

Introduction to Creo Parametric 4.0

Overview

Course Code TRN-5101-T

Course Length 40 Hours

In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 4.0. 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 4.0.

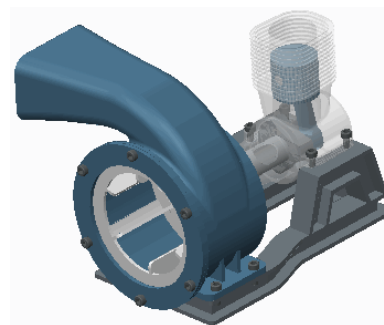
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0.



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.
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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
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Module	26	Capturing and Managing Design Intent
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Module	27	Resolving Failures and Seeking Help
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Module	28	Project II
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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. Using the Geometry Selection Filter
- vi. Understanding Selection Filters
- vii. Selecting Items Using Direct Selection
- viii. Selecting Items Using Query Selection
- ix. Using the Search Tool

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. Assigning Materials
 - iv. Analyzing Mass Properties
 - v. Using the Measure Tools
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- vi. Using the Measure Summary Tool
- vii. Creating Planar Part Cross-Sections
- viii. 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
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- 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
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*Knowledge Check Questions***Module 27. Resolving Failures and Seeking Help**

- i. Understanding and Identifying Failures
- ii. Understanding the Notification Center
- iii. Analyzing Geometry Failures
- iv. Analyzing Open Section Failures
- v. Analyzing Missing Part Reference Failures
- vi. Analyzing Missing Component Failures
- vii. Analyzing Missing Component Reference Failures
- viii. Analyzing Invalid Assembly Constraint Failures
- ix. Recovering Models
- x. 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
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Detailing Using Creo Parametric 4.0

Overview

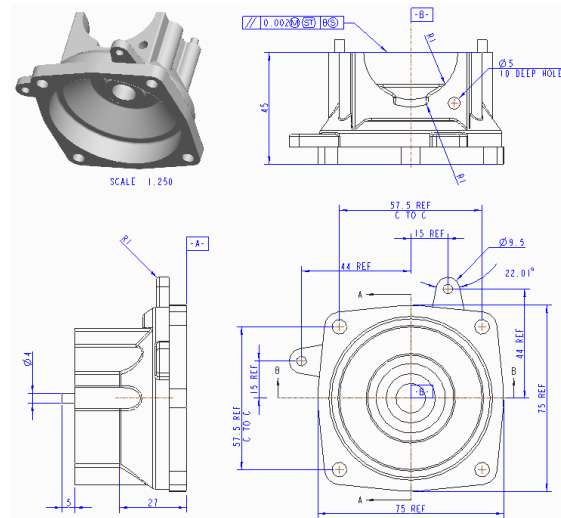
Course Code TRN-5105-T

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

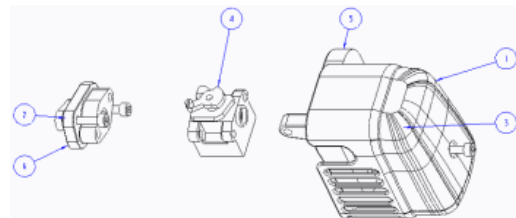
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0.



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 4.0

Audience

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

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. Creating Datum Feature Symbols
- vi. Creating Datum Targets
- vii. 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
- vi. Embedding Images in Drawings

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

Surfacing Using Creo Parametric 4.0

Overview

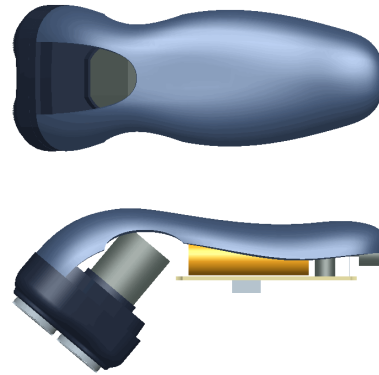
Course Code TRN-5106-T

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 to 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.

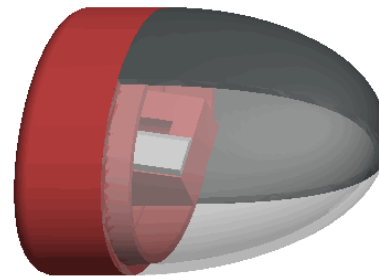
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo 4.0 U360.



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 4.0

Audience

- This course is intended for mechanical designers, design engineers, industrial designers, and related roles
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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
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- 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
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- 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 Model
- iv. Creating Surfaces in the Master Model
- 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
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Advanced Modeling using Creo Parametric 4.0

Overview

Course Code TRN-5102-T

Course Length 24 Hours

The Advanced Modeling using Creo Parametric 4.0 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 4.0.

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 PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0.



Course Objectives

- Learn advanced selection techniques
 - Create advanced datum features
 - Use advanced sketching techniques
 - Create advanced holes
 - Create advanced drafts and ribs
 - Create advanced shells
 - Create advanced rounds and chamfers
 - Use relations and parameters
 - Create advanced blends
 - Create sweeps with variable sections
 - Create helical sweeps and 3D 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 4.0

Audience

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

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

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

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
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Module 4. Advanced Hole Creation

- i. Creating Standard Holes
- ii. Lightweight Hole Display
- iii. Creating Sketched Holes
- iv. Creating On Point Holes
- v. Using the Top Clearance Option
- vi. Creating Cosmetic Threads

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. Using the Exclude Areas with Draft Option
- x. Creating Trajectory Ribs

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

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
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- 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

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

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

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
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- x. Using Trajpar with Solid Features
- xi. Using Trajpar and Datum Graphs with Solid Features

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

Module 12. Swept Blends and Advanced Bends

- 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

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

Module 14. Advanced Reference Management

- i. Editing Feature References
- ii. Replacing Feature References
- iii. Displaying Missing References
- iv. Replacing Sketcher References
- v. Replacing Sketcher Geometry

Module 15. Family Tables

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

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

Module 17. Advanced Copy

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

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
-

Advanced Assembly Design using Creo Parametric 4.0

Overview

Course Code TRN-5103-T

Course Length 24 Hours

In this course, you will learn how to use Creo Parametric 4.0 to create and manage complex assemblies. You will discover 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.

