Creo 3.0
Curriculum Guide
Live Classroom Curriculum Guide

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

Overview

Course Code: TRN-4500-T
Course Length: 1 Day

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

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

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

Prerequisites

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

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>1</th>
<th>Interface Enhancements</th>
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<td>Part Modeling Enhancements</td>
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<td>Drawing Enhancements</td>
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<tr>
<td>Module</td>
<td>8</td>
<td>Sheetmetal Enhancements</td>
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</table>
Course Content

Module 1. Interface Enhancements
   i. Interface Appearance Enhancements
   ii. File Save Enhancements
   iii. Active Window Enhancements
   iv. Graphics Enhancements
   v. Applying Realistic Appearances
   vi. Model Orientation Enhancement
   vii. Right-Click Menu Enhancements
   viii. Active Component Enhancements

Knowledge Check Questions

Module 2. Part Modeling Enhancements
   i. Draft Enhancements
   ii. Pattern Enhancements
   iii. UDF and Paste Special Enhancements
   iv. Collapse Feature Enhancements
   v. Feature Operation Enhancements
   vi. Creating Constant Width Rounds
   vii. Creating Spinal Bends
   viii. Creating Toroidal Bends
   ix. Editing Feature References
   x. Replacing Feature References

Knowledge Check Questions

Module 3. Surfacing Enhancements
   i. Boundary Blend Enhancements
   ii. Untrimming Surface Copies
   iii. Flattening Quilts
   iv. Analyzing Connections
   v. Previewing Style Features
   vi. Reusing Creo Sketch Curves
   vii. Creating Curves from Isolines
   viii. Joining Freestyle Geometry
   ix. Aligning Freestyle Geometry

Knowledge Check Questions

Module 4. Flexible Modeling Enhancements
   i. Managing Tangency
   ii. Attaching Moved Geometry
   iii. Recognizing Rounds and Chamfers
   iv. Editing Chamfers
v. Editing Non-Circular Rounds
vi. Creating Flexible Patterns
vii. Recognizing Patterns and Propagating Changes

Knowledge Check Questions

Module 5. Utilizing Intelligent Fasteners
i. Understanding the Intelligent Fastener Extension
ii. Assembling Intelligent Fasteners
iii. Manipulating Intelligent Fasteners
iv. Assembling Intelligent Fasteners Using Advanced Options
v. Manipulating Intelligent Fasteners Using Advanced Options

Knowledge Check Questions

Module 6. Utilizing Design Exploration
i. Understanding Design Exploration
ii. Exploring Part and Assembly Designs
iii. Creating Design Exploration Branches
iv. Opening and Saving Design Exploration Sessions
v. Using Design Exploration Options
vi. Utilizing Update Control with Copy Geometry Features

Knowledge Check Questions

Module 7. Drawing Enhancements
i. Drawing Standards Enhancements
ii. Printing Enhancements
iii. Dimension Creation Enhancements
iv. Note Creation Enhancements

Knowledge Check Questions

Module 8. Sheetmetal Enhancements
i. Rip Enhancements
ii. Flatten Form Enhancements
iii. Bending in Multiple Planes
iv. Creating Multiple Bend Reliefs
v. Bend Line Relief Placement
vi. Creating Die Forms
vii. Creating Die Forms Using Annotations

Knowledge Check Questions
Introduction to Creo Parametric 3.0

Overview

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<th>Course Code</th>
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<tr>
<td>Course Length</td>
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In this course, you will learn core modeling skills and quickly become proficient with Creo Parametric 3.0. Topics include sketching, part modeling, assemblies, drawings, and basic model management techniques. The course also includes a comprehensive design project that enables you to practice your new skills by creating realistic parts, assemblies, and drawings. After completing the course, you will be well prepared to work effectively on product design projects using Creo Parametric 3.0.

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

Course Objectives

- Learn the basic Creo Parametric modeling process
- Understand Creo Parametric concepts
- Learn how to use the Creo Parametric interface
- Select and edit geometry, features, and models
- Sketch geometry and use tools
- Create sketches for features
- Create datum planes and datum axes
- Create extrudes, revolves, and profile ribs
- Utilize internal sketches and embedded datums
- Create sweeps and blends
- Create holes, shells, and drafts
- Create rounds and chamfers
- Group, copy, and mirror items
- Create patterns
- Measure and inspect models
- Assemble with constraints
- Assemble with connections
- Explode assemblies
- Lay out drawings and create views
- Create drawing annotations
• Use layers
• Investigate parent/child relationships
• Capture and manage design intent
• Resolve failures and seek help
• Comprehensive two part Design Project

Prerequisites

• None

Audience

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

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<tr>
<td>Module</td>
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<td>Selecting Geometry, Features, and Models</td>
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<td>Creating Sketcher Geometry</td>
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<td>Creating Datum Features: Planes and Axes</td>
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<td>Creating Extrudes, Revolves, and Ribs</td>
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<td>Module</td>
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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. Selecting Items using Direct Selection
   vi. Selecting Items using Query Selection
   vii. Using the Search Tool
   viii. Using the Smart Selection Filter
   ix. Understanding Selection Filters

Knowledge Check Questions

Module 5. Editing Geometry, Features, and Models
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

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

Module 8. Creating Sketches for Features

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

Knowledge Check Questions
Module 9. Creating Datum Features: Planes and Axes
   i. Creating Datum Features Theory
   ii. Creating Datum Axes
   iii. Creating Datum Planes

Knowledge Check Questions

Module 10. Creating Extrudes, Revolves, and Ribs
   i. Creating Solid Extrude Features
   ii. Adding Taper to Extrude Features
   iii. Common Dashboard Options: Extrude Depth
   iv. Common Dashboard Options: Feature Direction
   v. Common Dashboard Options: Thicken Sketch
   vi. Creating Solid Revolve Features
   vii. Common Dashboard Options: Revolve Angle
   viii. Creating Profile Rib Features

Knowledge Check Questions

Module 11. Sketcher Workflow
   i. Analyzing Open and Closed Sections
   ii. Creating Internal Sketches
   iii. Analyzing Sketcher Workflow
   iv. Creating Embedded Datum Features

Knowledge Check Questions

Module 12. Creating Sweeps and Blends
   i. Creating Sweeps with Open Trajectories
   ii. Creating Sweeps with Closed Trajectories
   iii. Analyzing Sweep Feature Attributes
   iv. Creating Blends by Selecting Parallel Sections
   v. Creating Blends by Sketching Sections
   vi. Analyzing Blend Options

