sequence

Knowledge Check Questions

Module 5. Creo Illustrate Animations

i. Figure Animations
ii. Animation Control Tools
iii. Playback Controls
iv. Tracks and Keys
v. Timeline Control Tools
vi. Recording Control Tools
vii. Creating an Animation
viii. Using Animation Effects
ix. Editing Tracks
x. Editing Keys
xi. Easing Movement
xii. Previewing the Animation Path
xiii. Exporting a Figure Animation as a Movie

Knowledge Check Questions

Module 6. Create and Manage Annotations, Sub-Assemblies, and Parts Lists

i. Annotating Figures
ii. Customizing Tooltips
iii. Notes and Callout Annotations
iv. Leader Line Annotations
v. Stamp Annotations
vi. The Symbols Library

vii. Measurement Tools
viii. Assemble Parts
ix. Automatically Generate an Item List
x. Structure Edit Mode
xi. Display Parts in the Structure Edit Viewing Area
xii. Advanced sBOM Editing Tools
xiii. Investigating Creo Illustrate Options

Knowledge Check Questions

Module 7. Publishing and Exporting 3-D Illustrations

i. Publishing C3DI Files
ii. Accessing the Save Figure As Menu

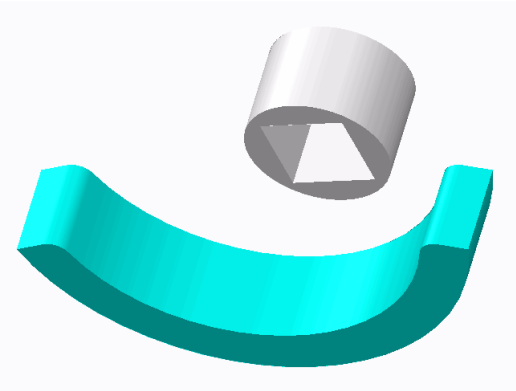
Knowledge Check Questions

PTC® University

Behavioral Modeling Using Creo Parametric 3.0

Overview

Course Code	TRN-4520-T
Course Length	1 Day



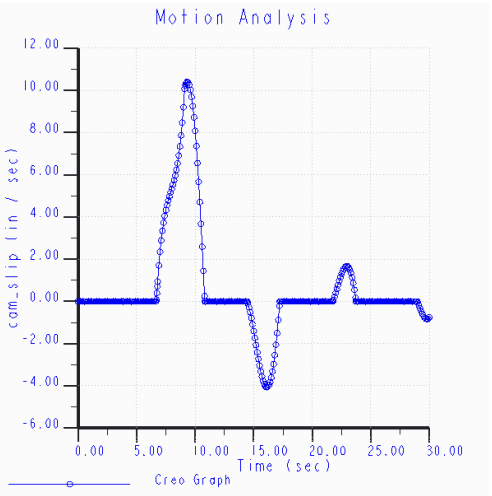
This course is designed for experienced users who want to add additional features to meet or exceed the design specifications of their products.

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

Course Content

Module 1. Introduction to the Behavioral Modeling Process

i. Behavioral Modeling Process
ii. Identifying BMX Analysis Types
iii. Identifying the Differences Between Creo Parametric Analyses

Knowledge Check Questions

Module 2. Creating Measurement Features on Creo Parametric Models

i. Using the Measure Tools
ii. Using the Measure Summary Tool
iii. Creating a Measurement Feature

Knowledge Check Questions

Module 3. Creating Model Property Features on Creo Parametric Models

i. Comparing Model Property Analyses
ii. Measuring Mass Properties
iii. Measuring X-Section Mass Properties
iv. Measuring Pairs Clearance

Knowledge Check Questions

Module 4. Creating Analysis Features on Creo Parametric Models

i. Comparing Analysis Features
ii. Creating a Relation Analysis Feature
iii. Creating a Motion Analysis Feature
iv. Creating a Creo Simulate Analysis Feature
v. Creating an MS Excel Analysis Feature
vi. Creating an External Analysis Feature
vii. Monitoring the Parameters of Analysis Features
viii. Statistical Design Study

Knowledge Check Questions

Module 5. Creating User-Defined Analysis Features on Creo Parametric Models

i. Introduction to User-Defined Analysis Features
ii. Creating Field Points
iii. Creating a Construction Group
iv. Creating User-Defined Analysis Features

Knowledge Check Questions

Module 6. Conducting Design Studies and Optimizing Models

i. Comparing Design Studies
ii. Translating Design Specifications
iii. Performing Sensitivity Analysis
iv. Performing Feasibility Design Studies
v. Performing Optimization Design Studies

Knowledge Check Questions

Module 7. Project

PTC® University

Web Based Curriculum Guide

- Introduction to Creo Illustrate 3.0

PTC® University

Introduction to Creo Illustrate 3.0

Overview

Course Code	WBT-4415-0
Course Length	8 Hours

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 dynamically 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 service information content including service procedures, parts identification, training materials, and product assembly and disassembly. Finally, you will learn how to create markup and annotations in figures and animations.

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

- 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 step-wise documentation using the Sequencer module
- Use the Creo Illustrate Animator tools to create animated illustrations
- Create parts list callouts and annotations in illustration figures
- Save, export, and publish illustrations

Prerequisites

- Familiarity with Windows-based file systems and mouse operations
- Familiarity with creating 2-D and 3-D illustrations from CAD data sources

Audience

- This course is intended for technical publications illustrators, technical publications authors, training authors, manufacturing instructors, and users responsible for parts definition, technical marketing, and service planning. People in related roles will also benefit from taking this course.

Table of Contents