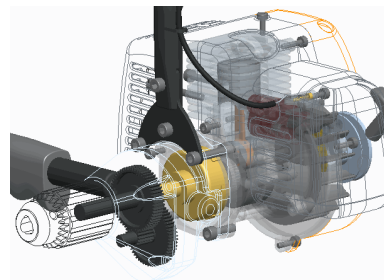
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0



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 4.0
- Update to Creo Parametric 4.0 from Creo Parametric 3.0

Audience

- This course is intended for design engineers and mechanical designers. People in related roles will also benefit from taking this course.
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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. Creating Flexible Components with Varied Material
- v. 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. Using Automatic Representations
- v. Excluding Components Using Simplified Reps
- vi. Defining Simplified Reps Using the Component Chooser
- vii. Creating a Default Envelope Simplified Rep
- viii. Creating Part Simplified Reps
- ix. 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 Appearance States
- vii. Creating Layer States in an Assembly
- viii. 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
-

Sheetmetal using Creo Parametric 4.0

Overview

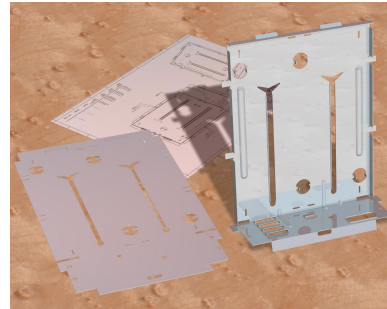
Course Code TRN-5107-T

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 4.0 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.

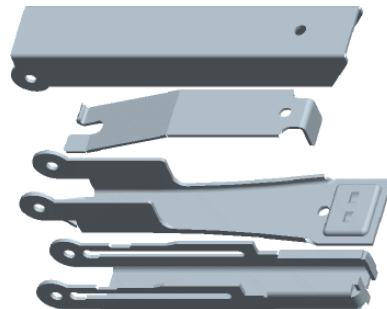
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0.



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 4.0

Audience

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

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

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

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

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

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
- vii. Utilizing Dependency Control with Punch and Die Forms

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

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

Module 9. Detailing Sheetmetal Designs

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

Module 10. Design Project

i. Designing a Stapler

Mechanism Design using Creo Parametric 4.0

Overview

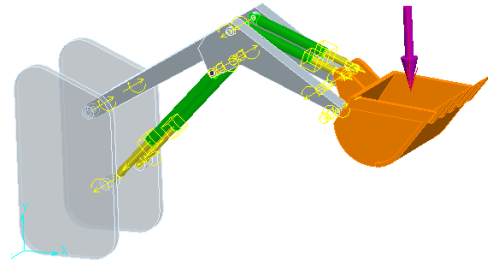
Course Code TRN-5121-T

Course Length 8 Hours

In this course, you will learn about creating mechanism connections, configuring the mechanism model, creating a kinematic analysis, and evaluating results. Mechanism Design using Creo Parametric is designed for experienced users who want to add motion to their models by creating mechanism connections and servo motors. In Creo Parametric you can add motion to your models using the standard mechanism functionality, often referred to as the Mechanism Design Extension (MDX). These topics will enable you to simulate the range of motion between components in your moving assemblies, create gear connections that simulate the gear ratios, create Cam connections that enable Creo Parametric parts to “push” other parts they come into contact with, and check for collisions between moving components. After completing this course, you will be prepared to work on mechanism designs using Creo Parametric Mechanism Design.

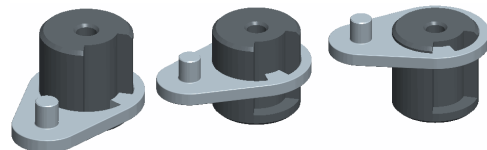
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0.



Course Objectives

- Introduce the mechanism design process
- Create mechanism connections
- Configure motion and analysis
- Evaluate analysis results



Prerequisites

- Introduction to Creo Parametric

Audience

- This course is intended for design engineers and mechanical designers who need to add and evaluate the motion of their assemblies. People in related roles will also benefit from taking this course.
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Agenda

Day 1

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

Course Content

Module 1. Introduction to the Mechanism Design Process

- i. Introduction to Mechanism Design
- ii. Understanding the Mechanism Design Process
- iii. Creating the Model
- iv. Verifying the Mechanism
- v. Adding Servo Motors
- vi. Preparing for Analysis of a Mechanism
- vii. Analyzing the Mechanism
- viii. Evaluating Analysis Results

Module 2. Creating Mechanism Connections

- i. Creating Mechanism Bodies
- ii. Understanding Constraints and Connection Sets
- iii. Understanding Predefined Connection Sets
- iv. Configuring Motion Axis Settings
- v. Using Rigid Connection Sets
- vi. Using Pin Connection Sets
- vii. Using Slider Connection Sets
- viii. Using Cylinder Connection Sets
- ix. Using Planar Connection Sets
- x. Using Ball Connection Sets
- xi. Using Weld Connection Sets
- xii. Using Bearing Connection Sets
- xiii. Using General Connection Sets
- xiv. Using Slot Connection Sets
- xv. Creating Cam-Follower Connections
- xvi. 3D Contact
- xvii. Creating Generic Gear Connections
- xviii. Creating Dynamic Gear Connections
- xix. Creating Belt Connections
- xx. Using the Drag and Snapshot Tools