Knowledge Check Questions

Module 13. Creating Holes, Shells, and Draft
   i. Common Dashboard Options: Hole Depth
   ii. Creating Coaxial Holes
   iii. Creating Linear Holes
   iv. Creating Radial and Diameter Holes
   v. Exploring Hole Profile Options
   vi. Creating Shell Features
   vii. Creating Draft Features
   viii. Creating Basic Split Drafts

Knowledge Check Questions
Module 14. Creating Rounds and Chamfers
   i. Creating Rounds Theory
   ii. Creating Rounds by Selecting Edges
   iii. Creating Rounds by Selecting a Surface and Edge
   iv. Creating Rounds by Selecting Two Surfaces
   v. Creating Full Rounds
   vi. Creating Round Sets
   vii. Creating Chamfers by Selecting Edges
   viii. Analyzing Basic Chamfer Dimensioning Schemes
   ix. Creating Chamfer Sets

Knowledge Check Questions

Module 15. Project I
   i. The Air Circulator
   ii. Piston Assembly Components
   iii. Crankshaft, Engine Block, Impeller, and Impeller Housing
   iv. The Frame and Bolt

Module 16. Group, Copy, and Mirror Tools
   i. Creating Local Groups
   ii. Copying and Pasting Features
   iii. Moving and Rotating Copied Features
   iv. Mirroring Selected Features
   v. Mirroring All Features
   vi. Creating Mirrored Parts

Knowledge Check Questions

Module 17. Creating Patterns
   i. Direction Patterning in the First Direction
   ii. Direction Patterning in the Second Direction
   iii. Axis Patterning in the First Direction
   iv. Axis Patterning in the Second Direction
   v. Direction Patterning with Multiple Direction Types
   vi. Creating Reference Patterns of Features
   vii. Creating Reference Patterns of Components
   viii. Deleting Patterns or Pattern Members

Knowledge Check Questions

Module 18. Measuring and Inspecting Models
   i. Viewing and Editing Model Properties
   ii. Investigating Model Units
   iii. Analyzing Mass Properties
   iv. Using the Measure Tools
   v. Using the Measure Summary Tool
vi. Creating Planar Part Cross-Sections
vii. Measuring Global Interference

Knowledge Check Questions

Module 19. Assembling with Constraints
i. Understanding Assembly Theory
ii. Creating New Assembly Models
iii. Understanding Constraint Theory
iv. Understanding Assembly Constraint Status
v. Assembling Components using the Default Constraint
vi. Orienting Components
vii. Creating Coincident Constraints using Geometry
viii. Creating Coincident Constraints using Datum Features
ix. Creating Distance Constraints
x. Creating Parallel, Normal, and Angle Constraints
xi. Assembling using Automatic
xii. Utilizing the Accessory Window

Knowledge Check Questions

Module 20. Assembling with Connections
i. Understanding Connection Theory
ii. Dragging Connected Components
iii. Assembling Components using the Slider Connection
iv. Assembling Components using the Pin Connection
v. Assembling Components using the Cylinder Connection
vi. Analyzing Collision Detection Settings

Knowledge Check Questions

Module 21. Exploding Assemblies
i. Creating and Managing Explode States
ii. Creating Explode Lines
iii. Animating Explode States

Knowledge Check Questions

Module 22. Drawing Layout and Views
i. Analyzing Drawing Concepts and Theory
ii. Analyzing Basic 2-D Orientation
iii. Utilizing the Drawing Tree
iv. Creating New Drawings and Applying Formats
v. Creating and Orienting General Views
vi. Managing Drawing Sheets
vii. Adding Drawing Models
viii. Creating Projection Views
ix. Creating Cross-Section Views
x. Creating Detailed Views  
xi. Creating Auxiliary Views  
 xii. Creating Assembly and Exploded Views  
 xiii. Modifying Drawing Views  
 xiv. Creating New Drawings using Drawing Templates

Knowledge Check Questions

Module 23. Creating Drawing Annotations  
i. Analyzing Annotation Concepts and Types  
 ii. Creating Tables from File  
 iii. Creating BOM Balloons  
 iv. Showing, Erasing, and Deleting Annotations  
 v. Cleaning Up Dimensions  
 vi. Manipulating Dimensions  
 vii. Creating Driven Dimensions  
viii. Inserting Notes  
 ix. Analyzing Drawing Associativity  
x. Publishing Drawings

Knowledge Check Questions

Module 24. Using Layers  
i. Understanding Layers  
 ii. Creating and Managing Layers  
 iii. Utilizing Layers in Part Models  
 iv. Utilizing Layers in Assembly Models

Knowledge Check Questions

Module 25. Investigating Parent/Child Relationships  
i. Understanding Parent Child Relationships  
 ii. Viewing Part Parent Child Information  
 iii. Viewing Assembly Parent Child Information  
 iv. Viewing Model, Feature, and Component Information

Knowledge Check Questions

Module 26. Capturing and Managing Design Intent  
i. Handling Children of Deleted and Suppressed Items  
 ii. Reordering Features  
 iii. Inserting Features  
 iv. Redefining Features and Sketches  
v. Capturing Design Intent in Sketches  
 vi. Capturing Design Intent in Features  
 vii. Capturing Design Intent in Parts  
viii. Capturing Design Intent in Assemblies

Knowledge Check Questions
Module 27. Resolving Failures and Seeking Help

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

Knowledge Check Questions

Module 28. Project II

i. The Air Circulator
ii. Piston Assembly
iii. Engine Block and Drawing
iv. Blower Assembly
v. Engine Blower Assembly
vi. Completing the Design
Advanced Modeling using Creo Parametric 3.0

Overview

Course Code: TRN-4502-T
Course Length: 3 Days

The Advanced Modeling using Creo Parametric 3.0 training course teaches you how to use advanced part modeling techniques to improve your product designs. In this course, you will learn how to create and modify design models using advanced sketching techniques and feature creation tools. You will also learn how to reuse existing design geometry when creating new design models. After completing this course, you will be well prepared to work efficiently with complex product designs using Creo Parametric 3.0.