Module 3. Configuring Motion and Analysis

- i. Understanding Servo Motors
 - ii. Understanding Analysis Definitions
 - iii. Creating Geometry Servo Motors
 - iv. Creating Motion Axis Servo Motors
 - v. Creating Slot Motors
 - vi. Graphing the Magnitude of Servo Motors
-

- vii. Assigning Constant Motion
- viii. Assigning Ramp Motion
- ix. Assigning Cosine Motion
- x. Assigning SCCA Motion
- xi. Assigning Cycloidal Motion
- xii. Assigning Parabolic Motion
- xiii. Assigning Polynomial Motion
- xiv. Assigning Table Motion

Module 4. Evaluating Analysis Results

- i. Generating Measure Results for Analysis
 - ii. Creating Analysis Measure Definitions
 - iii. Evaluating Playback Results
 - iv. Understanding the Animate Dialog Box
 - v. Checking for Collisions
 - vi. Creating Motion Envelopes
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Mechanism Simulation Using Creo Parametric 4.0

Overview

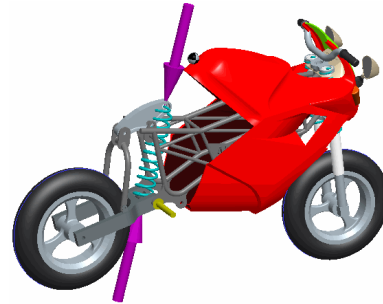
Course Code TRN-5122-T

Course Length 8 Hours

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

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 PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0



Course Objectives

- Understand the mechanism dynamics option
- Apply force motors, springs, and dampers to assemblies
- Apply forces, torques, and gravity to assemblies
- Create dynamic analyses
- Create force balance analyses
- Create static analyses
- Measure forces, velocities, accelerations, and other reactions
- Evaluate results



Prerequisites

- Introduction to Creo Parametric
- Mechanism Design using Creo Parametric

Audience

- This course is intended for design engineers and mechanical designers who need to add and evaluate the motion of their assemblies. People in related roles will also benefit from taking this course.
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Agenda

Day 1

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

Course Content

Module 1. Introduction to the Mechanism Simulation Process

- i. Introduction to Mechanism Simulation
- ii. Understanding the Mechanism Simulation Process
- iii. Creating the Model
- iv. Verifying the Mechanism
- v. Adding Dynamic Entities
- vi. Preparing for Analysis of a Mechanism
- vii. Analyzing the Mechanism
- viii. Evaluating Analysis Results

Knowledge Check Questions

Module 2. Adding Dynamic Entities to a Mechanism

- i. Defining Mass Properties for a Dynamic Analysis
- ii. Creating Force Motors
- iii. Creating Springs
- iv. Creating Dampers
- v. Creating Dynamic Gear Connections
- vi. Creating Belt Connections
- vii. Using Dynamic Properties and Set Zero Position
- viii. Applying Friction and Restitution
- ix. Applying Force and Torque Loads
- x. Applying Gravity

Knowledge Check Questions

Module 3. Analyzing the Mechanism Model

- i. Understanding Mechanism Dynamics Option Analysis Definitions
- ii. Configuring a Dynamic Analysis
- iii. Configuring a Static Analysis
- iv. Configuring a Force Balance Analysis
- v. Defining Initial Configurations
- vi. Creating Measures
- vii. Understanding Redundancies and Degrees of Freedom

Knowledge Check Questions

Module 4. Evaluating Analysis Results

- i. Running Mechanism Analyses
- ii. Evaluating Playback Results for Collisions
- iii. Configuring Playback Results
- iv. Evaluating Results Using Display Arrows
- v. Graphing Measure Results

Knowledge Check Questions

Module 5. Project

i. The Stunt Bike

Introduction to Creo Simulate 4.0

Overview

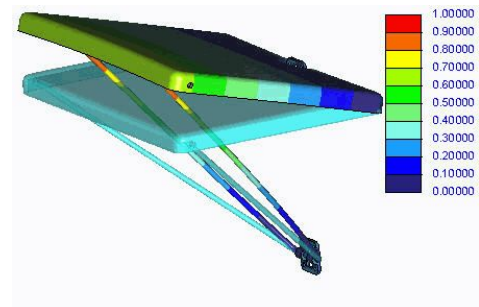
Course Code TRN-5104-T

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.

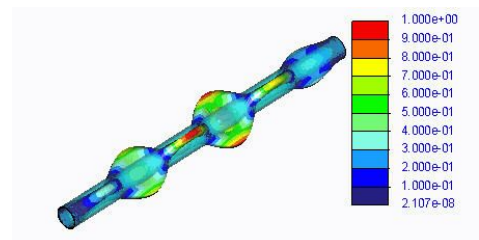
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Simulate 4.0



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
-

Cabling using Creo Parametric 4.0

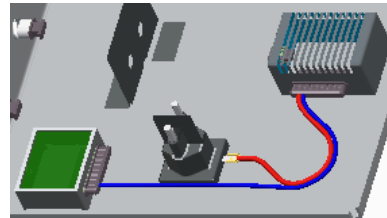
Overview

Course Code	TRN-5115-T
Course Length	24 Hours

In this course, you will learn how to create 3-D electrical harnesses using Creo Parametric 4.0. This includes using Creo Schematics to pass schematic diagram information into the 3-D harness designs created within Creo Parametric 4.0. You will learn how to route electrical harnesses both with and without schematic diagram information, create flattened harnesses for manufacturing, and document harness designs by creating flattened harness drawings that include customized BOM tables and wire list information.

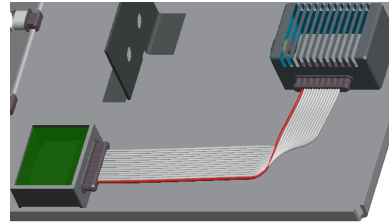
A significant portion of the course is devoted to a cabling design project, during which you will create a full wiring harness with minimal *picks and clicks* to solidify techniques learned previously in the course.

After successfully completing the course, you will be able to create 3-D electrical harnesses and associated manufacturing deliverables using Creo Parametric 4.0. Optionally, you may wish to attend the Introduction to Creo Schematics course. This will enable a full understanding of the schematic design process used to provide schematic data for the creation of electrical harness assemblies in Creo Parametric 4.0. 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

- Understand the basic Creo Parametric cabling process
- Create harness assembly structures
- Set up for cabling
- Route wires and cables
- Modify wire routings
- Route and utilize networks
- Establish logical references
- Create harness components and cosmetics
- Create a flat harness
- Document harness designs
- Navigate through a comprehensive design project



Prerequisites

- Introduction to Creo Parametric 4.0 or equivalent experience
- Introduction to Creo Schematics 4.0 (optional)

Audience

- This course is intended for engineers involved in the 3-D routing and documenting of electrical wiring and cabling harnesses. People in related roles will also benefit from taking this course.

Agenda

Day 1

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

Day 2

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

Day 3

Module	11	Documenting Harness Designs
Module	12	Project (Creo Schematics-Based)
Module	13	Project (Manual Routing)

Course Content

Module 1. Introduction to the Creo Parametric Basic Cabling Process

- i. Step 1: Assembly and Cabling Setup
- ii. Step 2: Routing Wires and Cables
- iii. Step 3: Flattening the Harness
- iv. Step 4: Creating the Harness Drawing

Module 2. Creating Harness Assembly Structures

- i. Understanding Cabling Assembly Structure
- ii. Understanding Electrical Assembly Structure: Sub-Assembly
- iii. Understanding Electrical Assembly Structure: No Sub-Assembly
- iv. Understanding Electrical Assembly Structure: Sub-Assemblies at Top Level
- v. Utilizing Simplified Representations for Cabling
- vi. Creating Cabling Assembly Structures
- vii. Sharing Routing Geometry using Copy Geometry
- viii. Sharing Routing Geometry using Shrinkwrap
- ix. Creating and Configuring Connectors
- x. Assembling Connectors

Module 3. Setting Up for Cabling

- i. Understanding the Cabling Interface
- ii. Creating a Harness Part
- iii. Creating a Wire Color Appearance File
- iv. Manually Designating Connector and Entry Ports
- v. Designating Connectors using Entry Port Parameters
- vi. Designate Components On-the-Fly
- vii. Creating Wire Spools
- viii. Creating Cable Spools
- ix. Creating Ribbon Cable Spools

Module 4. Routing Wires and Cables

- i. Routing Wires using Simple Route
 - ii. Inserting and Editing Wire Locations
 - iii. Creating Wire Locations from References
 - iv. Routing Wires using Follow Cable
 - v. Rerouting Wires
 - vi. Routing Cables
 - vii. Routing Ribbon Cables
-

Module 5. Modifying Wire Routings

- i. Deleting Wires and Segments
- ii. Editing Location Properties
- iii. Modifying Wire Packing
- iv. Modifying Routing Dimensions
- v. Modifying Wire Lengths
- vi. Utilizing Information Tools

Module 6. Routing and Utilizing Networks

- i. Creating Networks
- ii. Checking Networks
- iii. Routing using Networks
- iv. Modifying Networks While Routing
- v. Sharing and Connecting Networks

Module 7. Establishing Logical References

- i. Logical References Overview
- ii. Investigating Creo Schematics Diagrams
- iii. Exporting Data from Creo Schematics
- iv. Importing Creo Schematics Data Into Creo Parametric
- v. Autodesignating Components
- vi. Viewing Designation Information

Module 8. Routing Wires and Cables Using Logical Data

- i. Routing with Logical Data
- ii. Routing with Logical Data and Networks
- iii. Viewing Routing Information
- iv. Updating Creo Schematics Designs
- v. Updating Creo Parametric with New Creo Schematics Data

Module 9. Creating Harness Components and Cosmetics

- i. Creating Splices with Logical Data
 - ii. Creating Splices Manually
 - iii. Creating Sheath Spools
 - iv. Creating Bundles
 - v. Creating Cabling Cosmetics
 - vi. Creating Custom Components
-

Module 10. Creating Flat Harnesses

- i. Creating a Flat Harness Model
- ii. Using Manual Fan
- iii. Using Auto Fan
- iv. Modifying Flattened Segments
- v. Assembling Harness Components
- vi. Analyzing Harness Component Operations
- vii. Viewing Harness Information
- viii. Closing Loops
- ix. Investigating Additional Flatten Features

Module 11. Documenting Harness Designs

- i. Creating Cabling Assembly Views
- ii. Creating Harness Views
- iii. Creating Harness Report Tables
- iv. Placing Harness BOM Tables
- v. Placing Connector Pinout Tables
- vi. Placing Spool BOM Tables
- vii. Placing Harness From and To Tables
- viii. Showing Cabling Detail Items

Module 12. Project (Creo Schematics-Based)

- i. The Electrical Cabinet
- ii. Set Up for Cabling
- iii. Routing Wires and Cables
- iv. Modifying Wires and Cable Routing
- v. Flattening the Harness
- vi. Creating a Harness Drawing