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

Course Objectives

- Learn advanced selection techniques
- Create advanced datum features
- Use advanced sketching techniques
- Create advanced holes
- Create advanced drafts and ribs
- Create advanced shells
- Create advanced rounds and chamfers
- Use relations and parameters
- Create advanced blends
- Create sweeps with variable sections
- Create helical sweeps
- Create swept blends
- Learn advanced layer techniques
- Learn advanced reference management techniques
- Create family tables
- Reuse features
- Learn advanced copy techniques
- Create advanced patterns

Prerequisites

- Introduction to Creo Parametric 3.0
Audience

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

## Day 1

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<tr>
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<th>1</th>
<th>Advanced Selection</th>
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<td>Advanced Datum Features</td>
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<tr>
<td>Module</td>
<td>3</td>
<td>Advanced Sketching</td>
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<tr>
<td>Module</td>
<td>4</td>
<td>Advanced Hole Creation</td>
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<td>Advanced Drafts and Ribs</td>
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<th>Relations and Parameters</th>
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<td>Sweeps with Variable Sections</td>
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<td>Helical Sweeps</td>
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<th>Advanced Layers</th>
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<tr>
<td>Module</td>
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<td>Advanced Reference Management</td>
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Module 1. Advanced Selection
   i. Advanced Chain Selection
   ii. Advanced Surface Selection
   iii. Using the Search Tool

Knowledge Check Questions

Module 2. Advanced Datum Features
   i. Creating Datum Graphs
   ii. Creating Datum Coordinate Systems
   iii. Creating Points On or Offset from Entities
   iv. Creating Points at Intersections
   v. Creating Points using an Offset Coordinate System
   vi. Sketching Geometry Datums
   vii. Creating Curves Through a Point or Vertex
   viii. Creating a Curve Through a Point Array
   ix. Creating a Curve from a Cross-Section
   x. Creating a Curve From Equation
   xi. Creating Composite Curves
   xii. Creating a Curve from Curve Intersections
   xiii. Creating a Curve at Surface Intersection
   xiv. Projecting and Wrapping Curves
   xv. Trimming Curves
   xvi. Creating Offset Curves
   xvii. Creating Cosmetic Sketches

Knowledge Check Questions

Module 3. Advanced Sketching
   i. Using Sketched Curves
   ii. Sketching Ellipses
   iii. Sketching Elliptical Fillets
   iv. Sketching Splines
   v. Modifying Splines — Basic Operations
   vi. Modifying Splines — Advanced Operations
   vii. Importing and Exporting Spline Points
   viii. Sketching Conics
   ix. Sketching Text
   x. Thickening Edges
   xi. Analyzing Sketcher Convert Options
   xii. Locking Sketcher Entities
   xiii. Analyzing Sketcher Dimension Options
   xiv. Sketcher Diagnostic Tools
Knowledge Check Questions

Module 4. Advanced Hole Creation
i. Creating Standard Holes
ii. Lightweight Hole Display
iii. Creating Sketched Holes
iv. Creating On Point Holes
v. Creating Cosmetic Threads

Knowledge Check Questions

Module 5. Advanced Drafts and Ribs
i. Drafting Intent Surfaces
ii. Analyzing Draft Hinges and Pull Direction
iii. Creating Drafts with Multiple Angles
iv. Using the Extend Intersect Surfaces Draft Option
v. Creating Drafts Split at Sketch
vi. Creating Drafts Split at Curve
vii. Creating Drafts Split at Surface
viii. Creating Drafts with Variable Pull Direction
ix. Creating Trajectory Ribs

Knowledge Check Questions

Module 6. Advanced Shells
i. Analyzing Shell References and Thickness Options
ii. Excluding Surfaces from Shells
iii. Extending Shell Surfaces
iv. Analyzing Shell Corner Options

Knowledge Check Questions

Module 7. Advanced Rounds and Chamfers
i. Analyzing Round Profile
ii. Analyzing Round Creation Methods
iii. Creating Rounds Through Curve
iv. Creating Variable Radius Rounds
v. Auto Round
vi. Creating Rounds by Reference
vii. Analyzing Round References and Pieces
viii. Using Intent Edges for Rounds
ix. Using Round Transitions
x. Creating Constant Width Rounds
xi. Analyzing Additional Chamfer Types
xii. Analyzing Advanced Chamfer Dimensioning Schemes
xiii. Analyzing Chamfer Creation Methods
xiv. Creating Corner Chamfers
xv. Creating Chamfers by Reference
xvi. Analyzing Chamfer References and Pieces
xvii. Using Intent Edges for Chamfers
xviii. Using Chamfer Transitions

Knowledge Check Questions

Module 8. Relations and Parameters
i. Understanding Relation Theory
ii. Understanding Relation Types
iii. Understanding Basic Relation Operators and Functions
iv. Understanding Advanced Relation Operators and Functions
v. Exact Relation
vi. Creating Parameters
vii. Understanding Advanced Parameter Options
viii. Creating Relations
ix. Creating Relations for Patterns
x. Creating Section Relations
xi. Using the Evalgraph Function
xii. Using Simultaneous Equations

Knowledge Check Questions

Module 9. Advanced Blends
i. Creating Blends by Selecting Non-Parallel Sections
ii. Analyzing Blend Section Tools
iii. Analyzing Blend Tangency
iv. Creating Rotational Blends by Selecting Sections
v. Creating Rotational Blends by Sketching Sections
vi. Analyzing Rotational Blend Options
vii. Analyzing Rotational Blend Tangency

Knowledge Check Questions

Module 10. Sweeps with Variable Sections
i. Understanding Sweeps with Variable Sections Theory
ii. Creating Sweeps using a Constant Section
iii. Creating Sweeps Normal to Trajectory
iv. Creating Sweeps Using Constant Normal Direction
v. Creating Sweeps with Variable Sections Normal to Projection
vi. Analyzing Horizontal and Vertical Control in Sweeps
vii. Creating Sweeps with Variable Sections Utilizing Multiple Trajectories
viii. Creating Sweeps with Variable Sections using Tangent Trajectories
ix. Analyzing Sweeps with Variable Sections Trajectory Options and Rules
x. Using Trajpar with Solid Features
xi. Using Trajpar and Datum Graphs with Solid Features
Knowledge Check Questions

Module 11. Helical Sweeps
   i. Understanding Helical Sweeps Theory
   ii. Creating Helical Sweeps for Springs
   iii. Creating Helical Sweeps for Threads
   iv. Analyzing Helical Sweep Profile and Pitch Variations
   v. Utilizing Variable Sections in Helical Sweeps