Module 13. Project (Manual Routing)

- i. The Electrical Cabinet
 - ii. Set Up for Cabling
 - iii. Routing Wires and Cables
 - iv. Modifying Wires and Cable Routing
 - v. Flattening the Harness
 - vi. Creating a Harness Drawing
-

Introduction to Model Based Definition with Creo Parametric 4.0

Overview

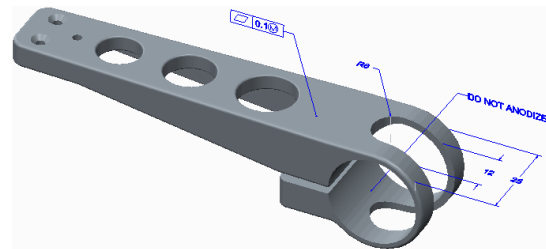
Course Code TRN-5109-T

Course Length 8 Hours

In this course, you will learn how to create an MBD model using a start part. You will discover how annotate mode provides the tools to create an MBD model using the MBD process within Creo Parametric 4.0. You will also learn about creating 3-D annotations and how to utilize them in combined states. You will manage the visibility of annotations within combination states. You will also create and modify various annotation elements, including dimensions, ordinate dimensions, datum reference features, geometric tolerances, surface finishes, symbols, notes, and datum targets. After completing this course, you will have a better understanding of the MBD process and how to create MBD models using the MBD process.

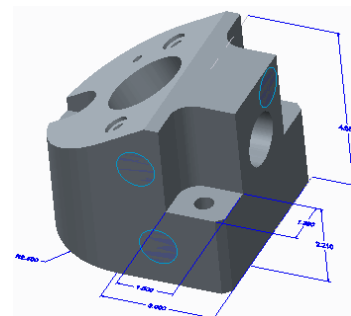
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0 F000.



Course Objectives

- Understand the introduction to Model Based Definition
- Prepare models for annotation
- Create annotation elements
- Modify annotation elements
- Complete combination states
- Publish for technical data packages



Prerequisites

- Core Creo knowledge in parts and drawings
- Experience in Creo modeling
- Experience in Creo drawing creation

Audience

- This course is intended for any CAD designer involved in creating Model Based Definition models at his or her company. People in related roles will also benefit from taking this course.
-

Agenda

Day 1

Module	1	Introduction to Model Based Definition
Module	2	Preparing Models for Annotation
Module	3	Creating Annotation Elements
Module	4	Modifying Annotation Elements
Module	5	Completing Combination States
Module	6	Publishing for Technical Data Packages

Course Content

Module 1. Introduction to Model Based Definition

- i. Understanding Model Based Definition
- ii. Understanding MBD Model Types
- iii. Understanding Types of Drawing Information
- iv. Organizing Drawing Information
- v. Using Schema
- vi. Understanding 3-D Annotations
- vii. Reviewing Annotation Types
- viii. Avoiding a Mass of Unreadable Data
- ix. Understanding the MBD Process
- x. Creating a Technical Data Package

Module 2. Preparing Models for Annotation

- i. Understanding the Annotation Mode Interface
- ii. Understanding the Annotation Mode Tab
- iii. Accessing Combined States
- iv. Understanding Organizational Schema
- v. Reviewing Annotations and Combined States
- vi. Reviewing Minimally Dimensioned Parts
- vii. Utilizing Start Parts for MBD
- viii. Creating New Combined States for MBD
- ix. Managing Visibility of Annotations and Supplemental Geometry
- x. Defining View Orientations
- xi. Determining Features to Annotate
- xii. Understanding Site Maps
- xiii. Understanding Annotation Orientations
- xiv. Creating Annotation Orientations

Module 3. Creating Annotation Elements

- i. Organizing Annotations into Combined States
 - ii. Understanding Semantic References
 - iii. Understanding Dimension Annotation Elements
 - iv. Planning for Dimension Annotations
 - v. Creating Driving Dimension Annotations
 - vi. Creating Driven Dimension Annotations
 - vii. Creating Ordinate Driven Dimension Annotations
 - viii. Understanding Syntax Checking
 - ix. Understanding Datum Feature Symbol Annotation Elements
 - x. Creating Datum Feature Symbols on Geometry
-

- xi. Creating Datum Feature Symbols in Dimensions and Gtols
- xii. Creating Geometric Tolerance Annotations
- xiii. Placing Gtol Datum Reference Frames
- xiv. Creating Surface Finish Annotations
- xv. Creating Symbol Annotations
- xvi. Creating Note Annotations
- xvii. Creating Hole Note Annotations from Driving Dimensions
- xviii. Creating Datum Target Annotations

Module 4. Modifying Annotation Elements

- i. Modifying Dimension Annotation Display
- ii. Manipulating Dimension Annotations
- iii. Setting Features with Asymmetric Tolerances to Mid-Spec
- iv. Modifying Dimension Annotation Extension Lines
- v. Modifying Note Annotations
- vi. Controlling Cross-Section Annotation Clipping
- vii. Using Security Markings
- viii. Modifying Dimension and Note Annotation Arrows
- ix. Creating Hyperlinks

Module 5. Completing Combination States

- i. Using Annotation Features with Annotation Elements
- ii. Creating Annotation Features
- iii. Adding Annotation Elements to Annotation Features
- iv. Adding Semantic References to Annotation Elements
- v. Creating and Associating Appearance States to Combined States
- vi. Adding Datum Geometry to Combined States
- vii. Associating Site Map Notes to Geometry
- viii. Reviewing Combination States

Module 6. Publishing for Technical Data Packages

- i. Understanding a Technical Data Package
 - ii. Printing Combination States
 - iii. Pre-Check List for Publishing
 - iv. Publishing to Creo View
 - v. Preparing Combination States for Publishing
 - vi. Analyzing the Publishing Process
 - vii. Understanding Creo View File Formats
 - viii. Understanding the Creo View User Interface
 - ix. Orienting the Model in Creo View
-

- x. Viewing Annotations in Creo View
- xi. Filtering Entities in the Viewing Area
- xii. Creating a STEP AP203 File from Creo Parametric
- xiii. Adding Additional Files to a Creo View File

Introduction to Creo Options Modeler 4.0

Overview

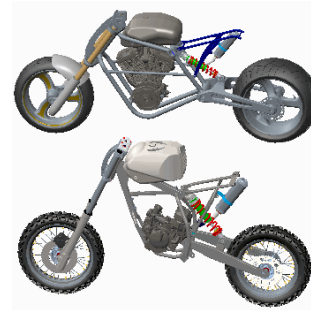
Course Code TRN-5113-T

Course Length 8 Hours

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.

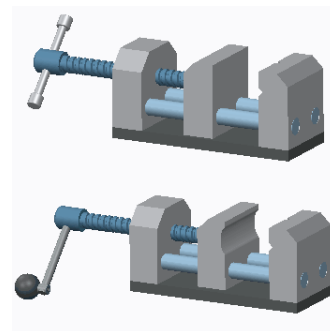
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in Pro/FICIENCY intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0 (F000).



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 4.0
- Advanced Assembly Design using Creo Parametric 4.0

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

Course Content

Module 1. Introduction to Creo Options Modeler

- i. Introduction to Options Modeler
- ii. Exploring a Configurable Assembly

Knowledge Check Questions

Module 2. Creating Interchange Assemblies

- i. Understanding Interchangeability
- ii. Replacing Components using Family Table
- iii. Understanding Interchange Assemblies
- iv. Replacing using a Functional Interchange Assembly

Knowledge Check Questions

Module 3. Creating Configurable Modules and Products

- i. Creating Configurable Modules and Products
- ii. Creating a Configurable Module from an Interchange Assembly
- iii. Adding Module Variants to Configurable Modules
- iv. Assembling with Configurable Products
- v. Creating Configurable Product Sub-Assemblies
- vi. Transferring Components into Configurable Modules
- vii. Utilizing Family Tables in Configurable Modules

Knowledge Check Questions

Module 4. Defining Options and Choices

- i. Defining Options
- ii. Defining Choices
- iii. Assigning Components to Choices

Knowledge Check Questions

Module 5. Utilizing Configurable Assemblies

- i. Understanding the Variant Builder
- ii. Utilizing the Variant Builder
- iii. Managing Configurations
- iv. Saving New Product Variants

Knowledge Check Questions

Module 6. The Options Modeler with Very Large Assemblies and Windchill

- i. The Options Modeler and Very Large Assemblies
- ii. Using the Options Modeler with Windchill

Knowledge Check Questions

Module 7. Project

- i. Designing Variations for the Power Head
 - ii. The Drill Variation
-

- iii. The Pump Variation
- iv. The Auger Variation

Introduction to Creo Direct 4.0

Overview

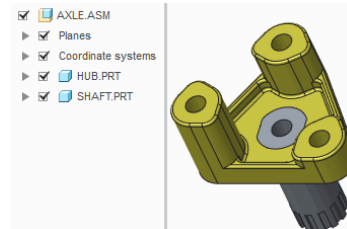
Course Code TRN-5110-T

Course Length 8 Hours

In this course, you will learn direct modeling using Creo Direct 4.0. 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.

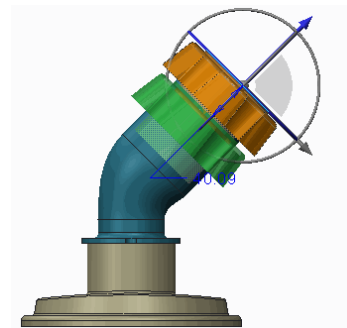
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Direct 4.0.



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 Mini 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 Guides
- iv. Sketching Lines
- v. Sketching Arcs
- vi. Sketching Rectangles and Parallelograms
- vii. Sketching Circles
- viii. Sketching Fillets
- ix. Sketching Chamfers
- x. Sketching Construction Geometry
- xi. Sketching Text
- xii. Sketching Ellipses
- xiii. Sketching Splines
- xiv. 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
- xii. Mirroring Selected Geometry
- xiii. Using the Geometry Search Tool

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
- vi. Modifying Part Geometry Using Boolean Operations
- vii. Using the Mirror Component Tool

Knowledge Check Questions

Introduction to Creo Layout 4.0

Overview

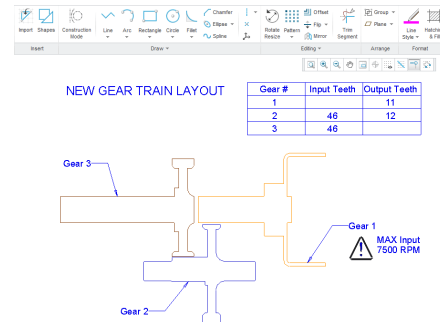
Course Code TRN-5112-T

Course Length 8 Hours

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.

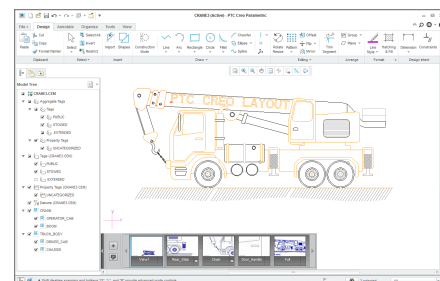
At the end of each module, you will complete a set of review questions to reinforce critical topics from that module. At the end of the course, you will complete a course assessment in PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Layout 4.0 M020.