Knowledge Check Questions

Module 12. Swept Blends
   i. Understanding Swept Blend Theory
   ii. Creating Swept Blends by Selecting Sections
   iii. Creating Swept Blends by Sketching Sections
   iv. Analyzing Swept Blend Section Options
   v. Analyzing Swept Blend Section Plane Control
   vi. Analyzing Horizontal and Vertical Control in a Swept Blend
   vii. Analyzing Swept Blend Tangency
   viii. Analyzing Swept Blend Options
   ix. Analyzing Swept Blend Rules
   x. Creating Spinal Bends
   xi. Creating Toroidal Bends

Knowledge Check Questions

Module 13. Advanced Layers
   i. Understanding Layers
   ii. Creating and Managing Layers
   iii. Creating Layer States
   iv. Creating Layer Rules
   v. Creating Layers in Assemblies

Knowledge Check Questions

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

Knowledge Check Questions

Module 15. Family Tables
   i. Understanding Family Table Theory
   ii. Creating a Family Table
   iii. Patternizing Family Table Instances
   iv. Creating a Multi-Level Family Table
Module 16. Reusing Features

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

Knowledge Check Questions

Module 17. Advanced Copy

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

Knowledge Check Questions

Module 18. Advanced Patterns

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

Knowledge Check Questions
Advanced Assembly Design using Creo Parametric 3.0

Overview

<table>
<thead>
<tr>
<th>Course Code</th>
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<tbody>
<tr>
<td>Course Length</td>
<td>3 Days</td>
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In this course, you will learn how to use Creo Parametric 3.0 to create and manage complex assemblies. You will learn how to use advanced assembly tools that enable you to add and maintain designs, increase your efficiency, and increase system performance when working with large assemblies. In addition, you will learn the basics of using and creating predefined assembly structures and skeletons, which are both valuable tools typically used in a top-down design process. The course also includes an assembly design project that enables you to practice your new skills by performing various design tasks in an assembly model.

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

Course Objectives

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

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

Audience

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

## Day 1

<table>
<thead>
<tr>
<th>Module</th>
<th>Topic</th>
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<tbody>
<tr>
<td>1</td>
<td>Advanced Component Selection</td>
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<td>2</td>
<td>Using Advanced Assembly Constraints</td>
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<tr>
<td>3</td>
<td>Creating and Using Component Interfaces</td>
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<td>4</td>
<td>Utilizing Intelligent Fasteners</td>
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<tr>
<td>5</td>
<td>Creating and Using Flexible Components</td>
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<td>6</td>
<td>Restructuring and Mirroring Assemblies</td>
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## Day 2

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<thead>
<tr>
<th>Module</th>
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<tr>
<td>7</td>
<td>Using Assembly Features and Shrinkwrap</td>
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<td>8</td>
<td>Replacing Components in an Assembly</td>
</tr>
<tr>
<td>9</td>
<td>Understanding the Basics of Simplified Reps</td>
</tr>
<tr>
<td>10</td>
<td>Creating Cross-Sections, Display Styles, Layer States, and Combined Views</td>
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## Day 3

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<thead>
<tr>
<th>Module</th>
<th>Topic</th>
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<tr>
<td>11</td>
<td>Substituting Components using User Defined, Envelopes, and Simplified Reps</td>
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<tr>
<td>12</td>
<td>Understanding Advanced Simplified Rep Functionality</td>
</tr>
<tr>
<td>13</td>
<td>Creating and Using Assembly Structure and Skeletons</td>
</tr>
<tr>
<td>14</td>
<td>Utilizing Design Exploration</td>
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<tr>
<td>15</td>
<td>Project</td>
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Course Content

Module 1. Advanced Component Selection
   i. Locating Components in the Model Tree
   ii. Using the Assembly Model Tree Search Field
   iii. Selecting Multiple Components

Knowledge Check Questions

Module 2. Using Advanced Assembly Constraints
   i. Constraining Components using Fix
   ii. Constraining Two Coordinate Systems
   iii. Constraining a Point on a Line
   iv. Constraining a Point on a Surface
   v. Constraining an Edge on a Surface
   vi. Constraining a Point on a Point
   vii. Creating a Tangent Constraint
   viii. Configuring Constraint Sets with Parameters

Knowledge Check Questions

Module 3. Creating and Using Component Interfaces
   i. Understanding Component Interfaces
   ii. Using a Placing Component Interface
   iii. Using a Receiving Component Interface
   iv. Creating a Component Interface using the Save as Interface Dialog Box
   v. Auto Placing Components
   vi. Copying and Pasting Components
   vii. Repeating Component Placement

Knowledge Check Questions

Module 4. Utilizing Intelligent Fasteners
   i. Understanding the Intelligent Fastener Extension
   ii. Assembling Intelligent Fasteners
   iii. Manipulating Intelligent Fasteners
   iv. Assembling Intelligent Fasteners Using Advanced Options
   v. Manipulating Intelligent Fasteners Using Advanced Options

Knowledge Check Questions

Module 5. Creating and Using Flexible Components
   i. Adding Flexibility to a Component
   ii. Placing Flexible Components in an Assembly
   iii. Adding Flexibility to Already Placed Components
   iv. Using Flexible Parameters

Knowledge Check Questions

Module 6. Restructuring and Mirroring Assemblies
i. Restructuring and Reordering Assembly Components
ii. Creating Mirrored Assemblies
iii. Creating Mirrored Components
iv. Creating Mirrored Sub-Assemblies

Knowledge Check Questions

Module 7. Using Assembly Features and Shrinkwrap
i. Understanding Assembly Features
ii. Understanding Assembly Feature Intersections
iii. Creating an Assembly Cut
iv. Creating Assembly Holes
v. Creating a Shrinkwrap Feature
vi. Creating a Shrinkwrap Model
vii. Summarizing Shrinkwrap Features and Models

Knowledge Check Questions

Module 8. Replacing Components in an Assembly
i. Understanding Component Replace
ii. Replacing Components using Family Table
iii. Replacing Components using Reference Model
iv. Replacing Components using By Copy
v. Replacing Unrelated Components
vi. Understanding Interchange Assemblies
vii. Replacing using a Functional Interchange Assembly

Knowledge Check Questions

Module 9. Understanding the Basics of Simplified Reps
i. Retrieving Assembly Subsets
ii. Understanding Standard Simplified Reps
iii. Understanding Custom Simplified Reps
iv. Lightweight Graphics Representations
v. Using Graphics Simplified Reps
vi. Using Geometry Simplified Reps
vii. Excluding Components using Simplified Reps
viii. Defining Simplified Reps Using the Component Chooser
ix. Creating a Default Envelope Simplified Rep
x. Creating Part Simplified Reps
xi. Opening Simplified Reps

Knowledge Check Questions

Module 10. Creating Cross-Sections, Display Styles, Layer States, and Combined Views
i. Understanding Assembly Cross-Sections
ii. Creating Assembly Cross-Sections
iii. Creating Offset Assembly Cross-Sections
   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

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

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
Module 14. Utilizing Design Exploration

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

Knowledge Check Questions

Module 15. Project

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

Knowledge Check Questions
# Introduction to Creo Simulate 3.0

## Overview

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<thead>
<tr>
<th>Course Code</th>
<th>TRN-4504-T</th>
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<tbody>
<tr>
<td>Course Length</td>
<td>5 Days</td>
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In this course, you will learn how to test, validate, and optimize product designs with the Creo Simulate module. Creo Simulate enables you to simulate structural and thermal loads on product designs. You will complete comprehensive, hands-on lab exercises that simulate realistic analysis and design optimization activities. You will also be introduced to advanced topics such as dynamic analyses, combined mechanical and thermal analyses, and Optimization Studies. After completing the course, you will be able to run engineering analyses and optimizations on your product design models.

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

## Course Objectives

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

- Three months of Pro/ENGINEER Wildfire 5.0 or Creo Parametric experience

Audience

- This course is intended for design engineers and mechanical designers. People in related roles will also benefit from taking this course.
<table>
<thead>
<tr>
<th>Day 1</th>
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<tbody>
<tr>
<td>Module 1: Introduction to Creo Simulate</td>
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<tr>
<td>Module 2: Theoretical Foundations</td>
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<tr>
<td>Module 3: Simulation Models</td>
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<tr>
<td>Module 4: Materials and Material Properties</td>
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<td>Module 5: Structural Constraints</td>
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<td>Module 6: Structural Loads</td>
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<thead>
<tr>
<th>Day 2</th>
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<tbody>
<tr>
<td>Module 7: Meshing</td>
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<tr>
<td>Module 8: Convergence</td>
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<td>Module 9: Structural Analysis</td>
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<tr>
<td>Module 10: Introduction to Results Evaluation</td>
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<th>Day 3</th>
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<tr>
<td>Module 11: Refining the Design</td>
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<tr>
<td>Module 12: Basic Model Debugging</td>
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<td>Module 13: Singularities</td>
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<tr>
<td>Module 14: Analyzing Assemblies</td>
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<td>Module 15: Shells</td>
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<td>Module 16: Idealizations</td>
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<th>Day 5</th>
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<tbody>
<tr>
<td>Module 17: Thermal Analysis</td>
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<tr>
<td>Module 18: Advanced Analysis</td>
</tr>
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<td>Module 19: Project</td>
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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
Detailing using Creo Parametric 3.0

Overview

Course Code: TRN-4505-T
Course Length: 3 Days

In this course, you will learn how to quickly create detailed drawings using information captured within 3-D design models. You will also learn how to create drawings, how to detail drawings, and how to take advantage of the parametric and associative nature of Creo Parametric 3.0 when configuring drawings. After completing this course, you will be able to create production drawings suitable for manufacturing.

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

Course Objectives

- Understand the drawing development process
- Create new drawings using formats and drawing templates
- Create different types of views in drawings
- Create dimensions and notes
- Control display options using layers
- Apply dimensional and geometric tolerances in drawings
- Add draft geometry and symbols to drawings
- Use layers in drawings to control the display of views and detail items
- Create drawing tables and a bill of materials
- Create drawing formats
- Configure the drawing environment
- Manage large drawings
Prerequisites

- Introduction to Creo Parametric 3.0

Audience

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


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<thead>
<tr>
<th>Day 1</th>
<th>Module</th>
<th>1 Introduction to Drawings</th>
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<tr>
<td></td>
<td>Module</td>
<td>2 Creating New Drawings</td>
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<tr>
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<td>Module</td>
<td>3 Creating Drawing Views</td>
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<tr>
<th>Day 2</th>
<th>Module</th>
<th>4 Adding Model Details to Drawings</th>
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<td>Module</td>
<td>5 Adding Notes to Drawings</td>
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<td></td>
<td>Module</td>
<td>6 Adding Tolerance Information</td>
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<tr>
<td></td>
<td>Module</td>
<td>7 Adding Draft Geometry and Symbols</td>
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<th>Day 3</th>
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<th>8 Using Layers in Drawings</th>
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<td></td>
<td>Module</td>
<td>9 Creating and Using Tables in Drawings</td>
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<td>Module</td>
<td>10 Using Report Information in Drawings</td>
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<td>Module</td>
<td>11 Creating Drawing Formats</td>
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<td>Module</td>
<td>12 Configuring the Drawing Environment</td>
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<tr>
<td></td>
<td>Module</td>
<td>13 Managing Large Drawings</td>
</tr>
</tbody>
</table>
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

Module 2. Creating New Drawings
   i. Creating Drawings Using Formats and Sheets
   ii. Creating Drawing Templates
   iii. Creating Drawings Using Drawing Templates

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

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

Module 5. Adding Notes to Drawings
   i. Adding and Editing Notes
   ii. Using Parametric Information and Special Characters in Notes

Module 6. Adding Tolerance Information
Module 7. Adding Draft Geometry and Symbols
   i. Creating and Editing Draft Geometry
   ii. Understanding Drawing Symbols
   iii. Using Surface Finish Symbols
   iv. Using the Symbol Palette and Custom Symbols
   v. Creating Symbols
Knowledge Check Questions

Module 8. Using Layers in Drawings
   i. Understanding Layers in Drawings
   ii. Using Layers in Drawings
Knowledge Check Questions

Module 9. Creating and Using Tables in Drawings
   i. Inserting Tables
   ii. Editing Table Properties
   iii. Creating Tables from File
   iv. Creating Hole Tables
Knowledge Check Questions

Module 10. Using Report Information in Drawings
   i. Creating Report Tables
   ii. Editing Report Tables
   iii. Creating BOM Balloons
   iv. Creating Part Catalog Drawings
Knowledge Check Questions

Module 11. Creating Drawing Formats
   i. Creating Drawing Formats
Knowledge Check Questions

Module 12. Configuring the Drawing Environment
   i. Configuring the Drawing Environment
Knowledge Check Questions

Module 13. Managing Large Drawings
   i. Understanding Drawing Regeneration
   ii. Managing Large Drawings
Knowledge Check Questions
Surfacing using Creo Parametric 3.0

Overview

Course Code TRN-4506-T
Course Length 3 Days

In this course, you will learn how to use various techniques to create complex surfaces with tangent and curvature continuities. You will also learn how 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.