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 4.0 (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.
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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

Course Content

Module 1. Introduction to Layout

- i. Exploring Layout Concepts
- ii. Understanding Layout File Types and Uses
- iii. Understanding the Layout Workflow
- iv. Exploring the Layout Interface
- v. Manipulating Layout Views
- vi. Selecting Geometry
- vii. Creating a Layout

Knowledge Check Questions

Module 2. Layout Sketching Methodology

- i. Sketching Using Cartesian and Polar Coordinates
- ii. Sketching Using Guides
- iii. Sketching Settings
- iv. Utilizing Layout Dimensions
- v. Utilizing Layout Constraints

Knowledge Check Questions

Module 3. Sketching Layout Geometry

- i. Sketching Lines
- ii. Sketching Arcs
- iii. Sketching Rectangles and Parallelograms
- iv. Sketching Circles
- v. Sketching Fillets
- vi. Sketching Chamfers
- vii. Sketching Construction Geometry
- viii. Sketching Text
- ix. Sketching Ellipses
- x. Sketching Centerlines and Center Points
- xi. Sketching Datum Geometry

Knowledge Check Questions

Module 4. Manipulating Layout Geometry

- i. Utilizing the Mirror and Flip Tools
- ii. Utilizing the Trim and Merge Tools
- iii. Patterning and Offsetting Layout Geometry
- iv. Translating, Rotating, and Scaling Layout Geometry
- v. Inserting 2-D Shapes
- vi. Measuring Layout Geometry
- vii. Utilizing 2-D Diagnostic Tools

Knowledge Check Questions

Module 5. Organizing Layout Geometry

- i. Exploring Geometry Color and Line Style
- ii. Grouping Layout Geometry
- iii. Creating Structures in Layout
- iv. Utilizing Geometry Tags
- v. Defining Property Tags
- vi. Understanding Sublayouts
- vii. Creating Sublayouts
- viii. Manipulating Sublayouts

Knowledge Check Questions

Module 6. Importing Layout Geometry

- i. Importing 2-D Geometry
- ii. Importing 3-D Cross-Sections
- iii. Importing Images

Knowledge Check Questions

Module 7. Annotating Layouts

- i. Creating Layout Symbols
- ii. Creating Layout Notes
- iii. Creating Layout Tables
- iv. Defining Layout Parameters
- v. Utilizing Hatching and Fill

Knowledge Check Questions

Module 8. Utilizing Layouts in 3-D Models

- i. Designating Public Geometry
- ii. Assembling Layouts
- iii. Utilizing Assembled Layouts in 3-D Models
- iv. Utilizing Layout Features in 3-D Models
- v. Utilizing Update Control for 3-D Layout Features

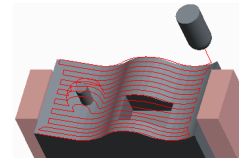
Knowledge Check Questions

Milling using Creo Parametric 4.0

Overview

Course Code TRN-5108-T

Course Length 40 Hours



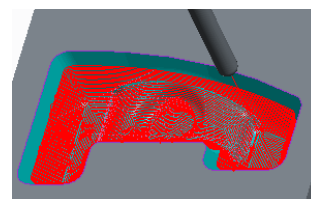
In this course, you will learn how to machine products using Creo Parametric manufacturing tools. This course covers creating tool paths for three axis milling machines. During the course, you will learn how to complete each phase of the manufacturing process. You will start by creating manufacturing models and configuring the manufacturing environment. This will include configuring tools, fixtures, and machining operations. You will then learn how to create milling sequences, holmaking 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 PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0 B000.

Course Objectives

- Understand the manufacturing process
- Create and configure manufacturing models
- Configure the manufacturing environment
- Create and modify milling sequences
- Create and modify holmaking 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
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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 Holmaking Sequences
Module	22	Creating Engraving Sequences
Module	23	Using the Process Manager
Module	24	Creating and Post-Processing CL Data Files

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
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- 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. Trajectory Milling
- vii. Trajectory Milling Multi-Step and Multi-Pass Parameters

Knowledge Check Questions

Module 21. Creating Holmaking Sequences

- i. Understanding Holmaking
- 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