Course Objectives

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

• Introduction to Creo Parametric 3.0

Audience

• This course is intended for mechanical designers, design engineers, industrial designers, and related roles
# Agenda

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   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

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

Knowledge Check Questions

Module 5. Basic Surfacing Tools
i. Creating Surface Extrude Features
ii. Creating Surface Revolve Features
iii. Creating Fill Surfaces
iv. Creating Sweep Surfaces with Open Trajectories
v. Creating Sweep Surfaces with Closed Trajectories
vi. Creating Blend Surfaces by Selecting Parallel Sections
vii. Creating Blend Surfaces by Selecting Non-Parallel Sections
viii. Creating Blend Surfaces by Sketching Sections
ix. Analyzing Blend Surface Section Tools
x. Analyzing Blend Surface Options
xi. Analyzing Blend Surface Tangency
xii. Creating Rotational Blend Surfaces by Selecting Sections
xiii. Creating Rotational Blend Surfaces by Sketching Sections
xiv. Analyzing Rotational Blend Surface Options
xv. Analyzing Rotational Blend Surface Tangency

Knowledge Check Questions

Module 6. Boundary Blend Surfaces
i. Understanding Boundary Curve Concepts
ii. Creating Boundary Blends in One Direction
iii. Creating Boundary Blends in Two Directions
iv. Analyzing Blended Surface Boundary Conditions
v. Analyzing Blended Surface Constraint Options
vi. Analyzing Blended Surface Control Points
vii. Creating Boundary Blends with Influencing Curves
viii. Analyzing Approximate Blended Surface Options
ix. Creating a Blend Tangent to Surfaces

Knowledge Check Questions

Module 7. Sweep Surfaces with Variable Sections
i. Understanding Sweeps with Variable Sections
ii. Creating Sweep Surfaces using a Constant Section
iii. Creating Sweep Surfaces using Normal to Trajectory
iv. Creating Sweep Surfaces using Constant Normal Direction
v. Creating Sweep Surfaces using Normal to Projection
vi. Analyzing Horizontal and Vertical Control in a Sweep Surface
vii. Creating Sweep Surfaces Utilizing Multiple Trajectories
viii. Creating Sweep Surfaces using Tangent Trajectories
ix. Analyzing Sweep Surface Trajectory Options and Rules
x. Using Trajpar with Sweep Surface Features
xi. Using Trajpar and Datum Graphs with Sweep Surface Features

Knowledge Check Questions

Module 8. Helical Sweeps
i. Understanding Helical Sweeps Theory
ii. Utilizing Helical Sweeps for Surfaces
iii. Analyzing Helical Sweep Surface Profile and Pitch Variations
iv. Utilizing Variable Sections in Helical Sweep Surfaces

Knowledge Check Questions

Module 9. Swept Blends
i. Understanding Swept Blend Theory
ii. Creating Swept Blend Surfaces by Selecting Sections
iii. Creating Swept Blend Surfaces by Sketching Sections
iv. Analyzing Swept Blend Surface Section Options
v. Analyzing Swept Blend Surface Section Plane Control
vi. Analyzing Horizontal and Vertical Control in a Swept Blend Surface
vii. Analyzing Swept Blend Surface Tangency
viii. Analyzing Swept Blend Surface Options
ix. Analyzing Swept Blend Rules

Knowledge Check Questions

Module 10. Analyzing Surface Curvature
i. Analyzing Surfaces Theory
ii. Defining Curvature
iii. Defining Curvature Continuity
iv. Analyzing Curvature of Curves
v. Analyzing Curvature of Surfaces
vi. Analyzing Curvature using Sections
vii. Analyzing Curvature using Normals
viii. Using Shaded Curvature Analysis for Surfaces
ix. Using Shaded Section Curvature Analysis
x. Creating Curvature Continuous Surfaces
xi. Analyzing Connections

Knowledge Check Questions
Module 11. Additional Surface Analysis Tools
   i. Using the Point Analysis Option
   ii. Using the Radius Analysis Option
   iii. Using the Dihedral Angle Analysis Option
   iv. Using the Offset Analysis Option
   v. Using the Draft Analysis Option
   vi. Using the Slope Analysis Option
   vii. Using the Reflection Analysis Option
   viii. Using the Shadow Analysis Option

Knowledge Check Questions

Module 12. Extending and Trimming Surfaces
   i. Extending Surfaces
   ii. Extending Surfaces Using Measurements
   iii. Analyzing Extend Surface Options
   iv. Creating a Surface Trim
   v. Trimming Surfaces with Geometry
   vi. Trimming Surfaces with Quilts Options
   vii. Trimming Surfaces with the Silhouette Trim Option
   viii. Trimming Surfaces with the Vertex Round Option

Knowledge Check Questions

Module 13. Manipulating Surfaces
   i. Copying and Pasting Surfaces
   ii. Offsetting Surfaces
   iii. Offsetting Surfaces with the Expand Option
   iv. Offsetting Surfaces with Draft
   v. Moving and Rotating Quilts
   vi. Mirroring Quilts
   vii. Merging Surfaces
   viii. Untrimming Surface Copies
   ix. Flattening Quilts

Knowledge Check Questions

Module 14. Creating and Editing Solids using Quilts
   i. Thickening Surface Quilts
   ii. Solidifying Quilts to Add Material
   iii. Solidifying Quilts to Remove Material
   iv. Solidifying Quilts to Replace Material
   v. Offsetting Surfaces using the Replace Option
   vi. Creating Rounds on Surfaces
   vii. Converting Solid Rounds to Surfaces

Knowledge Check Questions
Module 15. Master Model Technique
  i. Master Model Technique Theory
  ii. Creating a Master Model
  iii. Creating Framework in the Master
  iv. Creating Surfaces in the Master
  v. Refining and Completing the Master Model
  vi. Sharing Geometry from the Master Model
  vii. Completing Body Components

Knowledge Check Questions

Module 16. Project
  i. The Shaver
  ii. Creating the Master Model
  iii. Creating Framework in the Master Model
  iv. Creating Surfaces in the Master Model
  v. Refining and Completing the Master Model
  vi. Sharing Geometry from the Master Model
  vii. Creating a Body Component
Sheetmetal using Creo Parametric 3.0

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In this course, you will learn how to create sheetmetal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 3.0 and serves as the second stage of learning. In this course, you will learn how to design sheetmetal parts and assemblies, including sheetmetal production drawings. All the functions needed to create sheetmetal parts, drawings, and assemblies are covered. Upon completion of this course, you will be able to create sheetmetal design models, create the flat state of the model, and document both in production drawings.

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

Course Objectives

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

- Introduction to Creo Parametric 3.0

Audience

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

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Course Content

Module 1. Introduction to the Creo Parametric Sheetmetal Design Process
   i. Creo Parametric Sheetmetal Design Process

Knowledge Check Questions

Module 2. Sheetmetal Model Fundamentals
   i. Sheetmetal Model Fundamentals
   ii. Understanding Developed Length
   iii. Creating a New Sheetmetal Part in Assembly Mode
   iv. Creating a New Sheetmetal Model in Part Mode
   v. Converting Solid Models to Sheetmetal

Knowledge Check Questions

Module 3. Creating Primary Sheetmetal Wall Features
   i. Understanding Sheetmetal Wall Features
   ii. Creating Planar Walls
   iii. Extruded Sheetmetal Wall Features
   iv. Revolved Sheetmetal Wall Features
   v. Blend Sheetmetal Wall Features
   vi. Creating Offset Walls
   vii. Sheetmetal Wall Sketching Tools
   viii. Advanced Primary Walls

Knowledge Check Questions

Module 4. Creating Secondary Sheetmetal Wall Features
   i. Understanding Secondary Walls
   ii. Creating Secondary Flat Walls
   iii. Using Flange Walls
   iv. Using Extruded Walls
   v. Wall Dashboard Options
   vi. Using Partial and Overextended Walls
   vii. Understanding Relief
   viii. Creating Twist Wall Features
   ix. Extending and Trimming Walls
   x. Using the Merge Feature

Knowledge Check Questions

Module 5. Bending and Unbending Sheetmetal Models
   i. Creating Bend Features
   ii. Adding Transition to Bends
   iii. Bending in Multiple Planes
   iv. Creating Planar Bends
   v. Creating Unbend Features
Module 6. Sheetmetal Form Features
i. Punch Form Features
ii. Utilizing Punch Model Annotations
iii. Creating Die Forms
iv. Creating Die Forms Using Annotations
v. Creating Sketched Forms
vi. Flattening Forms and Unstamping Edges

Knowledge Check Questions

Module 7. Modifying Sheetmetal Models
i. Sheetmetal Cuts
ii. Notches and Punches
iii. Creating Multiple Bend Reliefs
iv. Bend Line Relief Placement
v. Creating Corner Relief
vi. Creating Rip Features
vii. Creating Edge Bends
viii. Joining Walls
ix. Patterning Walls
x. Mirroring Walls

Knowledge Check Questions

Module 8. Sheetmetal Setup and Tools
i. Bend Line Adjustments
ii. Using Bend Tables for Bend Allowances
iii. Fixed Geometry
iv. Info Tools and Reports
v. Design Rules
vi. Defaults and Parameters
vii. Using Conversion Features

Knowledge Check Questions

Module 9. Detailing Sheetmetal Designs
i. Adding the Flat and Formed States
ii. Auto Ordinate Dimensions
iii. Bend Line Notes
iv. Bend Order Tables

Knowledge Check Questions
Module 10. Design Project
   i. Designing a Stapler
Milling using Creo Parametric 3.0

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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, hoolmaking sequences, and post-process cutter location (CL) data to create machine code. After completing the course, you will be able to create numerical control (NC) programs for milling machines and post-process cutter location (CL) data to create machine specific code.

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

Course Objectives

- Understand the manufacturing process
- Create and configure manufacturing models
- Configure the manufacturing environment
- Create and modify milling sequences
- Create and modify hoolmaking 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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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
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   ii. Roughing Scans and Entry and Exit Parameters

Knowledge Check Questions
Module 19. Creating Finishing Sequences
   i. Basic Finishing
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Module 20. Creating Trajectory Milling Sequences
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   ii. Creating Sketched Milling Tools
   iii. Basic 2-Axis Trajectory Milling
   iv. 2-Axis Trajectory Milling Depth Control Parameters
   v. 2-Axis Trajectory Milling - Cutting Slices Parameters
   vi. Basic 3-Axis Trajectory Milling
   vii. 3-Axis Trajectory Milling Multi-Step and Multi-Pass Parameters

Module 21. Creating Holemaking Sequences
   i. Understanding Holemaking
   ii. Basic Drilling
   iii. Editing Drilling Toolpaths
   iv. Creating and Using Drill Groups

Module 22. Creating Engraving Sequences
   i. Engraving on Flat and Complex Surfaces

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

Module 24. Creating and Post-Processing CL Data Files
   i. Creating and Post-Processing CL Data Files
Introduction to Creo Direct 3.0

Overview

Course Code: TRN-4510-T
Course Length: 1 Day

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

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

Course Objectives

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

- Introduction to Creo Parametric or equivalent experience

Audience

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

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Course Content

Module 1. Introduction to Creo Direct
   i. Understanding Direct Modeling
   ii. Understanding the User Interface
   iii. Understanding the Live Toolbar
   iv. Opening and Creating a Creo Direct Model
   v. Orienting and Positioning the Model
   vi. Understanding Selection Behavior
   vii. Using Marquee Selection
   viii. Understanding Datum and Sketch Display

Knowledge Check Questions

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

Knowledge Check Questions

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

Knowledge Check Questions

Module 4. Creating Datums
   i. Creating Datums Theory
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

Module 6. Measurements and Sections
   i. Using the Measure Tools
   ii. Creating Part Cross-Sections
   iii. Creating Assembly Cross-Sections

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

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
Introduction to Creo Illustrate 3.0

Overview

Course Code: TRN-4415-T
Course Length: 1 Day

In this course, you will learn about Creo Illustrate and its role as a purpose-built, role-based solution for creating 3-D technical illustrations. You will learn how to dynamically create technical illustrations from existing 3-D CAD data. You will also learn how to map existing eBOM data to populate an illustration-specific sBOM. In addition, you will learn how to manipulate imported 3-D viewables to create service information content including service procedures, parts identification, training materials, and product assembly and disassembly. Finally, you will learn how to create markup and annotations in figures and animations.