Update to Creo Parametric 4.0 from Creo Parametric 3.0

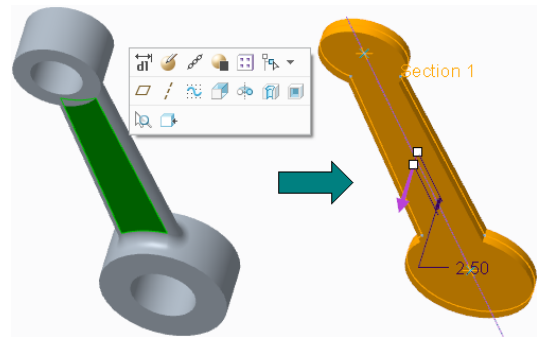
Overview

Course Code TRN-5100-T

Course Length 8 Hours

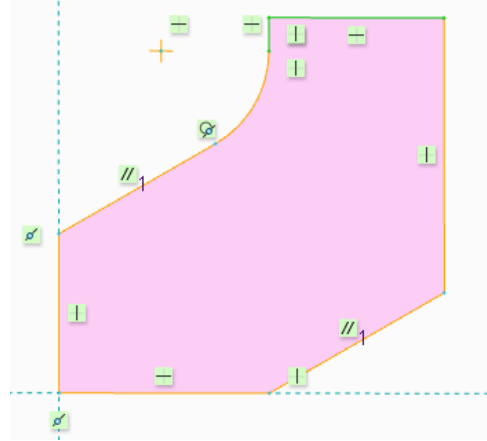
In this course, you will learn how to utilize the variety of functionality enhancements in Creo Parametric 4.0. You will be introduced to user interface enhancements such as the mini toolbar and the new geometry selection filter. You will examine the Part Modeling enhancements to features such as negative depth, extrude features with offset depth, and new midplane datum plane type. You will learn about the new Sketcher enhancements, including the clipping the model, customizing line thickness, the new constraint appearance, and using sketch references. You will investigate the new Assembly capabilities such as automatic representations, mirror symmetry, and assigning materials. You will examine the new Datum Feature Symbol, Datum Target, Geometric Tolerance, and Driven Dimension functionality and workflows for 2-D drawings and review various detailing enhancements. You will learn the new Freestyle surfacing enhancements including using multiple objects, importing OBJ files, and using N-Gon faces, as well as new Style enhancements such as G3 connections, creating periodic and nonperiodic closed curves, and drop curves. Finally, you will learn to use the updated tools in Sheetmetal mode such as Twist Walls, Edge Bend Relief, and Close Corner edge treatment, as well as the enhancement for utilizing dependency control with punch and die forms. 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 PTC University Proficiency intended to evaluate your understanding of the course as a whole.

This course has been developed using Creo Parametric 4.0



Course Objectives

- Use the Interface enhancements in Creo Parametric 4.0
- Use the Part Modeling enhancements in Creo Parametric 4.0
- Use the Sketcher enhancements in Creo Parametric 4.0
- Use the Assembly Modeling enhancements in Creo Parametric 4.0
- Use the Drawing enhancements in Creo Parametric 4.0
- Use the Surfacing enhancements in Creo Parametric 4.0
- Use the Sheetmetal enhancements in Creo Parametric 4.0



Prerequisites

- Introduction to Creo Parametric 3.0, or equivalent experience with 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.
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Agenda

Day 1

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

Course Content

Module 1. Interface Enhancements

- i. Interface Appearance Enhancements
- ii. Understanding Creo Parametric Basic Controls
- iii. Mini Toolbar Enhancement
- iv. Using the Geometry Selection Filter
- v. Customizing the User Interface
- vi. Toggle Full Screen Enhancement
- vii. Column Visibility Toggle Enhancement
- viii. Hidden Items Display Enhancement
- ix. Double-byte Characters Enhancement
- x. Locked Config.sup Options Enhancement
- xi. General Feature Location Enhancements
- xii. Using Real-Time Rendering
- xiii. Model View Dialog Box Enhancement
- xiv. Line Style Display Enhancement
- xv. Creating Appearance States
- xvi. Understanding and Identifying Failures
- xvii. Understanding the Notification Center
- xviii. Analyzing Geometry Failures

Knowledge Check Questions

Module 2. Part Modeling Enhancements

- i. Negative Depth Direction Enhancement
- ii. Creating Extrude Features with Offset Depth
- iii. Creating On Point Holes
- iv. Using the Top Clearance Option
- v. Datum Midplane Enhancement
- vi. Using the Exclude Areas with Draft Option
- vii. Group Enhancements
- viii. Accuracy Handling when Scaling or Changing Units

Knowledge Check Questions

Module 3. Sketcher Enhancements

- i. Clip Model Enhancement
- ii. Customizable Line Thickness Enhancement
- iii. Box Selection Enhancement
- iv. Sketching with On-the-Fly Constraints
- v. Sketching Lines
- vi. Sketching Text
- vii. Utilizing Sketch References

Knowledge Check Questions

Module 4. Assembly Enhancements

- i. Regeneration Status Enhancement
- ii. Using Automatic Representations
- iii. Creating Mirrored Assemblies
- iv. Creating Mirrored Components
- v. Creating Mirrored Sub-Assemblies
- vi. Assigning Materials
- vii. Creating Flexible Components with Varied Material
- viii. Outdated Mass Properties Enhancement
- ix. Managing Reference Backups
- x. Mechanism Motors as Features Enhancement
- xi. Initial Conditions from Playback of Other Analysis Enhancement
- xii. Creating Local Copy Geometry Features in Parts

*Knowledge Check Questions***Module 5. Drawing Enhancements**

- i. MBD Enhancements
- ii. Annotation Workflow Enhancement
- iii. Understanding Semantic References
- iv. Understanding Syntax Checking
- v. Creating Datum Feature Symbols
- vi. Creating Datum Targets
- vii. Applying Geometric Tolerances
- viii. Creating Driven Dimensions
- ix. Embedding Images in Drawings
- x. Replacing View Models
- xi. Non-Linear Cross-Hatching Enhancement
- xii. Text and Symbol Fonts Enhancement

*Knowledge Check Questions***Module 6. Surfacing Enhancements**

- i. Using Multiple Objects
- ii. Importing and Exporting OBJ Files into Freestyle
- iii. Using N-Gon Faces
- iv. Freestyle Usability Improvements
- v. Using G3 Connections
- vi. Creating Periodic and NonPeriodic Closed Curves
- vii. Using the Drop Curve Option
- viii. Preserving Analytic Geometry Enhancement

*Knowledge Check Questions***Module 7. Sheetmetal Enhancements**

- i. Creating Twist Wall Features
-

- ii. Utilizing Dependency Control with Punch and Die Forms
- iii. Creating Edge Bends
- iv. Flange Feature Close Corner Edge Treatment Enhancement
- v. Using Flexible Modeling in Sheetmetal Mode

Knowledge Check Questions