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

Course Objectives

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

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

Audience

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

### Day 1

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Course Content

Module 1. Introduction to Creo Illustrate
   i. Introducing Creo Illustrate
   ii. The Creo Illustrate Process
   iii. Understanding the Creo Illustrate User Interface
   iv. The Creo Illustrate Figure Viewing Area
   v. The Creo Illustrate Ribbon
   vi. The Creo Illustrate Ribbon Home Tab
   vii. The Creo Illustrate File Menu
   viii. The Creo Illustrate Quick Access Toolbar
   ix. The Creo Illustrate Primary Panel
   x. The Creo Illustrate Upper Data Panel
   xi. The Creo Illustrate Lower Data Panel
   xii. Creo Illustrate Structure Tree
   xiii. The Creo Illustrate Status Bar
   xiv. Starting Creo Illustrate
   xv. Working with Creo Illustrate

Knowledge Check Questions

Module 2. Create an Illustration and Work with Figures
   i. Import 3-D MCAD Data
   ii. Working with Figures
   iii. Page Setup
   iv. sBOM Structure versus Displayed Parts and Sub-Assemblies
   v. Figure Orientation
   vi. Selecting Parts and Sub-Assemblies
   vii. Find Parts and Sub-Assemblies
   viii. Changing the Figure Display
   ix. Figure Rendering Options
   x. Enhanced Lighting
   xi. Hide and Unhide Parts and Sub-Assemblies
   xii. Inset Views
   xiii. Filters

Knowledge Check Questions

Module 3. Exploding Assemblies
   i. Explode Figures
   ii. Transform Exploded Figures
   iii. Free Rotation Exploded Figures
   iv. Restore Part and Sub-Assembly Locations
   v. Create Explode Lines
   vi. Smart Explode
vii. Working with Smart Explode Selection
viii. Setting the Smart Explode Direction
ix. Change Smart Explode Options
x. Save and Exit Smart Explode
xi. Section an Assembly
xii. Orient the Sectioning Plane
xiii. Advanced Sectioning Features
xiv. Create Quarter Cut Sections
xv. Cap the Section
xvi. Set Specific Parts to be Sectioned

Knowledge Check Questions

Module 4. Authoring Sequenced Process Steps
i. The Sequencer Process
ii. Creating Sequence Steps
iii. Editing Sequence Steps
iv. Adding Tagged Symbols and Callouts
v. Reordering Steps within the Sequence

Knowledge Check Questions

Module 5. Creo Illustrate Animations
i. Figure Animations
ii. Animation Control Tools
iii. Playback Controls
iv. Tracks and Keys
v. Timeline Control Tools
vi. Recording Control Tools
vii. Creating an Animation
viii. Using Animation Effects
ix. Editing Tracks
x. Editing Keys
xi. Easing Movement
xii. Previewing the Animation Path
xiii. Exporting a Figure Animation as a Movie

Knowledge Check Questions

Module 6. Create and Manage Annotations, Sub-Assemblies, and Parts Lists
i. Annotating Figures
ii. Customizing Tooltips
iii. Notes and Callout Annotations
iv. Leader Line Annotations
v. Stamp Annotations
vi. The Symbols Library
vii. Measurement Tools
viii. Assemble Parts
ix. Automatically Generate an Item List
x. Structure Edit Mode
xi. Display Parts in the Structure Edit Viewing Area
xii. Advanced sBOM Editing Tools
xiii. Investigating Creo Illustrate Options

Knowledge Check Questions

Module 7. Publishing and Exporting 3-D Illustrations

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

Knowledge Check Questions
Behavioral Modeling Using Creo Parametric 3.0

Overview

Course Code: TRN-4520-T
Course Length: 1 Day

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

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

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

Course Objectives

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

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

Audience

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

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Course Content

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

Knowledge Check Questions

Module 2. Creating Measurement Features on Creo Parametric Models
   i. Using the Measure Tools
   ii. Using the Measure Summary Tool
   iii. Creating a Measurement Feature

Knowledge Check Questions

Module 3. Creating Model Property Features on Creo Parametric Models
   i. Comparing Model Property Analyses
   ii. Measuring Mass Properties
   iii. Measuring X-Section Mass Properties
   iv. Measuring Pairs Clearance

Knowledge Check Questions

Module 4. Creating Analysis Features on Creo Parametric Models
   i. Comparing Analysis Features
   ii. Creating a Relation Analysis Feature
   iii. Creating a Motion Analysis Feature
   iv. Creating a Creo Simulate Analysis Feature
   v. Creating an MS Excel Analysis Feature
   vi. Creating an External Analysis Feature
   vii. Monitoring the Parameters of Analysis Features
   viii. Statistical Design Study

Knowledge Check Questions

Module 5. Creating User-Defined Analysis Features on Creo Parametric Models
   i. Introduction to User-Defined Analysis Features
   ii. Creating Field Points
   iii. Creating a Construction Group
   iv. Creating User-Defined Analysis Features

Knowledge Check Questions

Module 6. Conducting Design Studies and Optimizing Models
   i. Comparing Design Studies
   ii. Translating Design Specifications
   iii. Performing Sensitivity Analysis
   iv. Performing Feasibility Design Studies
   v. Performing Optimization Design Studies
Knowledge Check Questions

Module 7. Project
Web Based Curriculum Guide

- Introduction to Creo Illustrate 3.0
Overview

In this course, you will learn about Creo Illustrate and its role as a purpose-built, role-based solution for creating 3-D technical illustrations. You will learn how to dynamically create technical illustrations from existing 3-D CAD data. You will also learn how to map existing eBOM data to populate an illustration-specific sBOM. In addition, you will learn how to manipulate imported 3-D viewables to create service information content including service procedures, parts identification, training materials, and product assembly and disassembly. Finally, you will learn how to create markup and annotations in figures and animations.

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